

# eta/DYNAFORM User's Manual

Version 5.6



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

Phone: +1 (248) 729 3010  
Support: +1 (800) 382 3362  
Fax: +1 (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 .....	10
1.6 MOUSE FUNCTIONS .....	13
1.7 SPECIFICATIONS .....	14
1.8 GEOMETRY DATA .....	14
1.9 RECOMMENDED NAMING CONVENTION .....	15
1.10 DIALOG BOXES .....	15
1.11 PROPERTY TABLES .....	16
1.12 CONFIGURATION FILE .....	17
<b>GETTING STARTED .....</b>	<b>20</b>
2.1 GETTING STARTED .....	20
2.2 STARTING WITH CAD DATA .....	20
2.3 GETTING STARTED WITH MODEL DATA (NASTRAN, LS-DYNA, etc.) ..	21
2.4 GETTING STARTED FROM SCRATCH .....	21
2.5 LOCAL COORDINATE SYSTEM .....	22
<b>FILE MANAGER .....</b>	<b>25</b>
3.1 NEW .....	25
3.2 OPEN (CTRL+O) .....	25
3.3 RESTART .....	26
3.4 SAVE .....	27
3.5 SAVE AS .....	27
3.6 IMPORT (F3) .....	27
3.7 EXPORT (SHIFT+F3) .....	28
3.8 SUBMIT DYNA FROM INPUT DECK .....	29
3.9 RUN DYNA RESTART .....	30
3.10 PRINT SETUP .....	31
3.11 PRINT .....	34
3.12 EXIT (CTRL+X) .....	34
<b>PART CONTROL .....</b>	<b>35</b>

4.1	CREATE PART (CTRL+P)	36
4.2	EDIT PART	37
4.3	DELETE PART	38
4.4	ADD... TO PART	39
4.5	TURN ON	45
4.6	CURRENT PART	45
4.7	SEPARATE PART	46
4.8	TRANSPARENT	46
4.9	SUMMARY	46
<b>PREPROCESS</b>		<b>48</b>
5.1	LINE/POINT (CTRL+L)	49
5.2	SURFACE (CTRL+S)	80
5.3	ELEMENT (CTRL+E)	102
5.4	NODE (CTRL+N)	127
5.5	MODEL CHECK & REPAIR (CTRL + R)	135
5.6	BOUNDARY CONDITIONS (CTRL+U)	143
5.7	NODE/ELEMENT SET (CTRL+V)	153
<b>DFE (DIE FACE ENGINEERING)</b>		<b>155</b>
6.1	PREPARATION	156
6.2	BINDER	197
6.3	ADDENDUM DESIGN	231
6.4	RE-ENGINEERING	275
6.5	MODIFICATION	283
6.6	DIE DESIGN CHECK	300
<b>BSE (BLANK SIZE ENGINEERING)</b>		<b>302</b>
7.1	PREPARATION	303
7.2	MSTEP	305
7.3	UNFOLD	308
7.4	DEVELOPMENT	309
<b>SETUP</b>		<b>333</b>
8.1	QUICK SETUP (QS)	333
8.2	AUTO SETUP	357
<b>SPRINGBACK COMPENSATION PROCESS</b>		<b>463</b>
9.1	COMPENSATION	465
9.2	RESULT CHECK	468
9.3	TOPOLOGY REPAIR	469
9.4	MORPHING	470

9.5 SURFACE MAPPING .....	471
<b>TOOL DEFINITION.....</b>	<b>474</b>
10.1 ANALYSIS SETUP .....	475
10.2 DEFINE TOOLS (CTRL+T).....	477
10.3 POSITION TOOLS.....	486
10.4 DRAWBEAD (CTRL+D).....	488
10.5 BLANK GENERATOR.....	507
10.6 GUIDE & PIN.....	510
10.7 DEFINE BLANK (CTRL+B).....	515
10.8 BLANK OPERATION .....	516
10.9 MATERIAL .....	521
10.10 PROPERTY .....	525
10.11 ANIMATE.....	526
10.12 TOOLS ON/OFF.....	527
10.13 SUMMARY .....	528
<b>OPTION MENU.....</b>	<b>530</b>
11.1 MESH CONTROL .....	531
11.2 FILE OPTION.....	533
11.3 SYSTEM LANGUAGE .....	535
11.4 MATERIAL LIBRARY STANDARD .....	535
11.5 LEFT MENU .....	536
11.6 SHOW ICON TIPS (toggle) .....	537
<b>UTILITIES.....</b>	<b>538</b>
12.1 ANGLE BETWEEN LINES .....	539
12.2 DISTANCE BETWEEN POINTS/NODES .....	539
12.3 RADIUS THROUGH 3PTS/3NDS .....	540
12.4 AREA OF SELECTED ELEMENTS .....	540
12.5 LENGTH OF LINE.....	540
12.6 SECTION CUT.....	541
12.7 DRAW ARROW .....	541
12.8 DEFINE TITLE .....	542
12.9 IDENTIFY NODE/POINT .....	542
12.10 IDENTIFY ELEMENT.....	542
12.11 FIND ELEMENT.....	543
12.12 FIND NODE .....	543
12.13 LOAD CURVE .....	543
12.14 SHOW LINE.....	558
12.15 COORDINATE SYSTEM.....	558
12.16 DATABASE STATISTIC.....	564

**VIEW OPTIONS.....566**

13.1 COLOR MAP ..... 566

13.2 ROTATION..... 568

13.3 LIGHT ..... 569

13.4 BACKGROUND COLOR..... 570

13.5 DEFINE ROTATION CENTER ..... 571

13.6 USER VIEW ..... 572

13.7 TRUE VIEW..... 573

13.8 EXPLODED VIEW ..... 573

13.9 LABEL NODES (toggle) ..... 574

13.10 LABEL ELEMENTS (toggle) ..... 574

13.11 PLATE NORMAL (Color)..... 574

13.12 SHADING MODE ..... 574

13.13 SHADING QUALITY ..... 575

13.14 SCALE ACTIVE WINDOW ..... 575

**ANALYSIS .....576**

14.1 LS-DYNA (CTRL+A)..... 577

14.2 LS-DYNA (SPRINGBACK ) ..... 586

14.3 LS-DYNA (GRAVITY LOADING) ..... 587

14.4 MSTEP ..... 587

**APPENDIX A .....588**

**APPENDIX B .....589**

**APPENDIX C .....590**

**APPENDIX D .....591**

**FINAL NOTES .....592**

# INTRODUCTION

*eta/DYNAFORM Version 5.6* 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 971 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 5.6 include:**

### **A process oriented module- AUTOSETUP**

This module has been greatly improved in the eta/DYNAFORM 5.6. The multi stage simulation setup provides a complete scheme for solving stamping problems and allows the user to set up line die simulation. This module is process oriented and enables user to easily set up all types of simulation.

AUTOSETUP allows the user to define stamping direction, forming simulation of tailor welded blanks, hydroforming simulation, multi stage simulation and trimming operation. In addition, Superplastic forming simulation and thermal forming simulation are added to this module. The enhanced material library in Auto Setup module allows the user to add the standard material library for corporations to the program interface for easier selection of common used materials.

### **SCP module**

Springback Compensation Process (SCP) is a new module in eta/DYNAFORM 5.6 for die compensation of sprung sheet metal parts. Springback is one of the crucial issues for sheet stamping industry. The springback compensation based on finite element technology is utilized to alleviate the severity of springback phenomena. SCP is used to modify the original die face with the springback

quantity generated from springback simulation to make the part dimension to fall within the tolerance of intended part design after springback. The eta/DYNAFORM also provides some practical tools for the user to repair the compensated die face and improve the efficiency for die face design.

### **Update Drawbead function**

The Drawbead function is updated in eta/DYNAFORM 5.6. The GUI of drawbead and operation style is overhauled to facilitate easy drawbead definition and modification. All operation in drawbead generation, including property definition, create, edit and restrain force definition, can be easily performed without advanced training.

### **More functional DFE module**

Compared to previous version, many functions in the DFE module of eta/DYNAFORM 5.6 are improved and added to satisfy the needs of the friendly and functional operation. The user can quickly import the product and complete the entire process of die face design. The major improvement includes:

- Added SIDE-STEP function. This function enables user to extend part boundary and provide side-step features for the subsequent addendum creation.
- Added DOUBLE ATTACH function. The user can design double attached die face by two part geometry. This function can be used with SYMMETRY function.
- Improved TIPPING function. The wall angle and wall angle contour can be displayed enabling the user to view the variation tendency of wall angle.
- Binder module has been completely overhauled in eta/DYNAFORM 5.6. Many new functions are added to this module to satisfy the user's design requirements for various binder shapes. The user can easily modify the desired binder shape by adding control lines. In addition, the user can also design the flange binder and butterfly shape on binder.
- Improved addendum module. In this module, many functions are integrated to simplify the editing and modification process. The algorithm of addendum generation has also been improved and optimized for the user to design more reasonable addendum.
- Improved RE-ENGINEERING module. The GUI of this module is enhanced. Some auxiliary functions are added for the user to convert the original addendum to the new addendum in eta/DYNAFORM. Additional functions are provided to modify the new addendum.

### **Enhanced BSE module**

In eta/DYNAFORM 5.6, trimline module is a brand new module utilized to compute trimline using the modified one-step solver developed by ETA. In addition, the user can perform multiple localized unfold operation using the Multi Unfold module in BSE. The blank nesting capabilities have also been improved and optimized. For nesting of complicated parts, the nesting result yields higher material utilization rate and more practical nesting pattern.

### **Implement Rotary Draw Bending**

Rotary Draw Bending is a newly developed Quick Setup module for tube bending analysis. The module is equipped with user-friendly and process oriented GUI. Furthermore, the module enables users to easily tube bending model.

## Enhanced Pre-processor functions

- Rearranged the functions for line/point, surface, element and node with the sequence of create, delete, modify/edit, transform and display.
- Added Shortest Curve function, which allows the user to calculate the shortest curve between two points in element and the total curve length exceeding two points.
- Improved Boundary Line function, which allows the user to create line along the boundary of finite element mesh or surface boundary.
- Added S-Line function to Line/Point menu.
- Improved Section Line function.
- Improved Modify Line function, which allows the user to modify lines by adding, moving, deleting points on the line.
- Added new transformation function. Integrated the Translate, Rotate, Mirror and Scale functions for line, surface, element and node. Added LCS2LCS function for the user to perform transformation operation on line, surface, element and node.
- Improved line RE-SPACE function.
- Added Mesh to Surface function, which allows the user to generate surface from mesh.
- Improved Surface Split function.
- Added Extend Surface function, which allows the user to extend the untrimmed surface of outer boundary.
- Added Check Surface function, which allows the user to check tiny surface, surface boundary, control points distribution and surface intersection.
- Improved Surface Mesh function.
- Improved Drag Mesh function.
- Added Trim Element function, which allows the user to trim elements with one or multiple closed or unclosed trim lines.
- Added Lance Element function, which allows the user to lance the selected element along one or multiple specified paths.
- Added Mesh Surface Deviation function in Model Check/Repair, which allows the user to measure the deviation between surface and element.
- Improved index file for the user to transfer information such as FLC and drawbeads from Preprocessor to Post-processor.
- Improved Animate function in Tool menu.
- Improved Identify Node/Point function, which allows the user to select multiple nodes/points and auto display the numbers of the selected nodes/points on the screen.
- Added LCSLCS ENT TRANSFORM function in Coordinate System, which allows the user to copy or move lines, surfaces, elements and nodes as well as their thickness, stress, strain information from one LCS to another.
- Improved Light function View Options menu.
- Added Background Color function in View Options menu, which allows the user to change the background color.
- Added #25 and #26 shell element formula, Thick shell and Solid element formula.
- Added Coarsening and Interface Force options in Forming Analysis.

## 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.

### **Support LS-DYNA 971 (7600 version)**

The input file created by eta/DYNAFORM 5.6 supports LS-DYNA 971 (7600 version) format. The eta/DYNAFORM User's Manual describes functions of the software interface. For further details about solver, please refer to the LS-DYNA User's Manual published by the LSTC.

## Chapter 1

# STRUCTURAL OVERVIEW

## 1.1 GENERAL

eta/DYNAFORM Version 5.6 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 1.2.1 eta/DYNAFORM menu

As illustrated in Figure 1.2.1, 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
SCP	Springback compensation analysis enables the parts after springback to meet the design requirements of final products
TOOLS	Provides functions to create, define, and modify tooling and blank definitions
OPTION MENU	Contain 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 1.3.1.



Figure 1.3.1 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**

Update 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 obtain the 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 a 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**

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

**LEFT VIEW**

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

**RIGHT VIEW**

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

**ISOMETRIC VIEW**

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

**SURFACE MESH**

Create auto-generated mesh on the selected surface.

**BLANK GENERATOR**

Mesh the flat blank via boundary lines and curves and simultaneously generate the initial tubular blank during Tubular Hydroforming or bending forming.

**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 1.4.1, 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. **MESSAGE PROMPT AREA**  
Display the eta/DYNAFORM comments and messages.

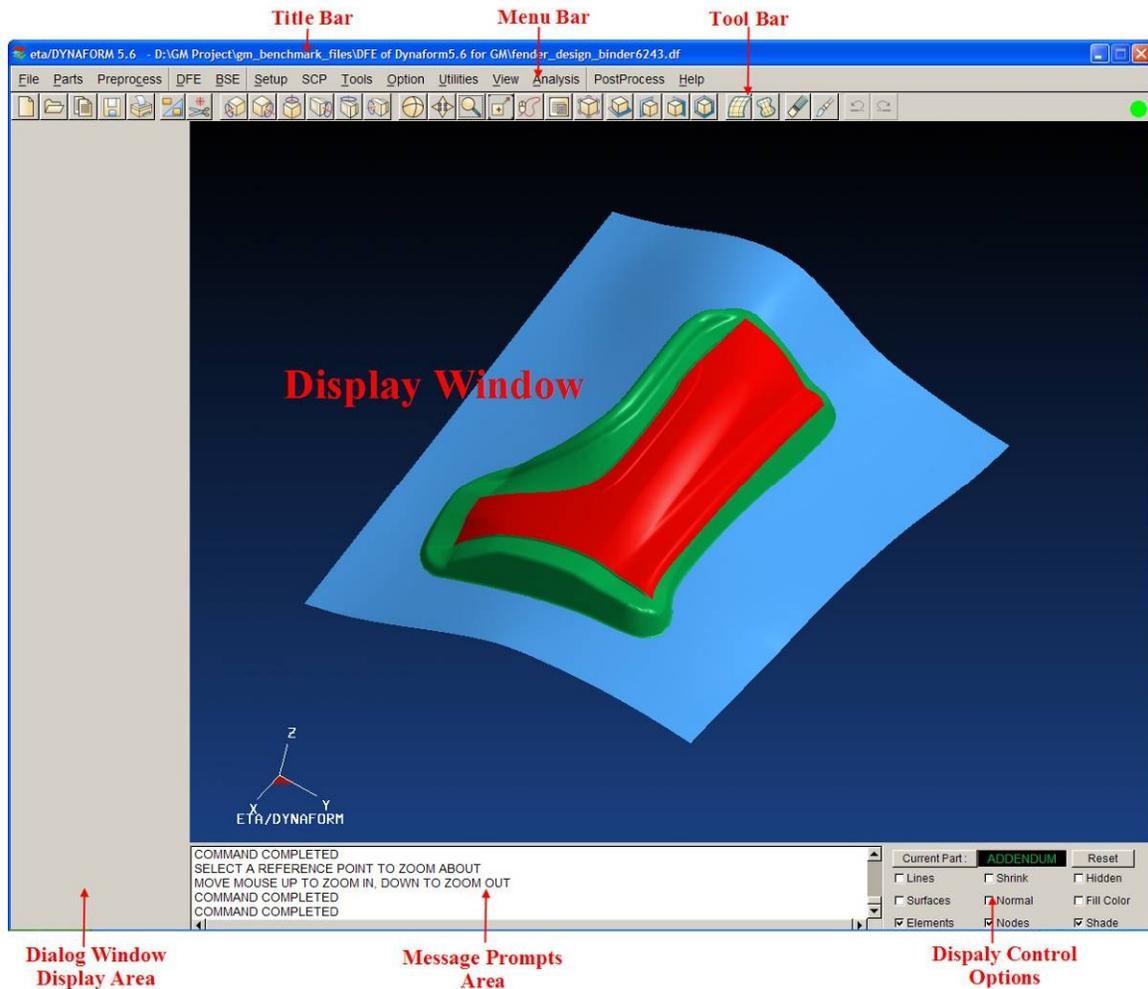


Figure 1.4.1 Interface of eta/DYNAFORM

## 1.5 DISPLAY OPTIONS

The DISPLAY OPTIONS window, as illustrated in Figure 1.5.1, is located at the bottom right corner of the display screen. It displays the current part and contains the following commonly used functions.

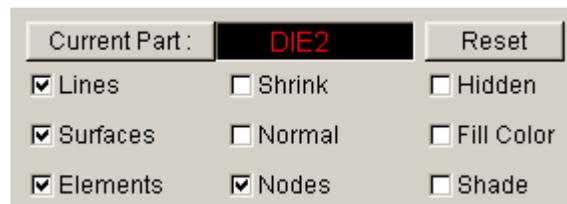


Figure 1.5.1 Display Options

### 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 **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 1.5.2. 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.

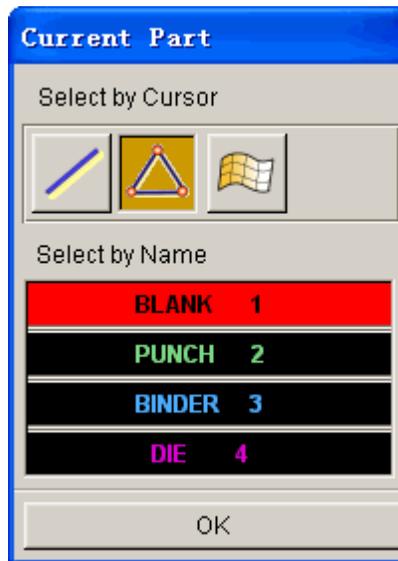


Figure 1.5.2 Current part dialog box

#### **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 1.5.3, SHRINK ELEMENTS is useful for locating any missing and/or degenerated elements in a shell or solid structure.

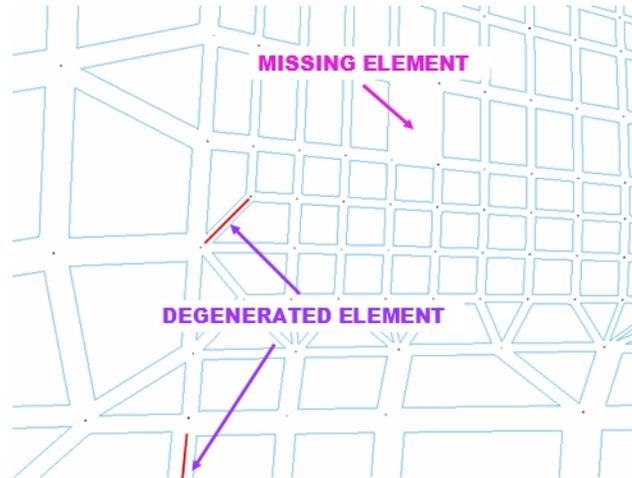


Figure 1.5.3 Shrink Elements

**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.

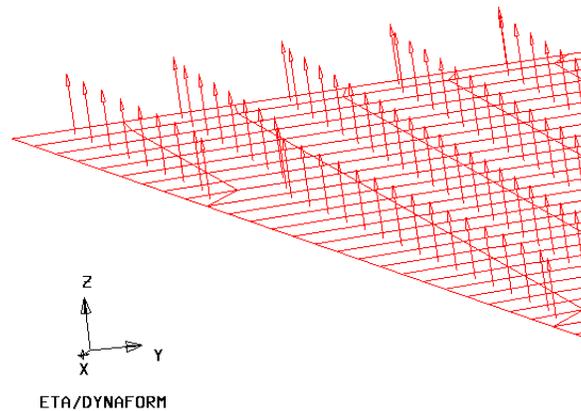


Figure 1.5.4 Elements Normal

If the user toggles on the PLATE NORMAL (COLOR) in the VIEW menu, the direction of positive element normal will be displayed using part color and gray color for negative normal direction. See Figure 1.5.5.

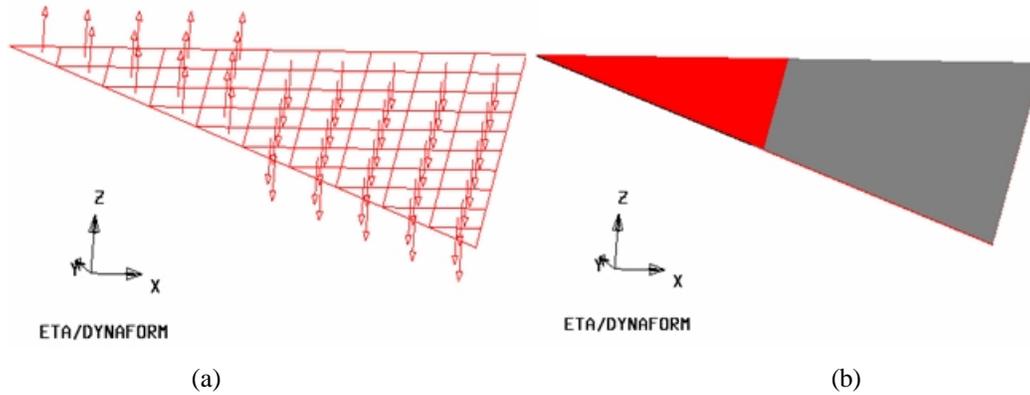


Figure 1.5.5 Display of elements normal direction (a) with arrows (b) with colors

### **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 or elements. The right mouse button is used to cancel some function. These three buttons can also be used with CTRL, SHIFT and ALT keys respectively for rotation, translation and scaling. For complicated models, eta/DYNAFORM provides the Feature Line mode and point cloud mode to accelerate the model display.

Control key	Left button	Middle button	Right button
Ctrl	Rotation	Translation	Scaling

<b>Ctrl+Alt</b>	Rotation (Point cloud)	Translation (Point cloud)	Scaling (Point cloud)
<b>Ctrl+Shift</b>	Rotation (Feature line)	Translation(Feature line)	Scaling(Feature line)

## 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
32,000	SURFACES
900,000	EDGE POINTS (SURFACE)
2,400,000	CONTROL POINTS (SURFACE)
1,000,000	ELEMENTS
1,000,000	NODES
1,000	PROPERTIES
1,000	MATERIALS
1,000	PIDS
500	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>
AutoCAD 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</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>	Abort the current function and exit the dialog box.
<b>APPLY</b>	Execute the current function without leaving the dialog box.
<b>BACK</b>	Return the user to the previous dialog box.

<b>CANCEL</b>	Reject the current operation.
<b>CLOSE</b>	Close the current dialog box.
<b>DONE</b>	Finish the current step in the dialog box and allow the user to proceed to the next step.
<b>EXIT</b>	Exit the current dialog box.
<b>OK</b>	Accept the data in the dialog box and forward the user to the next step.
<b>UNDO</b>	Reject the last step of the operation.
<b>REJECT</b>	Reject 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 1.11.1 enables the user to edit material properties such as K, N, R00, R45, R90, etc.

The screenshot shows a 'Material' dialog box with the following fields and values:

- Material Type: 36
- Title: DQSK
- FLD Curve: 0
- Card1 (selected):
 

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

Buttons at the bottom: Default, Reset, Ok, Cancel.

Figure 1.11.1 Material Type 36 Window

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

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

**OK** – Accept the currently displayed values and exit the table.

**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. All the parameters are shown in deck. At the bottom of the table the DEFAULT, RESET, OK, and CANCEL buttons allow the user to accept or reject the data.

## 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” (home) 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:

```
#DYNAFORM configure file
#[keywords] value

#[FILE_DIALOG_STYLE]: 0 - WINDOWS style, 1 - UNIX STYLE
#[SYSTEM_LANGUAGE]: ENGLISH,CHINESE,JAPANESE,KOREAN and so on
#[AUTOBACKUPTIME]: <=0 - don't autobackup, >0 - backup time( minutes )
#[AUTORESTART]: 0 - don't autorestart, 1 - autorestart.

[FILE_DIALOG_STYLE]
0

[SYSTEM_LANGUAGE]
English

[MATERIAL_LIB_TYPE]
```

```
0
[FADE_BACKGROUND]
1
[BACKGROUND_TOP_COLOR]
0,0,0
[BACKGROUND_BOTTOM_COLOR]
24,64,128
[LAST_CHECK_UPDATE]
March 23, 2007
[LAST_DATABASE_PATH]
D:/DF5.6/
[LAST_IMPORT_PATH]
D:/DF5.6/
[LS-DYNA_SOLVER_S]
C:\PROGRA~1\DYNAFO~1.6\lsdyna.exe
[LS-DYNA_SOLVER_D]
C:\PROGRA~1\DYNAFO~1.6\lsdyna_d.exe
[ETA-POSTPROCESS]
C:\PROGRA~1\DYNAFO~1.6\etapostprocessor.exe
[ETA-VPGDSA]

[AUTOBACKUPFILE]
timed.df
[AUTOBACKUPTIME]
0
[AUTORESTART]
0
[LAST_DBFILE_NAME]
fender.df
[MATERIAL_PATH]
C:\Documents and Settings\andy\My Documents\
```

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

**[MATERIAL\_LIB\_TYPE]** Define the default material library. eta/DYNAFORM provides four different standard material libraries.

**[FADE\_BACKGROUND]** Determine whether to show the fade background for graphic display area or not.

**[BACKGROUND\_TOP\_COLOR]** When displaying the Fade Background, set a color for the top of graphic display area. If only a unique background is displayed, set a background color for the entire graphic display area. The background color is in RGB format.

**[BACKGROUND\_BOTTOM\_COLOR]** When displaying the Fade Background, set a color for the bottom of graphic display area. If only a uniform background is displayed, this value is not used.

**[LAST\_CHECK\_UPDATE]** Record the last version update and check time.

**[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 automatically identify the last import or export database path.

**[LS-DYNA\_SOLVER\_S]** When submitting to LS-DYNA for calculation directly after setting, eta/DYNAFORM will search for the Single Precision Solver from this path.

**[LS-DYNA\_SOLVER\_D]** When submitting to LS-DYNA for calculation directly after setup, eta/DYNAFORM will search for the Double Precision Solver from this path.

**[ETA-POSTPROCESS]** When switching from Pre-processor to Post-processor, eta/DYNAFORM will search for the Post-processor from this path.

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

**[AUTOBACKUPTIME]** Define the time interval between automatic backup files.

**[AUTORESTART]** Post-process 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.

**[LAST\_DBFILE\_NAME]** Record the last opened database file.

**[MATERIAL\_PATH]** Specify the User Define Material path.

## Chapter 2

# GETTING STARTED

eta/DYNAFORM is organized into three major modules, a formability simulation module (FS), blank size engineering module (BSE), 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 data (IGES, VDA, UG, CATIA or any other geometry data) that will be translated and read directly into eta/DYNAFORM database.
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”, “df56”, 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, UG, CATIA 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 1.9 for 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 2.5, 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 5.3, 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 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.

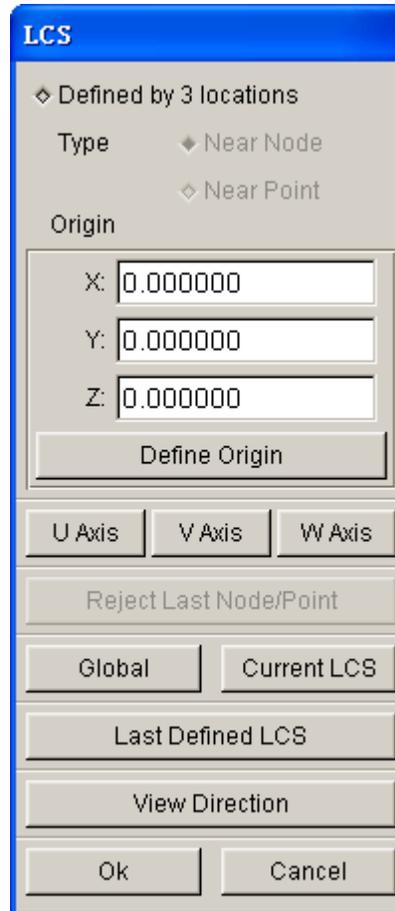


Figure 2.5.1 LCS dialog box

**GLOBAL**

Define the global coordinate system as the current LCS.

**CURRENT LCS**

Create a local coordinate system as the current LCS.

**LAST**

Define the last coordinate system 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.

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.

- **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.

- **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.

- **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 x 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.

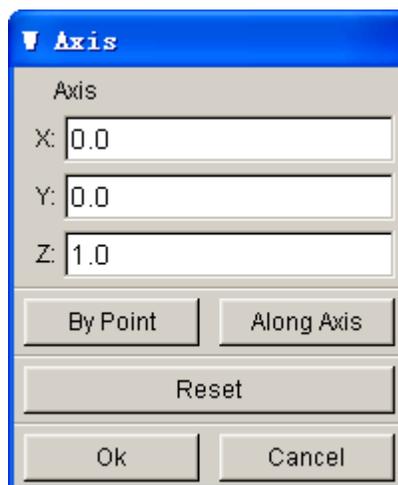


Figure 2.5.2 Define W Axis

## Chapter 3

# FILE MANAGER

The File Manager enables the 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.

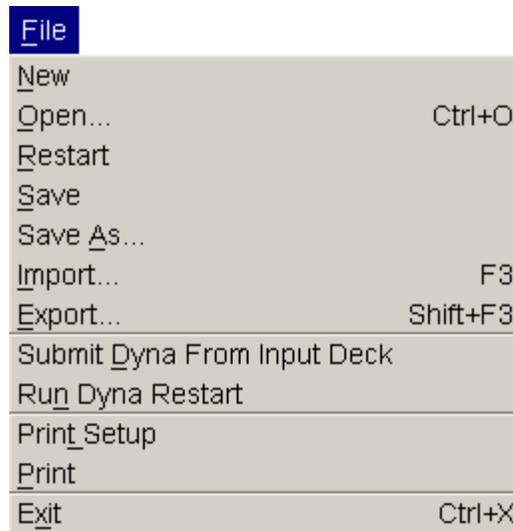


Figure 3.1 File Manager

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.

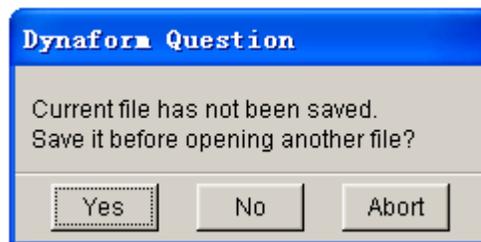


Figure 3.1.1 New File Prompt

- 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.

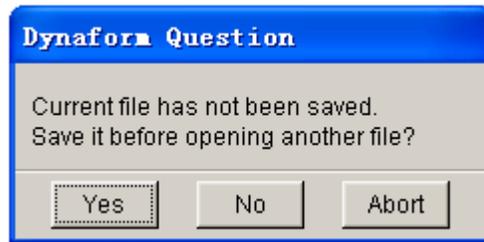


Figure 3.2.1 Save File Prompt

- 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 3.2.2.

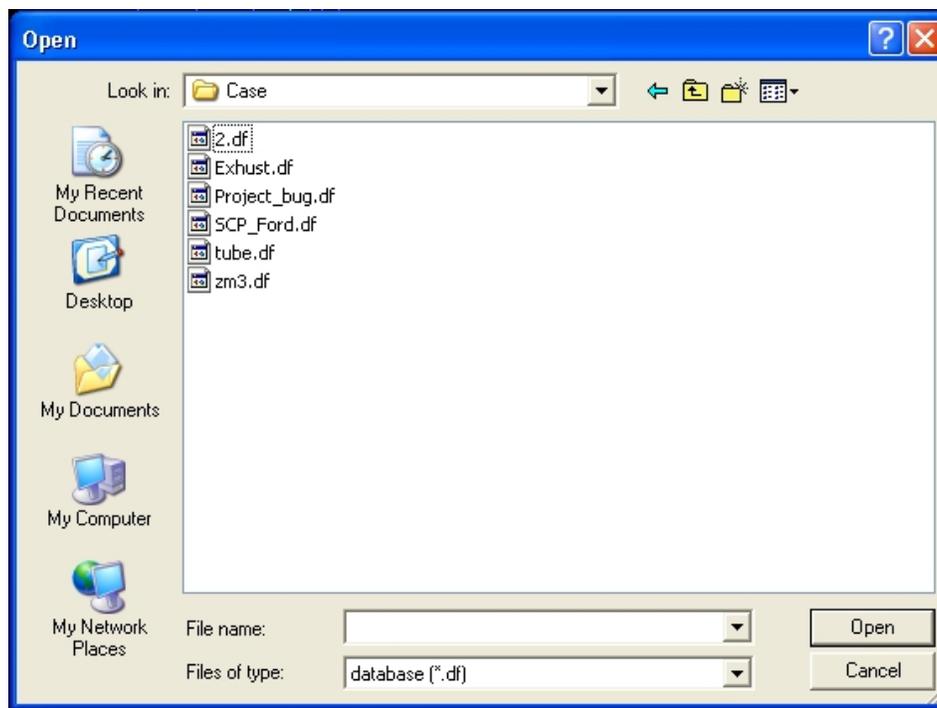


Figure 3.2.2 Open File window

### 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 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.

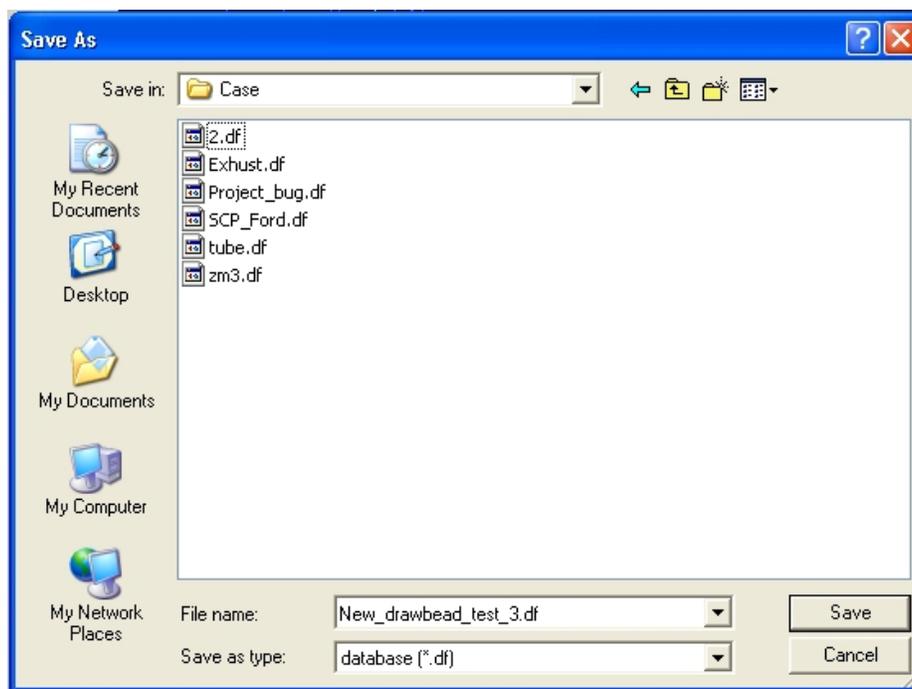


Figure 3.5.1 Save File As window

### 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.

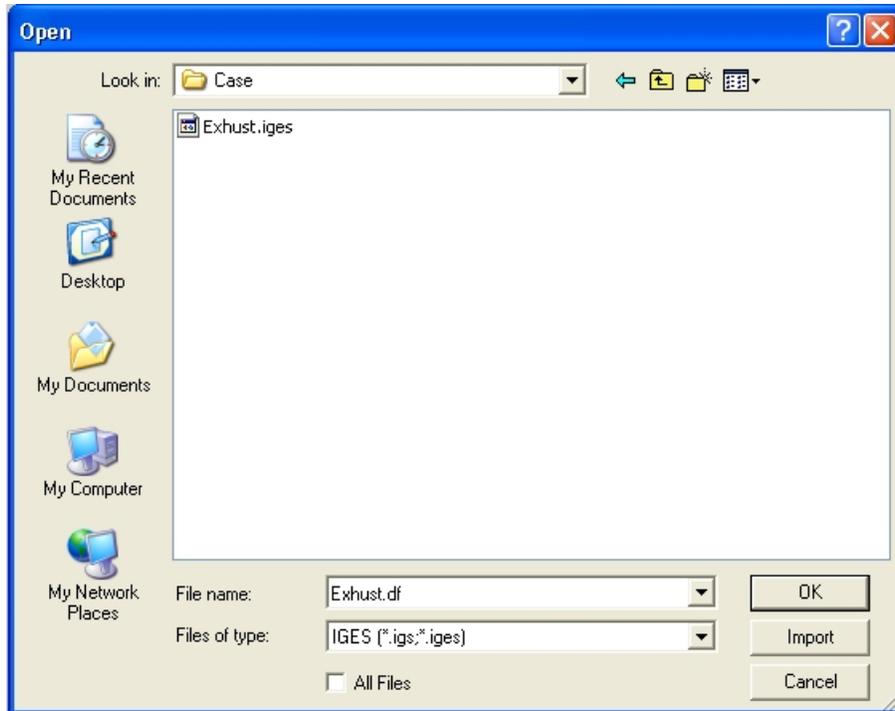


Figure 3.6.1 Import File window

**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

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:

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

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

**NASTRAN**

Entire or partial model data.

**INDEX**

Include 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.

**DYNAIN**

MULTISTAGE FILE. This allows the user to output the modified DYNAIN file as a new DYNAIN.

**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.

2. 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.

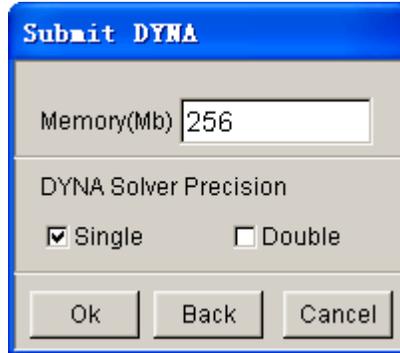


Figure 3.8.2 Define Memory dialog window

3. After the desired memory and solver type is setup, the user clicks OK to submit the previously selected calculation file directly to LS-DYNA solver. If the user does not define the solver in ANALYSIS SETUP dialog box, the program will pop up a Select Solver dialog box. Once the solver is selected, the program will submit the calculation file to LS-DYNA solver for calculation.

## 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.

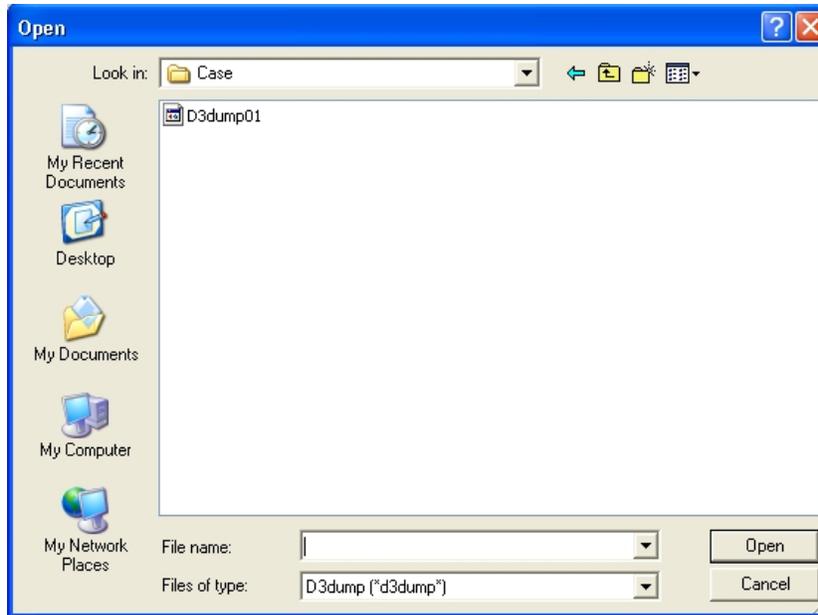


Figure 3.9.1 Select DYNA Restart File window

**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.

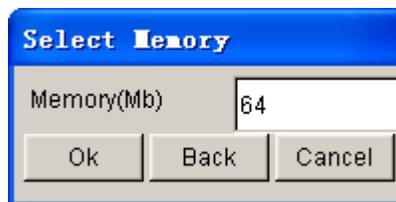


Figure 3.9.2 Define Memory

**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 13.1, 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.

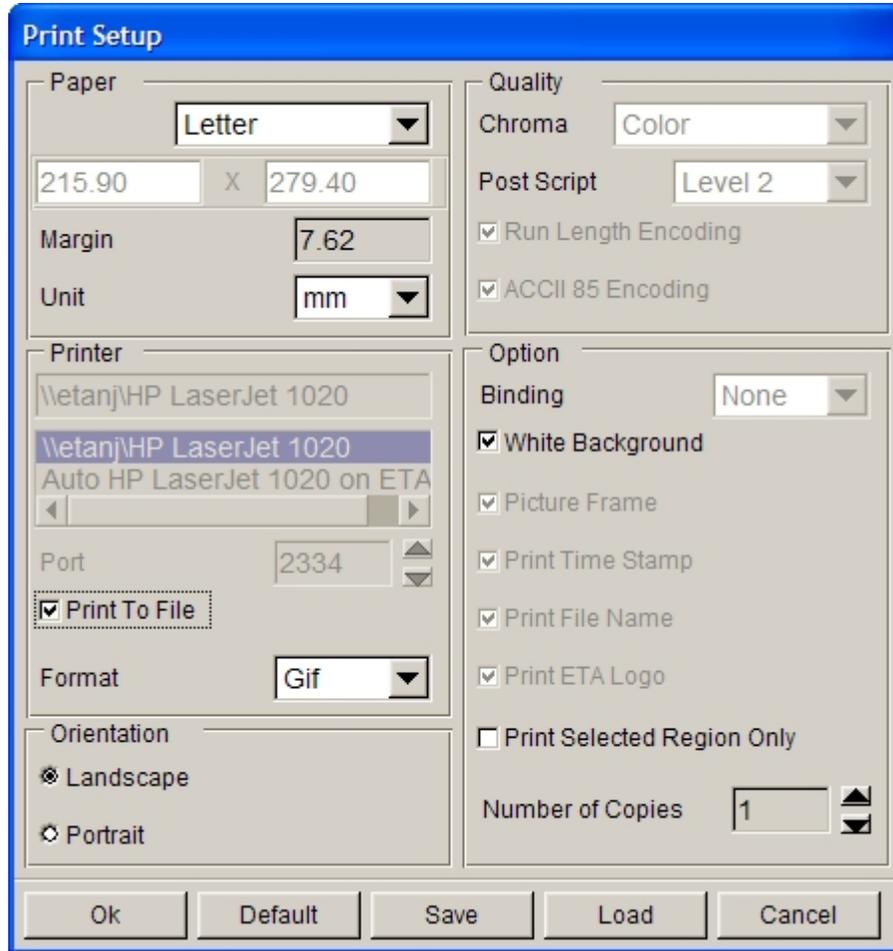


Figure 3.10.1 Print Setup window

### 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.

#### **WHITE BACKGROUND**

The background color of the image is white (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.

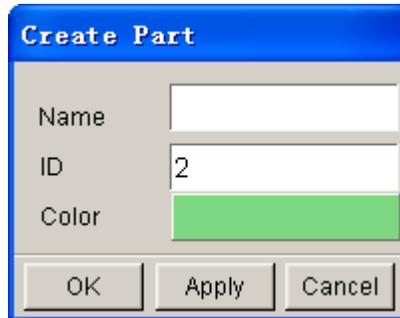


Figure 4.1.1 Create Part dialog box

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.

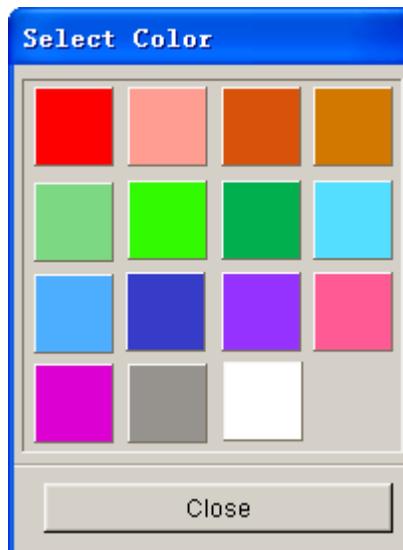


Figure 4.1.2 Select Color

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.

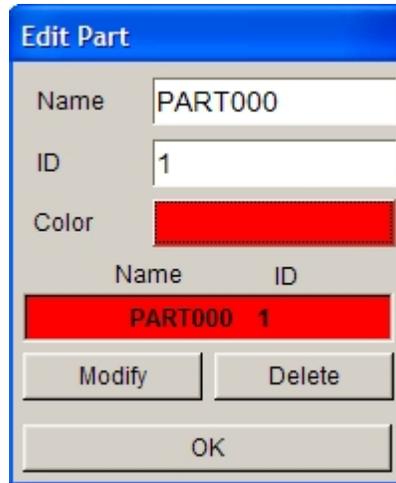


Figure 4.2.1 Edit Part dialog box

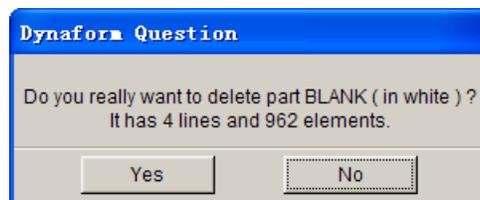
### 4.2.1 MODIFY

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.

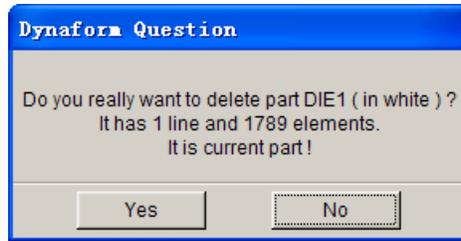
### 4.2.2 DELETE

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 enables the 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, the 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.

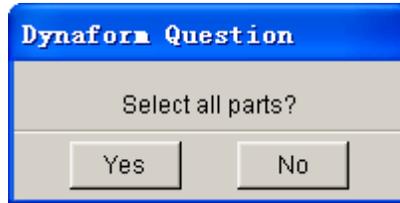


Figure 4.3.2 eta/DYNAFORM Question dialog box

## 4.4 ADD...TO PART

This function allows the user to organize lines, surfaces, and elements from one part to another. The user can also enter the part name manually to create a new part. The dialog box is illustrated in Figure 4.4.1.

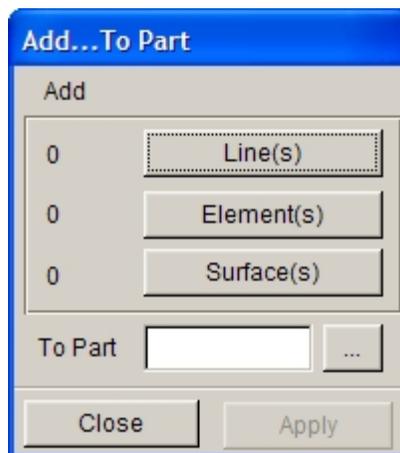


Figure 4.4.1 Add... To Part dialog box

### 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.

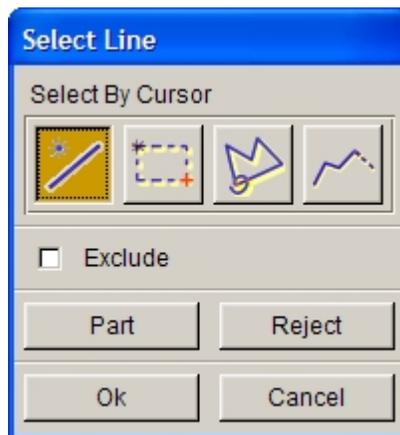


Figure 4.4.2 Select Line dialog box

- 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).

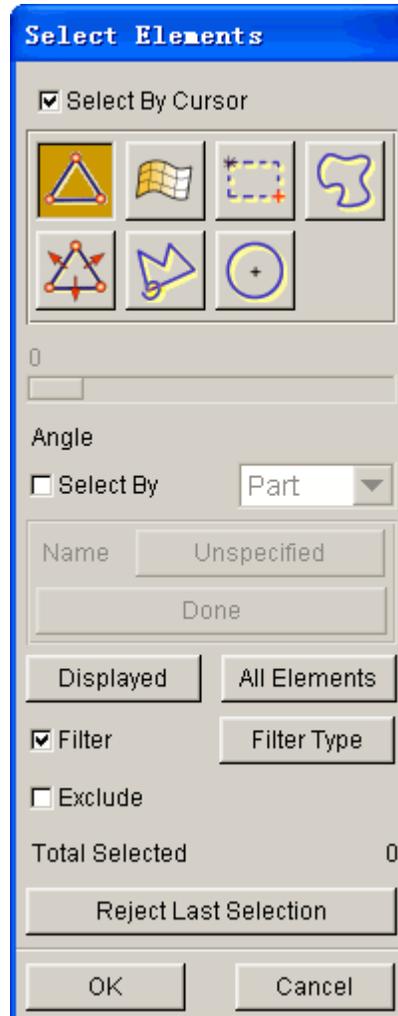


Figure 4.4.3 Select Element dialog box

- 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.

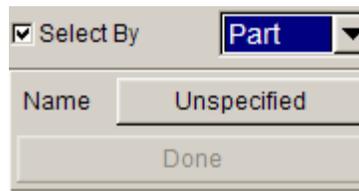


Figure 4.4.4 Select element by Part and/or Range



Figure 4.4.5 Select element by Range

#### 1. SELECT BY PART

This function allows the user to select elements by selecting parts, as shown in Figure 4.4.4. If this option is selected, after clicking the UNSPECIFIED button, the program will pop up a Select Part dialog box for the user to select a part. All elements in the selected part will be selected automatically and be highlighted on screen.

#### 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.

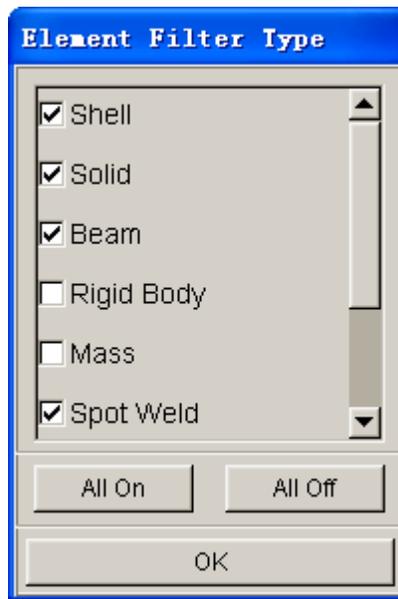


Figure 4.4.6 Element Filter Type dialog box

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

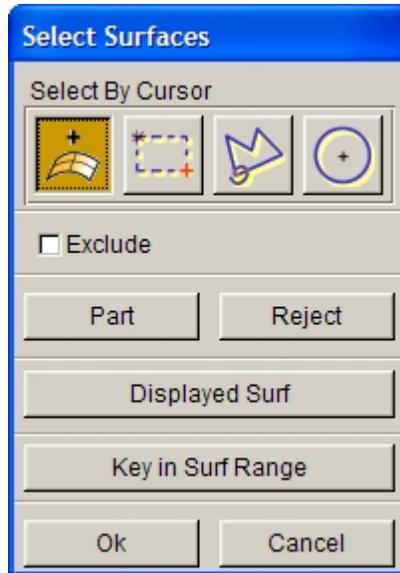


Figure 4.4.7 Select Surface dialog box

- **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.

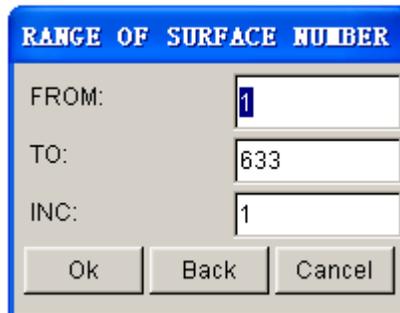


Figure 4.4.8 Define Surface Range dialog box

Other options for surface selection resemble 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.

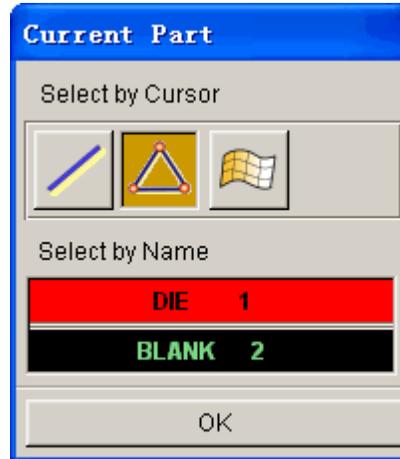


Figure 4.6.1 Current Part dialog box

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 4.5. 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.

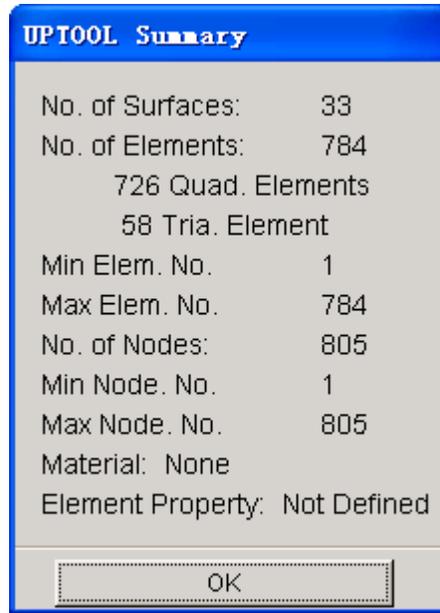
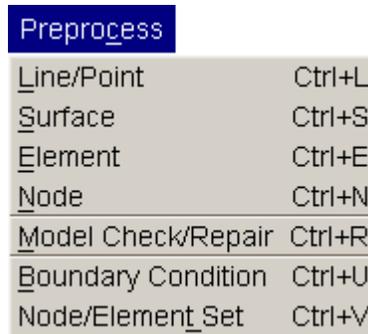


Figure 4.9.1 Part Summary dialog box

## Chapter 5

# PREPROCESS

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



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

Figure 5.1 Preprocess Menu

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.

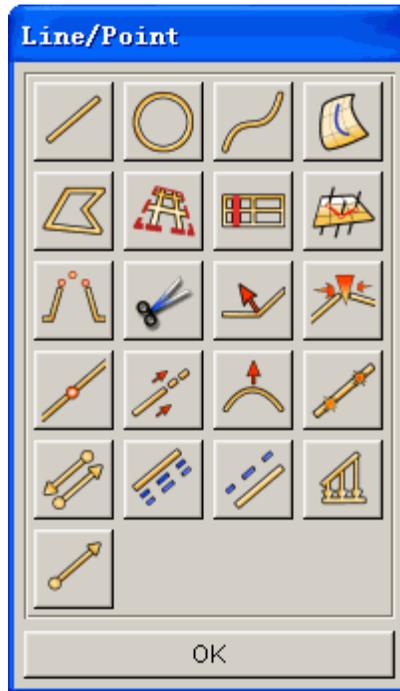


Figure 5.1.1 Line/Point dialog box

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

### 5.1.1 CREATE LINE



This function allows the user to draw a straight line. In eta/DYNAFORM, a line is formed by a sequence of points and is displayed by the set of straight line segments between adjacent points in the sequence. To create a straight line, the user has to define the location of two points. Various methods of creating points are described below.

1. To create a point of the line at an existing node or point.
  - Select NODE or POINT option from the INPUT COORDINATE dialog box as illustrated in Figure 5.1.4.
  - Select the location of the node or point. If the POINT option is selected, the following buttons are used to determine the generation of point.



At a point of an existing line close to the cursor location.



At an endpoint of a line.



At the midpoint of a line.



At the intersection of two lines.



On the workplane. Click on the workplane icon to display the grids on the display area and activate Work Plane and Grid Setting button. The user can generate a line on the workplane by clicking the left mouse button on two or multiple points, as shown in Figure 5.1.2.

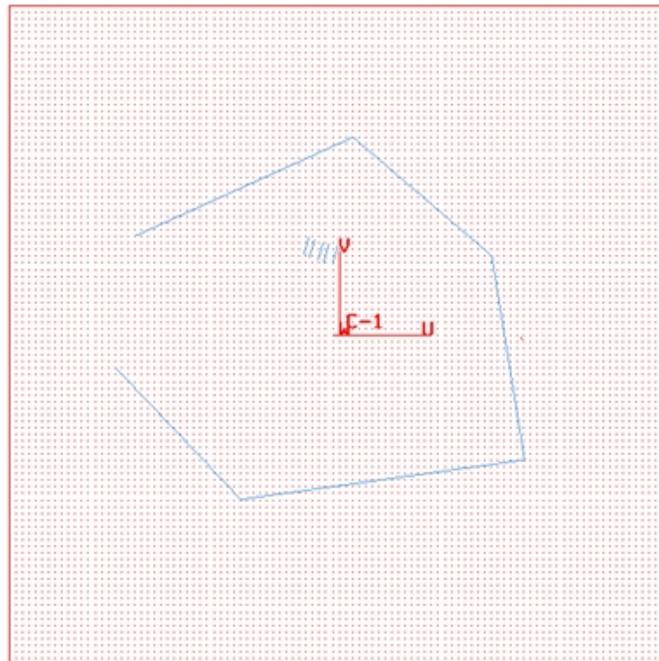


Figure 5.1.2 Working Plane and Grid

### 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.

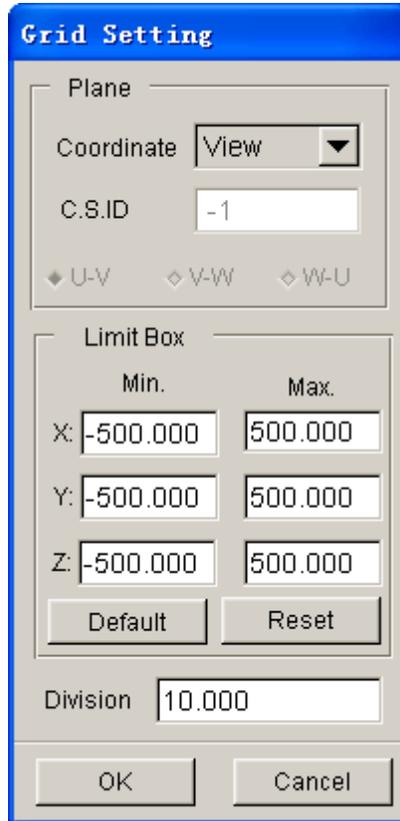


Figure5.1.3 Workplane and Grid Setting dialog box

**PLANE COORDINATE:**

Set the direction of the workplane, the default workplane is x-y plane of the Global 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 displayed grids.

**DIVISION**

Set the width of the grid division. The user can enter the width of grids.

2. To create a point by its coordinates, as shown in Figure 5.1.4.

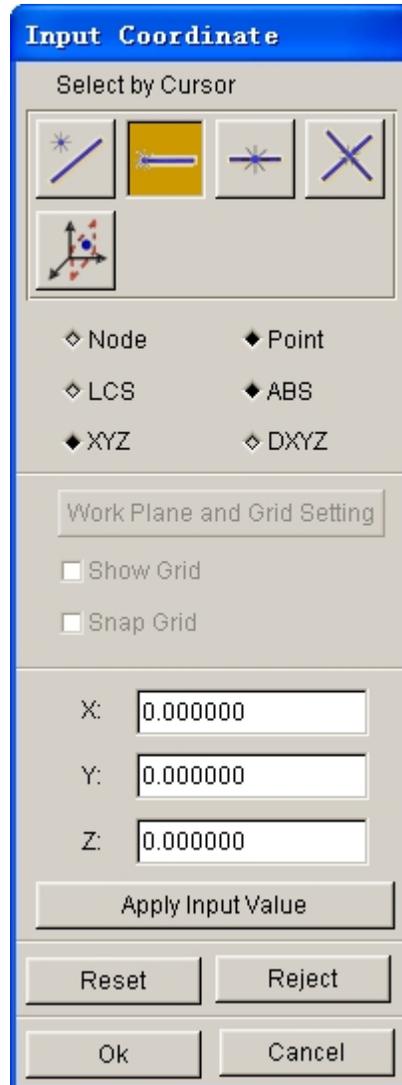


Figure 5.1.4 Input Coordinate dialog box

- Select the LCS or ABS option for the coordinates in the current local coordinate system or global system. Refer to Section 2.5, 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.
3. To cancel the last created point
    - Click REJECT once.
  4. To finish the definition of the line
    - Select OK or click on the middle mouse button.

5. 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.

## 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.

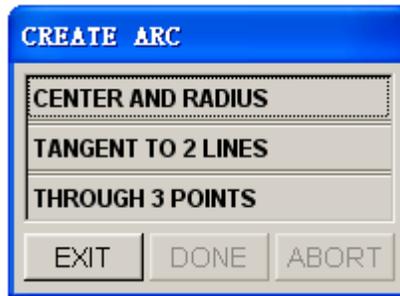


Figure 5.1.5 Create Arc dialog box

1. CENTER AND RADIUS
  - Define a LCS. Refer to Section 2.5, 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.

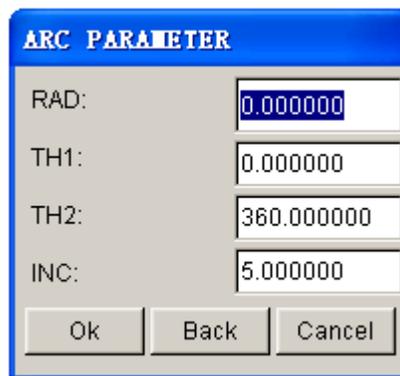


Figure 5.1.6 Arc Parameters dialog box

**RAD:** Radius of arc  
**TH1:** Beginning angle from U-axis  
**TH2:** Ending angle from U-axis  
**INC:** Angle increment between points (default is 5 degree) 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.

- Refer to section 5.1.1 and select three points/nodes in Figure 5.1.2 . 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, or their coordinates. A minimum of three points or nodes is required. The user selects the points and/or nodes using the INPUT COORDINATE function. Click on DONE button to complete spline curve generation.

## 5.1.4 SHORTEST CURVE



This function allows the user to calculate the shortest route between two nodes on element and the full length of the path exceeded two nodes, as shown in Figure 5.1.8.

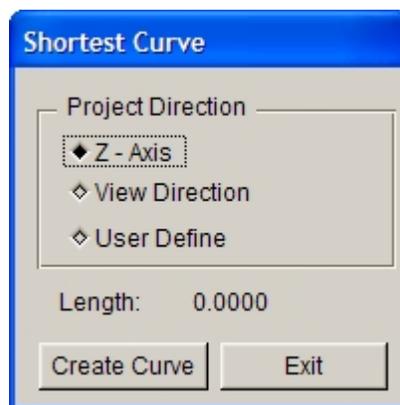


Figure 5.1.8 Shortest Curve dialog box

There are three types of projection direction for the user to choose: Z Axis, VIEW DIRECTION and USER DEFINE. If the user chooses USER DEFINE, the program will pop up a Vector dialog box as shown in Figure 5.1.9.

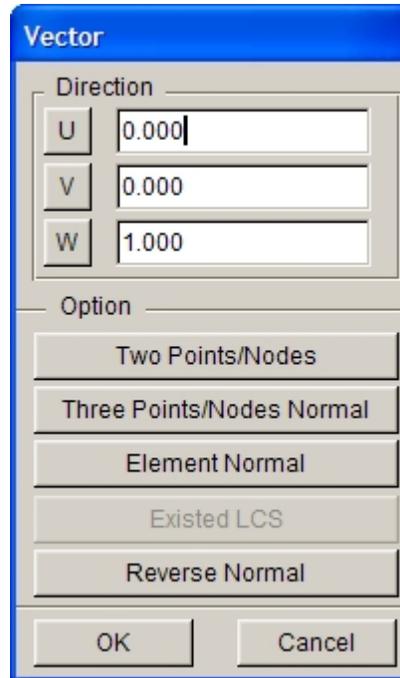


Figure 5.1.9 Vector dialog box

**U** — 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.

**V** — 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.

**W** — 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.

**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 5.1.10.



Figure 5.1.10 Select LCS dialog box

The user can click C.S.ID button to set the X, Y, Z axis of an existed local coordinate system as the direction vector.

**Reverse Normal** — Reverse the defined coordinates' direction.

After setting of projection direction, the user can select points directly on the screen and click the middle mouse button to confirm completion of this selection. The shortest path between two points will be highlighted on the screen, as shown in Figure 5.1.11. The full length of the path will be displayed in LENGTH option. The user can also click on CREATE CURVE button to generate the shortest curve on the screen.

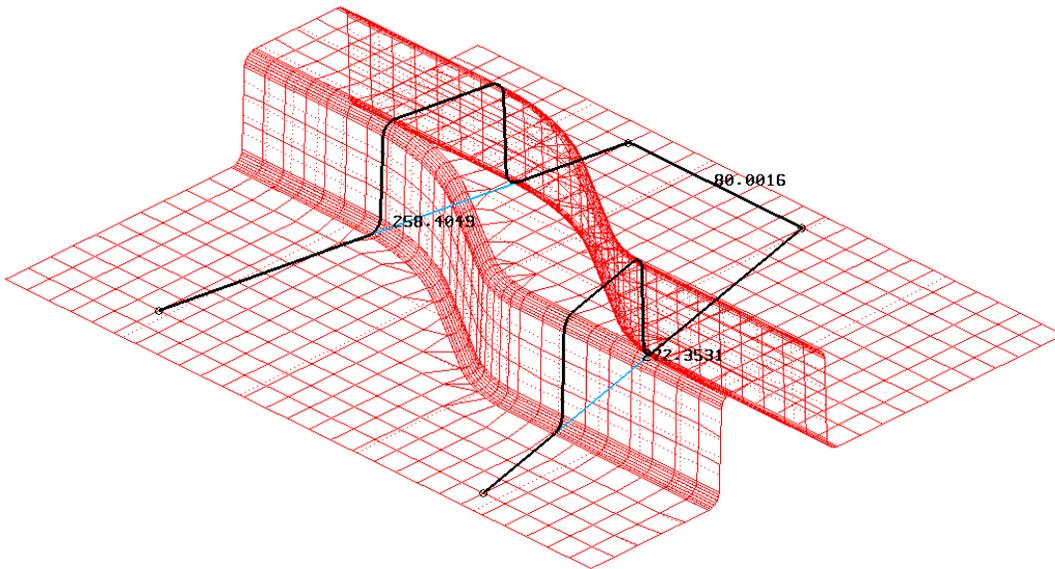


Figure 5.1.11 Displayed Path Length

## 5.1.5 BOUNDARY LINE

This function allows the user to create line along the boundary of finite element mesh or surface, as shown in Figure 5.1.12.

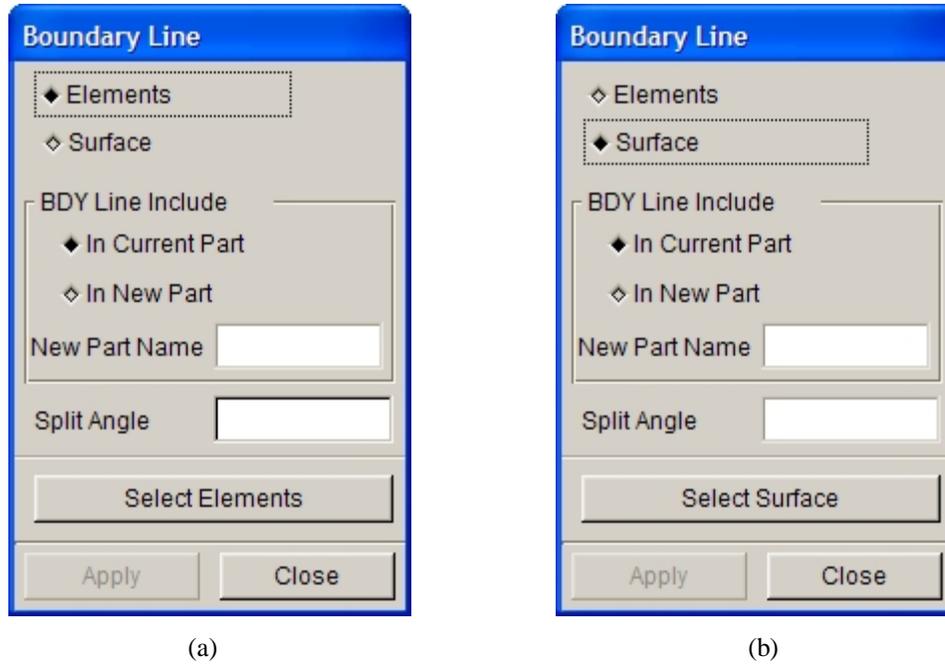


Figure 5.1.12 Boundary Line

### IN CURRENT PART

Create boundary line in the current part (default).

### IN NEW PART

This option places the created 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.

### 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. This option is disabled if the line is created via the boundary of surface.

## 5.1.6 S-LINE

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

- Select the surface.
- eta/DYNAFORM displays 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.



## 5.1.7 SECTION LINE

This function allows the user to create one or several section lines, measure section length, measure angle and show section lines. Click on this function, the program will display the dialog box as shown in Figure 5.1.13.

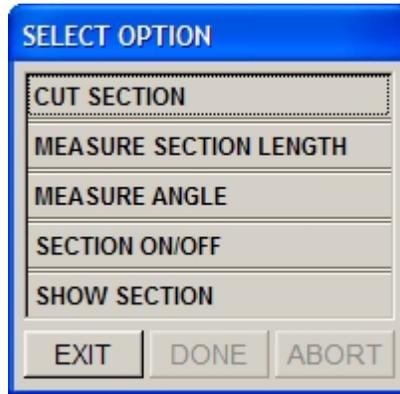


Figure 5.1.13 Select Option

### 5.1.7.1 CUT SECTION

This function allows the user to create one or several section lines. Section lines are a series of intersecting lines of coordinate plane and finite element mesh and surfaces. The generated section lines are included in part SURFSECL or ELEMSECL. After clicking on this function, the program will pop up a dialog box as shown in Figure 5.1.14.

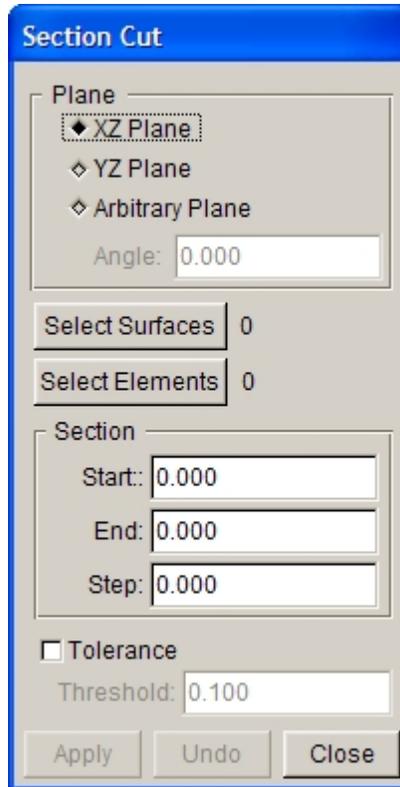


Figure 5.1.14 Define section plane

**XZ PLANE:** Cut section line on XZ plane.

**YZ PLANE:** Cut section line on YZ plane.

**ARBITRARY PLANE:** Cut section line on an arbitrary plane. After selection of this option, the ANGLE input box is activated for the user to enter a value.

**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.

**STEP:** Generate step of the section line.

**TOLERANCE:** Surface or element tolerance generated from section line.

The user can click APPLY to confirm the entry or click CLOSE to cancel the operation.

### 5.1.7.2 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 shown in Figure 5.1.15.



Figure 5.1.15 Select section dialog window

**CURSOR LOCATION:**

Select two points of the section line by cursor, and then the length of 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:**

If selecting the point through CURSOR LOCATION function, the first selected point can be canceled by clicking this option.

### 5.1.7.3 MEASURE ANGLE

This function allows the user to measure angle between two straight lines. The line is defined by two selected points. The information about the measured angle is displayed in the message prompt area.

### 5.1.7.4 SECTION ON/OFF

This function allows the user to only display SECTION layer on the screen, and set SECTION layer to current part layer.

### 5.1.7.5 SHOW SECTION LINE

This function is used to show the shape of each section line clearly in the 2D window. After creating section lines, click this option to display a dialog box with a list of the created section lines, as shown in Figure 5.1.16. Put the cursor on any column of the list and double click on the left mouse button, the corresponding serial numbers for all section lines will be displayed on the screen automatically. This function enables the user to delete or display the corresponding section lines conveniently.

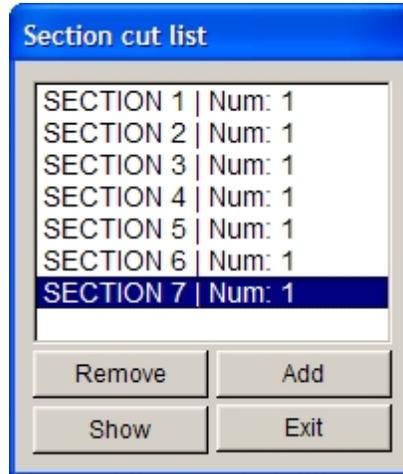


Figure 5.1.16 Section Cut List

- REMOVE:** Remove the section lines in the list.
- ADD:** Add the section lines in the list.
- SHOW:** Show the shape of the selected section lines in 2D window, as shown in Figure 5.1.17.

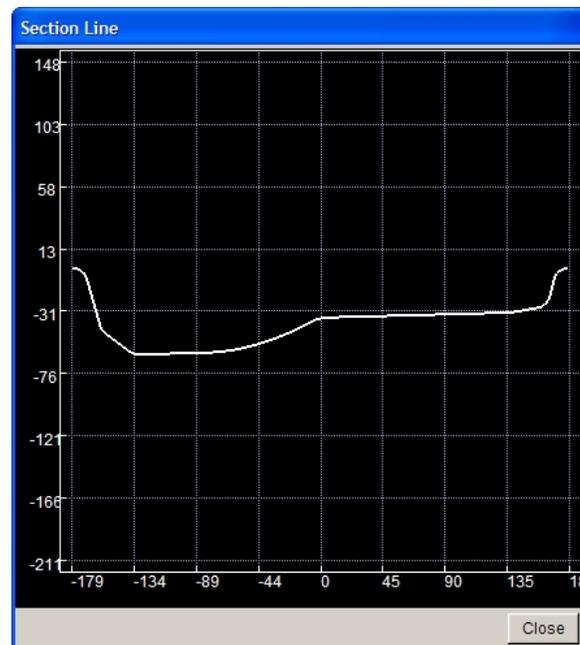


Figure 5.1.17 2D window to show section lines

In this 2D window, the user can move the position for section lines displayed in the 2D window by using Ctrl + middle mouse button, as shown in Figure 5.1.18. The section lines can be scaled by using Ctrl + right mouse button, as shown in Figure 5.1.19.

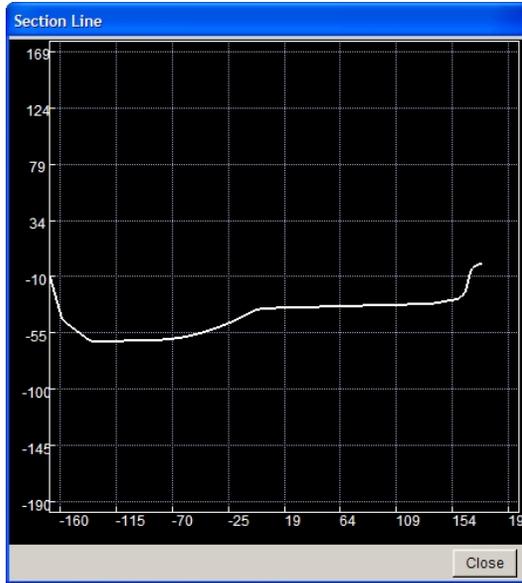


Figure 5.1.1 Move Section Line

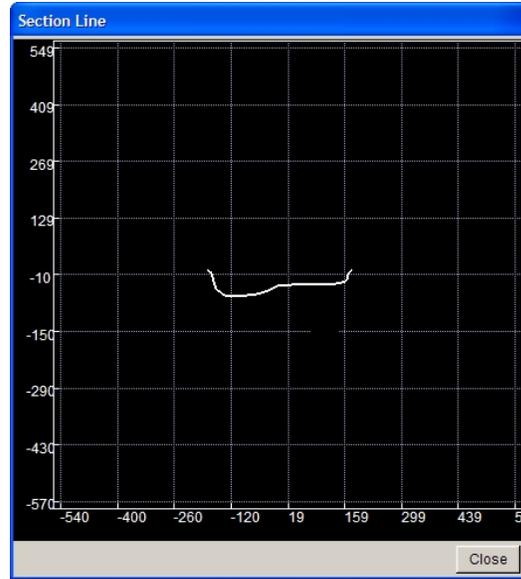


Figure 5.1.2 Zoom out Section Line

### 5.1.8 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.

### 5.1.9 BRIDGE LINE



This function generates a bridge 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 5.1.20 and prompts the user to select two lines.

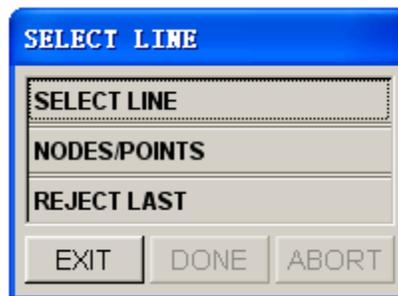


Figure 5.1.20 Select Line dialog box

There are three options available:

**SELECT LINE**

This is the default option. The user may start selecting lines from the screen without selecting this option. 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 by 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 5.1.21 illustrates the original lines and the generated bridge line.

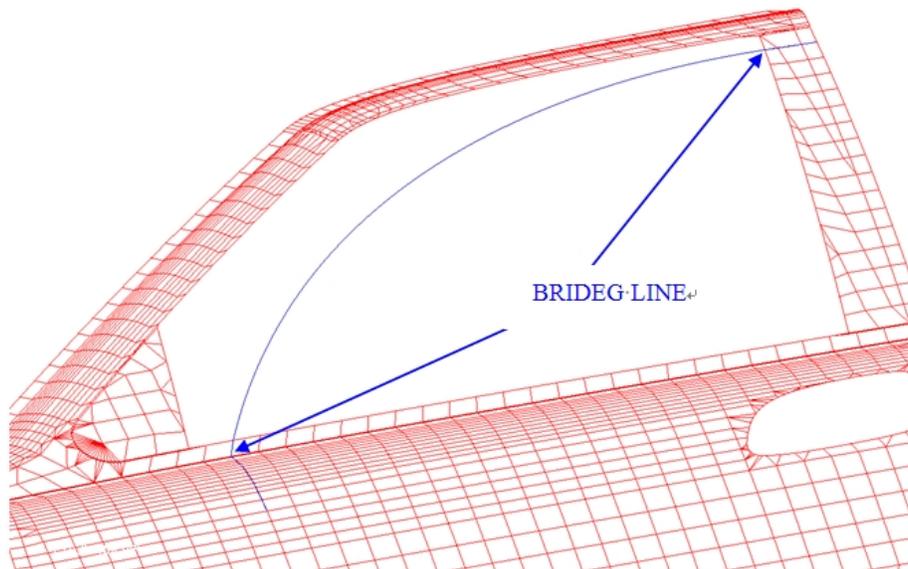


Figure 5.1.21 Bridge Line

### 5.1.10 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.

### 5.1.11 MODIFY LINE



This function allows the user to modify lines via inserting, moving and removing points on the lines, as shown in Figure 5.1.22.

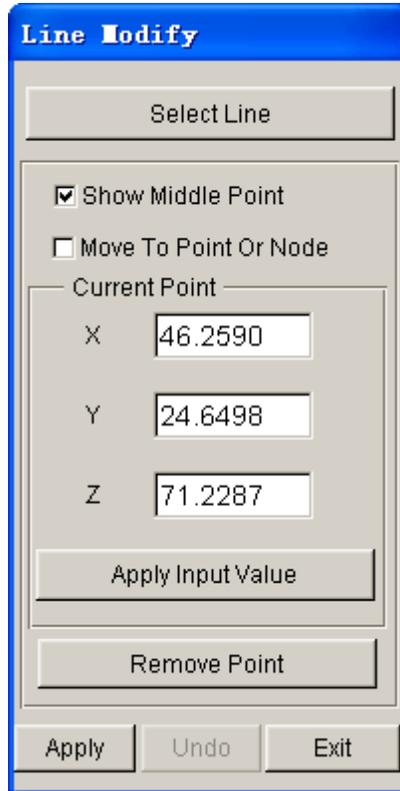


Figure 5.1.22 Line Modify dialog box

- 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 be modified.
- Enter the new coordinate value.
- Click APPLY INPUT VALUE.

This function allows the user to select SHOW MIDDLE POINT option to display middle point between two arbitrary points on the existed line, and to add the points by SHIFT+LEFT mouse button. Select MOVE TO POINT OR NODE option to accurately move an arbitrary point on the existed line to a point or a node in an element. Select the REMOVE POINT button to delete the arbitrary point on the existed line.

### 5.1.12 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 4.3.1, PART/ADD LINE, for more information.
2. Select the split location using one of the functions in INPUT COORDINATE dialog box. Refer to Section 5.1.1, CREATE LINE, for more information.

### 5.1.13 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 5.1.23, for the user to define the Line Gap value between lines. If the user doesn't toggle on the line gap option, all selected lines will be combined. The default is 0.5.



Figure 5.1.23 Combine Line dialog box

- 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 to complete the operation.
- The user can click Select Lines to continue the operation or click on EXIT button to exit Combine Line function.

### 5.1.14 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 5.1.24.

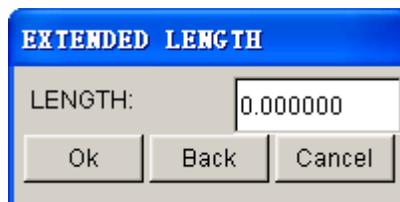


Figure 5.1.24 Extended Length dialog box

### 5.1.15 EDIT LINE



This function allows the user to edit lines.

After selecting a line, the program will display CURVE EDIT dialog box, as shown in Figure 5.1.25.

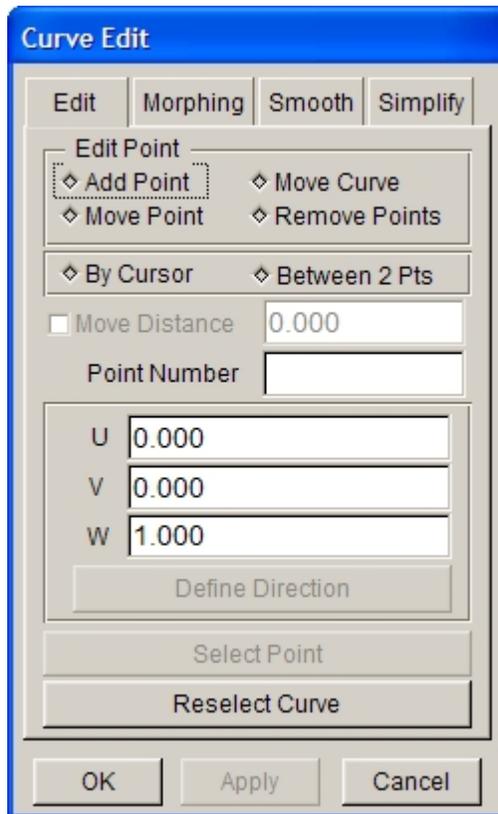


Figure 5.1.25 Curve Edit dialog box

There are four options available in this dialog box.

#### 1. EDIT

This function edits lines by editing points on the selected lines. Figure 5.1.25 shows the options for editing lines.

##### ADD POINT

If this option is selected, BY CURSOR LOCATION and BETWEEN TWO POINTS buttons are activated.

The user can select BY CURSOR LOCATION button to pick any point on the line to add.

If BETWEEN TWO POINTS button is selected, the user can add point on the selected line by cursor directly or click Select Point button. The program will display some points on the line and prompt the user to select two adjacent points. After selection of these two adjacent points, the POINT NUMBER window will be activated to prompt the user to enter the point number between two adjacent points. If the selected second point is not adjacent to the first point, the program will print a message: INVALID POINT SELECT, REJECTED.

### MOVE CURVE

This function allows the user to move a selected curve. Once this option is selected, the data windows for DX, DY and DZ will be activated. The user can enter any motion vector value relative to the global X, Y and Z directions, then click on the middle mouse button or APPLY button to confirm.

### MOVE POINT

This function enables the user to move point on the selected line. Once this option is selected, SELECT POINT button and three data windows are activated, as shown in Figure 5.1.26. The user can move point on the selected line directly by cursor or click SELECT POINT button to select a point on the line. Select MOVE DISTANCE option and enter the desired value. Then, select USER DEFINE option to define the move direction. Please refer to Section 5.1.4 Shortest Curve for details on USER DEFINE. Click on the middle mouse button or APPLY button to confirm.

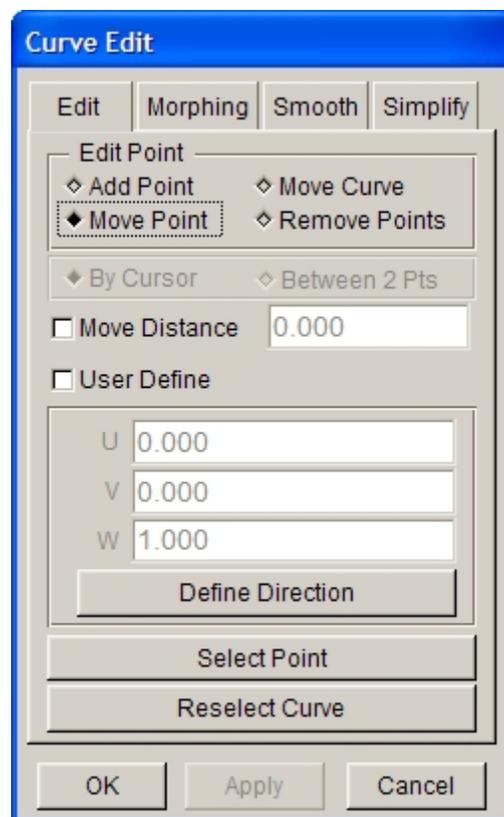


Figure 5.1.26 Move point on curve

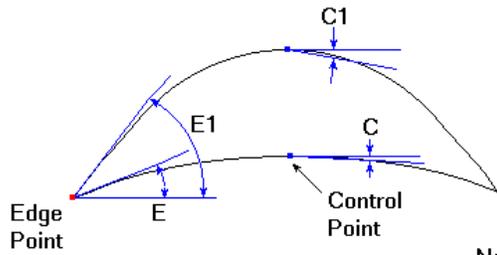
### REMOVE POINT

This function enables the user to remove point on the selected line. Once this option is selected, the Select Point button is activated. The user can select points on line or click on the middle mouse button to delete the selected points.

## 2. MORPHING

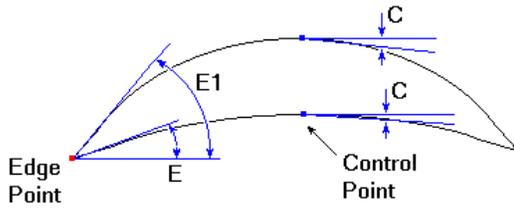
This function allows the user to modify lines by morphing. There are four methods in Morphing function. Figure 5.1.27 illustrates the features and results of various morphing methods.

DIFFERENT TYPES OF SURFACE SECTION FOR MORPHING

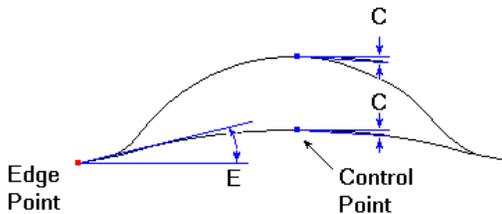


Type 1 - No constraint. Curvature at Control Point and Edge Point change.  
 C1 not equal to C.  
 E1 not equal to E.  
 This is the shape currently used in all morphing types. No very suitable.

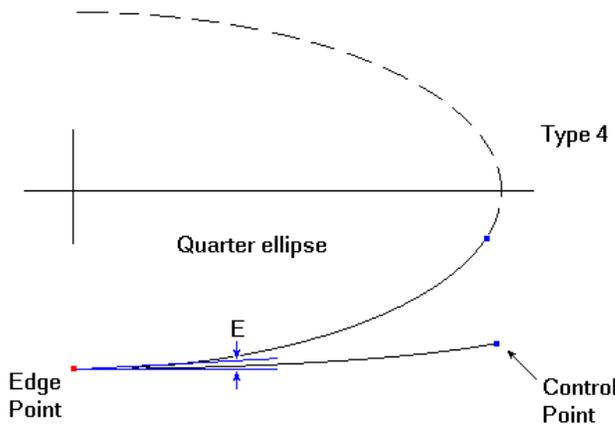
Note: C - Angle between the element plane and horizontal plane at Control Point. C1 - Angle C after morphing.  
 E - Angle between the element plane and horizontal plane at Edge Point. E1 - Angle E after morphing.



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 5.1.27 Morphing Methods

**NO CONSTRAINT:** There is no constraint imposed on the line when it is morphed. The morphed shape will assume a natural spline curve. Refer to type 1 as illustrated in Figure 5.1.27 for the shape of line after morphing.

**KEEP C ANGLE:** The tangent at the control point on the line is constrained. The morphed shape will maintain the original tangent direction at the control point of the line. Refer to type 2 as illustrated in Figure 5.1.27 for the shape of line after morphing.

**KEEP E ANGLE:** The tangent at both end points on the line is constrained. The morphed shape will maintain the original tangent direction at the end points of the line. Refer to type 3 as illustrated in Figure 5.1.27 for the shape of line after morphing.

**KEEP CONVEX:** Maintain the convex edge by changing the curvature of the line. Refer to type 4 as illustrated in Figure 5.1.27 for the shape of line after morphing.

Figure 5.1.28 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 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 line in real time. Click on the RIGHT MOUSE BUTTON to accept the morphed shape. Click on the APPLY button to update the morphed line automatically. eta/DYNAFORM calculates a default morphing direction that is normal to the line. The user may click on the USER DEFINE option to enter the desirable vector for the morphing direction. Please refer to Section 5.1.4 Shortest Curve for details on USER DEFINE.

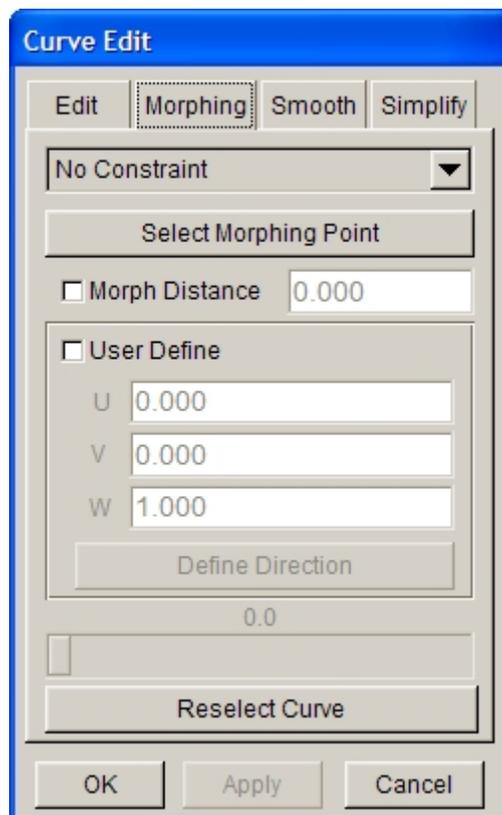


Figure 5.1.28 Morphing Curve

### 3. SMOOTH

This function enables the user to smooth the selected lines, as shown in Figure 5.1.29. There are four methods that can be used to smooth lines.

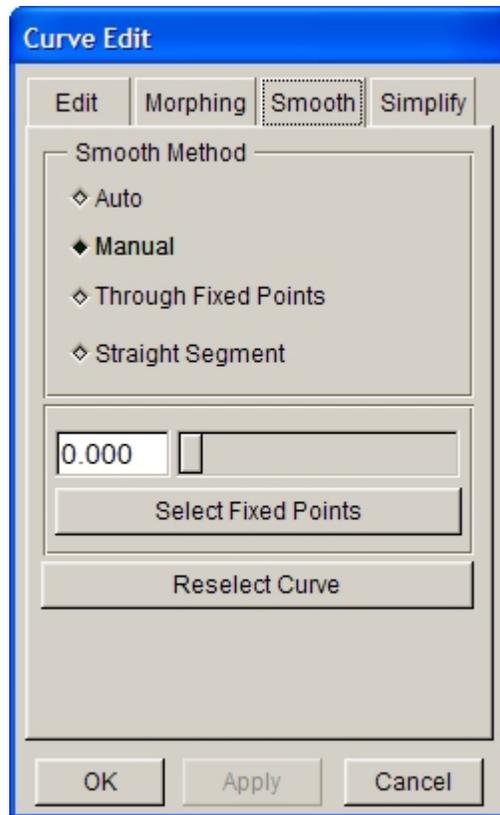


Figure 5.1.29 Smooth Curve

**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 control 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:** When this function is toggled ON, the Select Segment button will be activated, as shown in Figure 5.1.30. The user can select two points on the selected curve directly by cursor or click on SELECT SEGMENT button and then select point by cursor. The curve between two points will be smoothed to be a straight segment. Click the middle mouse button or Apply button to confirm.

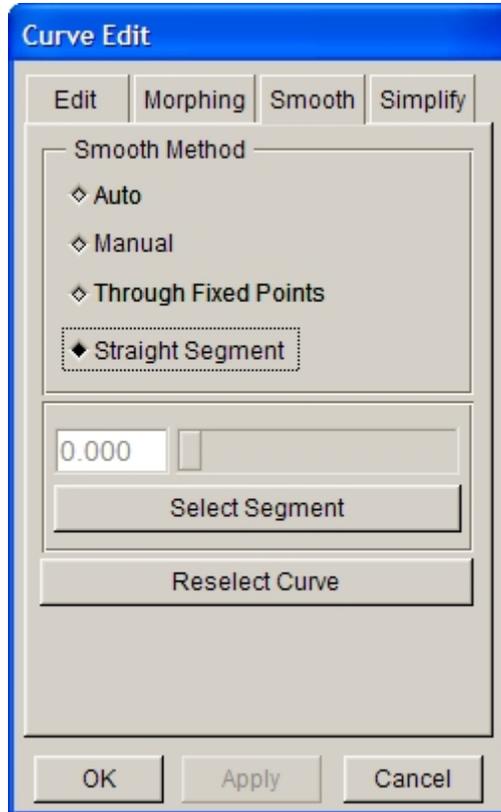


Figure 5.1.30 Straight Segment

#### 4. SIMPLIFY

This function is used to simplify the selected line by reducing the number of points on curve. The dialog box is illustrated in Figure 5.1.31.

##### **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 another line to simplify.

##### **APPLY**

Accept the operation without exiting the dialog box..

##### **OK**

Accept the operation and dismiss the dialog box.

##### **CANCEL**

Abort the operation and dismiss the dialog box.

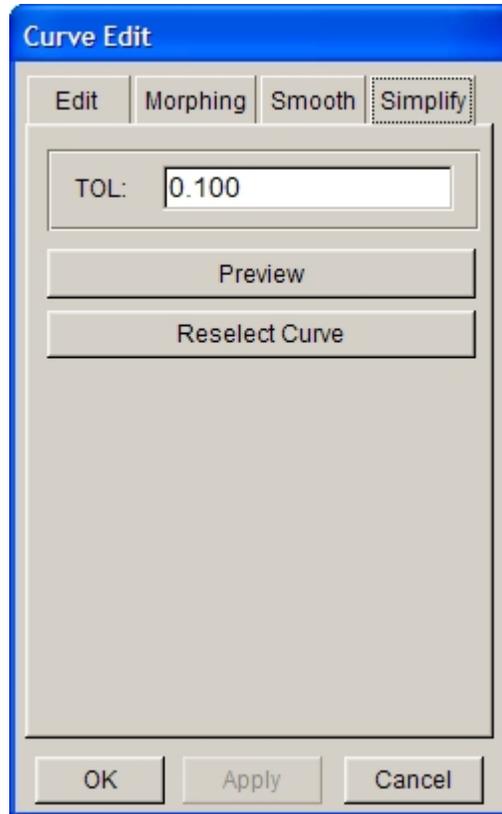


Figure 5.1.31 Simplify Curve

### 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 5.1.32.

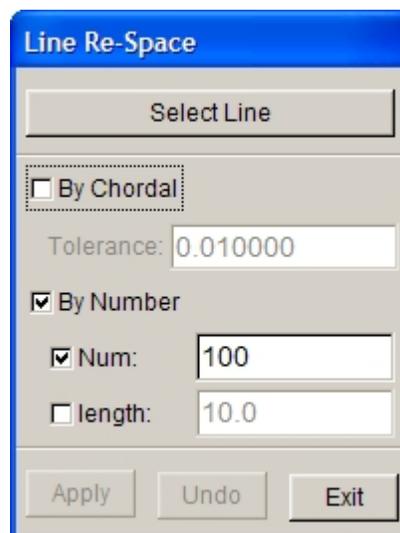


Figure 5.1.32 Line Re-Space dialog box

- Click on SELECT LINE and select the line using mouse cursor.

- The Line Re-space dialog box will pop-up as illustrated in Figure 5.1.32. There are three options available for the user to decide the density of point on line.

#### BY CHORDAL

This option enables the user to adjust the number of points on line by entering the tolerance. The number of points on each line cannot be more than 500.

#### NUMBER

This option shows the number of redistributed points on line. The default value is 100. The maximum number of points on each line is 500. If the input value is more than 500, the program will display a prompt message.

#### LENGTH

This option allows the user to confirm the number of points via the distance between points on line. The default value is 10.

- After entering the chordal tolerance, click APPLY.
- The user can click UNDO to cancel the current operation, or click SELECT LINE again to continue the curve selection.
- Click EXIT to exit this dialog box.

### 5.1.17 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.

### 5.1.18 TRANSFORM



This function allows the user to translate, rotate, mirror and scale the line, and any other transform operations according to the coordinates. Figure 5.1.33 shows the interface of this function.

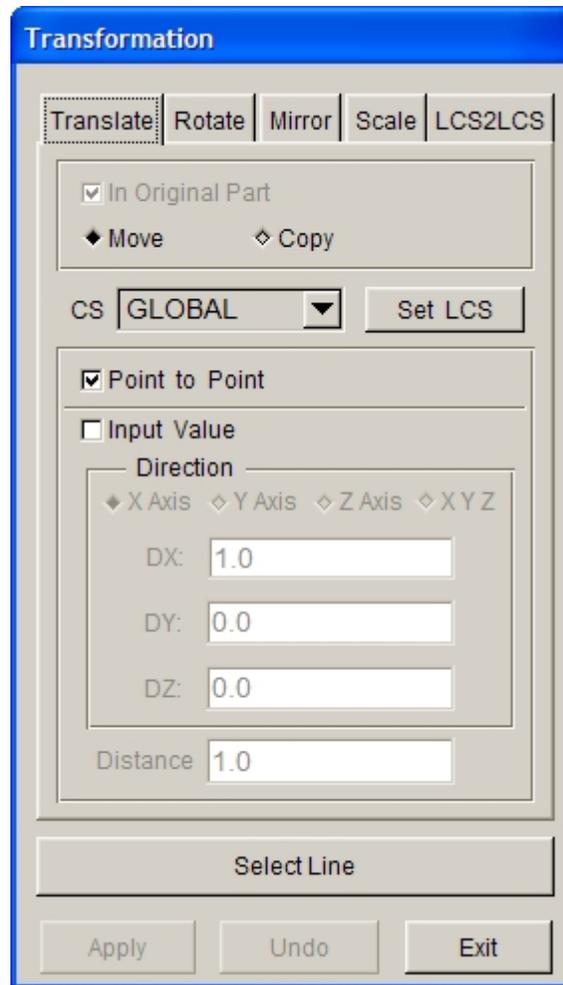


Figure 5.1.33 Transformation Interface

### 5.1.18.1 TRANSLATE

1. Select a translation type: MOVE or COPY, as shown in Figure 5.1.34. Once COPY is selected, IN ORIGINAL PART option is activated. The user can select or deselect IN ORIGINAL PART. If it is not selected, the copied line will be in the current part.
2. Select the coordinate system (CS). If SET LCS is selected, the program will display the LCS window for the user to define a new LCS. Details about defining LCS refer to Section 2.5, LOCAL COORDINATE SYSTEM. The transformation will be completed in the defined system. If SET LCS is not selected, LCS window will not be displayed, and the transformation will be completed in the current coordinate system.
3. Select or deselect POINT TO POINT MOVE. If POINT TO POINT MOVE is selected, the INPUT COORDINATE window as shown in Figure 5.1.4 will be displayed for the user to select two points. The program will perform translation transformation according to the differences of three coordinates for the selected two points. This option is only active during translation operation.
4. Select or deselect INPUT VALUE. If INPUT VALUE is selected, the Direction and Distance options will be activated. The user can select X Axis, Y Axis, Z Axis or XYZ, and enter the X, Y, Z values to determine the translation direction. The user can determine the translation distance by entering a value.
5. Select line. Refer to Section 4.4.1 PART/ADD LINE for more information.

6. Click APPLY button.

### 5.1.18.2 ROTATE

1. Select a rotation type: MOVE or COPY, as shown in Figure 5.1.34. Once COPY is selected, IN ORIGINAL PART option is activated. The user can select or deselect IN ORIGINAL PART. If it is not selected, the rotated line will be in the current part.

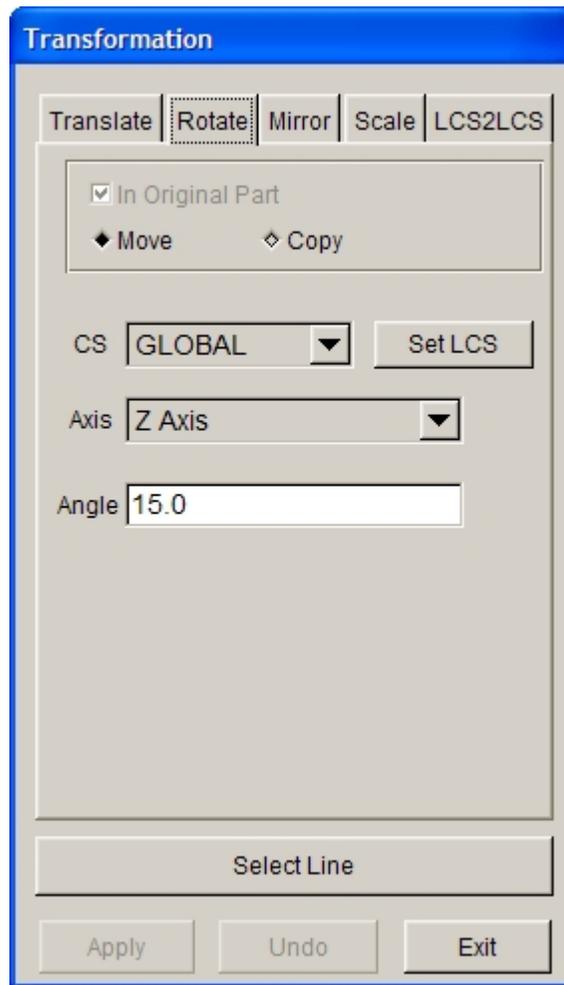


Figure 5.1.34 Rotate Line Interface

2. Select the coordinate system (CS). If SET LCS is selected, the program will display the LCS window for the user to define a new LCS. Details about defining LCS refer to Section 2.5, LOCAL COORDINATE SYSTEM. The transformation will be completed in the defined system. If SET LCS is not selected, LCS window will not be displayed, and the transformation will be completed in the current coordinate system.
3. Select the coordinate system. The user can select X Axis, Y Axis or Z Axis as the rotation direction of the line using drop-down menu.
4. Enter the rotation angle.
5. Select line. Refer to Section 4.4.1 PART/ADD LINE for more information.
6. Click APPLY button.

### 5.1.18.3 MIRROR

1. Select a mirror type: MOVE or COPY, as shown in Figure 5.1.35. Once COPY is selected, IN ORIGINAL PART option is activated. The user can select or deselect IN ORIGINAL PART. If it is not selected, the mirrored line will be in the current part.

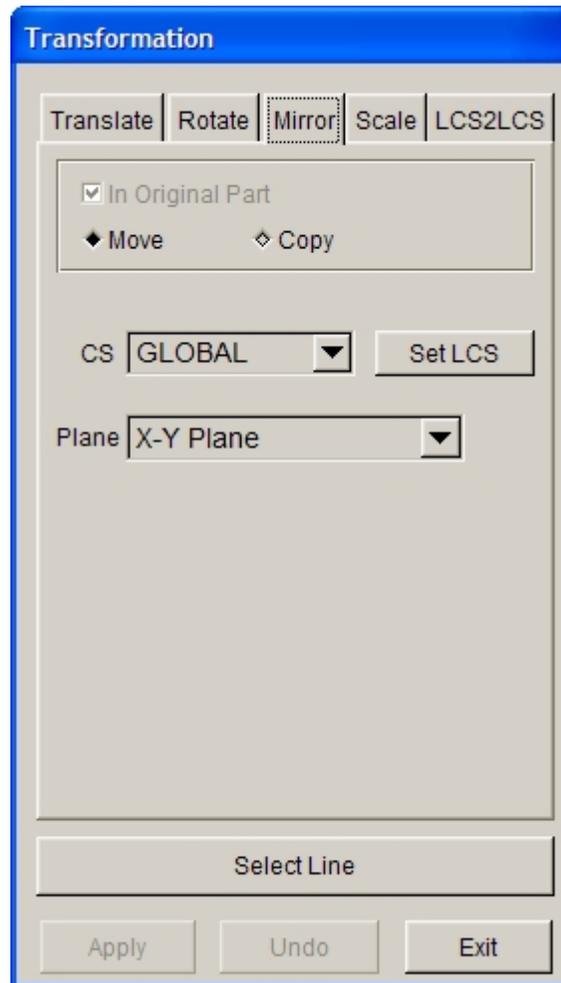


Figure 5.1.35 Mirror Line Interface

2. Select CS (coordinate system). If SET LCS is selected, the program will display the LCS window for the user to define a new LCS. Details about defining LCS refer to Section 2.5, LOCAL COORDINATE SYSTEM. The transformation will be completed in the defined system. If SET LCS is not selected, LCS window will not be displayed, and the transformation will be completed in the current coordinate system.
3. Select a coordinate plane as the mirror plane. The user can use the drop-down menu to select X-Y Plane, Y-Z Plane or Z-X Plane as the mirror plane of line.
4. Select line. Refer to Section 4.4.1 PART/ADD LINE for more information.
5. Click APPLY button.

### 5.1.18.4 SCALE

1. Select a Scale type: MOVE or COPY, as shown in Figure 5.1.36. Once COPY is selected, IN ORIGINAL PART option is activated. The user can select or deselect IN ORIGINAL PART. If it is not selected, the scaled line will be in the current part.

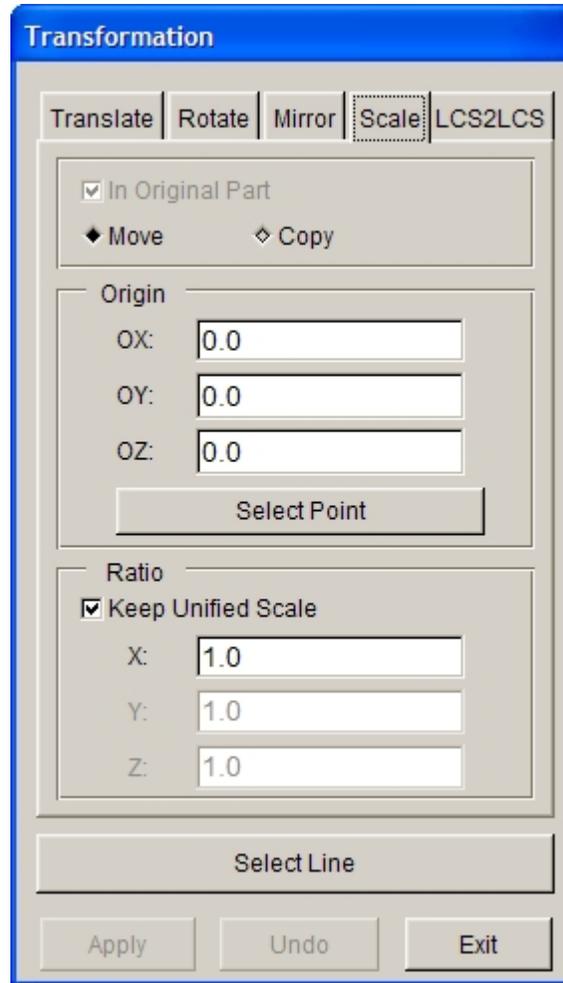


Figure 5.1.36 Scale Line Interface

2. The user can click SELECT POINT to define the coordinate origin for scaling.
3. Enter the scale factor. If KEEP UNIFIED SCALE is selected, the line can only be scaled in X direction. If KEEP UNIFIED SCALE is deselected, the line can be scaled in X, Y, Z directions.
4. Select line. Refer to Section 4.4.1 PART/ADD LINE for more information.
5. Click APPLY button.

### 5.1.18.5 LCS2LCS

1. Select a transformation type: MOVE or COPY, as shown in Figure 5.1.37. Once COPY is selected, IN ORIGINAL PART option is activated. The user can select or deselect IN ORIGINAL PART. If it is not selected, the transformed line will be in the current part.

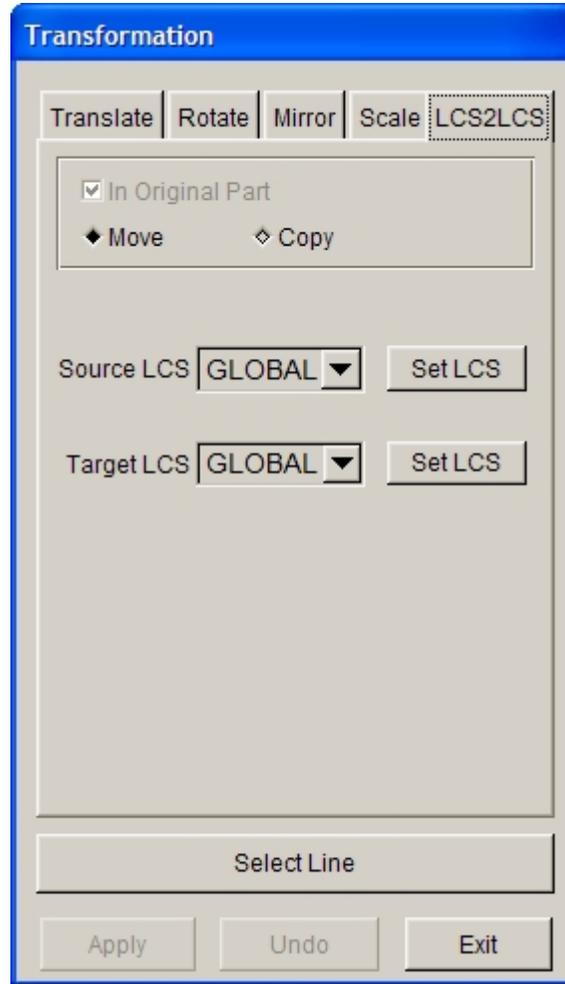


Figure 5.1.37 LCS2LCS Interface

2. Select Source LCS. If SET LCS is selected, the program will display the LCS window for the user to define a new LCS. Details about defining LCS refer to Section 2.5, LOCAL COORDINATE SYSTEM. Source LCS will be the new defined coordinate system. The user can also select Global coordinate system or the defined Local coordinate system from the drop-down menu of Source LCS as Source LCS.
3. Select Target LCS. The user can select SET LCS to define the new Local coordinate system, or select Global coordinate system or the defined Local coordinate system from the drop-down menu of Target LCS as Target LCS.
4. Select line to be transformed.
5. Click APPLY button.

### 5.1.19 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 5.1.38.
- The eta/DYNAFORM Question dialog box as illustrated in Figure 5.1.39, is displayed. There is an arrow in the display area indicating the offset direction.



Figure 5.1.38 Offset Distance window

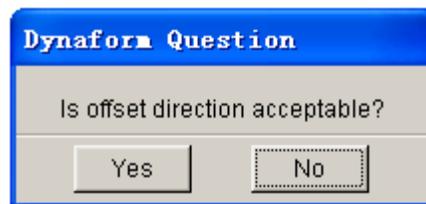


Figure 5.1.39 Dynaform Question dialog box

Click on YES button to accept the direction.

Click on NO button to reverse the direction.

## 5.1.20 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 for the projected lines. The projected direction is along the W-axis of the defined coordinate system.

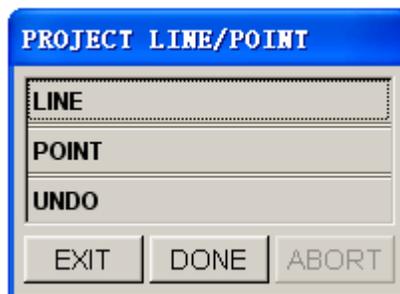


Figure 5.1.40 Project Line/Point window

- PROJECT LINE/POINT dialog box as illustrated in Figure 5.1.40 is displayed.

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.

### 5.1.21 SHOW LINE



This function allows the user to identify any existing line and its line direction. 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 5.1.41, two approaches can be used to show a line.

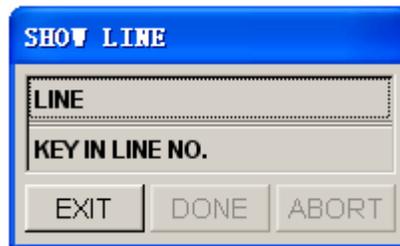


Figure 5.1.41 Show Line window

1. LINE  
Select line using mouse cursor.
2. KEY IN LINE NO.  
Enter the line number and then click on the OK button.

## 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.



Figure 5.2.1 Surface dialog box

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.

### 5.2.1 CREATE 2L



This function allows the user to create a surface from two selected lines (curves). The user can select a line using the options in the CONTROL KEYS dialog box as illustrated in Figure 5.2.2.



Figure 5.2.2 Control Keys dialog box

- 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 DONE 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 selected 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 normal 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.

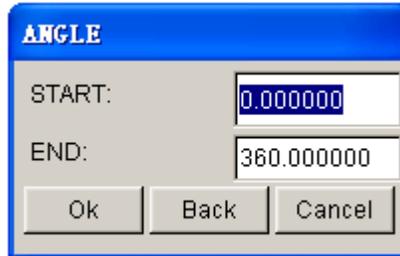


Figure 5.2.3 Angle dialog box

## 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.
- Select the first line as section (generatrix) and the second line as the sweep line.

## 5.2.6 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.

## 5.2.7 GENERATE MIDDLE SURFACE



This function enables the user to generate middle surfaces of 3D sheet-metal parts. After clicking on this function, the Select Surface dialog box appears to prompt the user in a message to select all surfaces in the entire solid model. If users only need to generate middle surfaces of partial solid model, they can select the corresponding surfaces, as shown in Figure 5.2.4.

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 5.2.4.

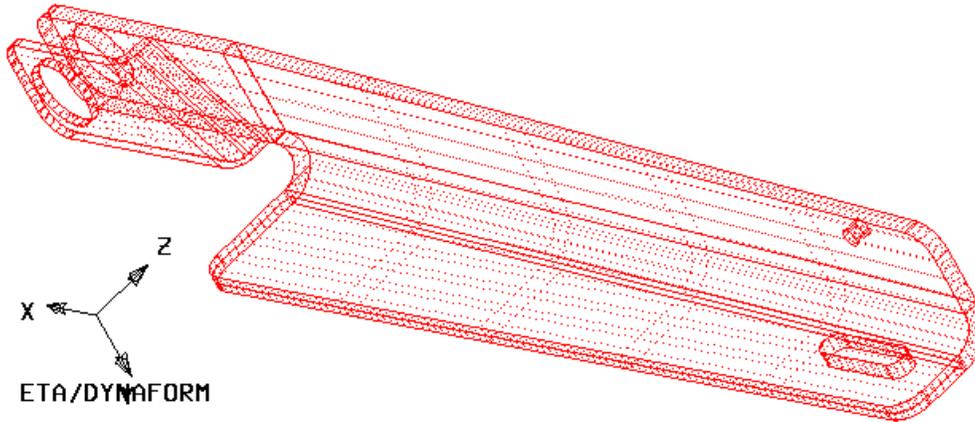


Figure 5.2.4 Select all surfaces used to generate mid surface

2. After selecting the entire surfaces, the program will generate the mid surface automatically as Figure 5.2.5 shown. At the same time a new part named MIDSRF is created and all the created mid surfaces are placed in this part.

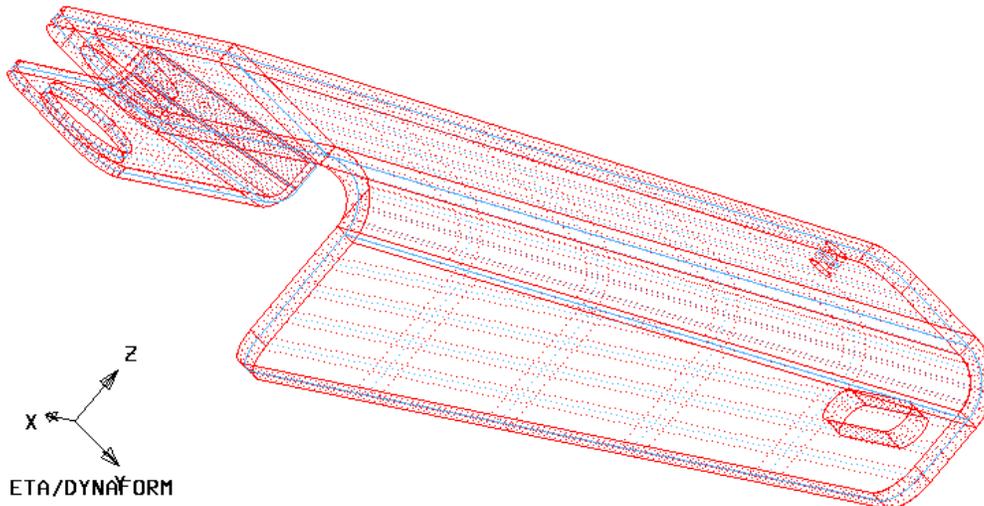


Figure 5.2.5 Mid Surfaces

3. After the mid surfaces are created, the SELECT OPTION dialog box as illustrated in Figure 5.2.6 shown will pop-up.

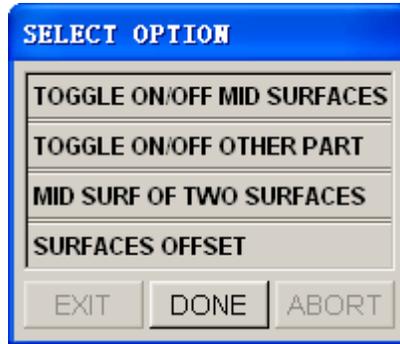


Figure 5.2.6 Select Options dialog box

### TOGGLE ON/OFF MID SURFACES

This function enables user to toggle on/off the MIDSRF part. Figure 5.2.7 shows the mid surface only.

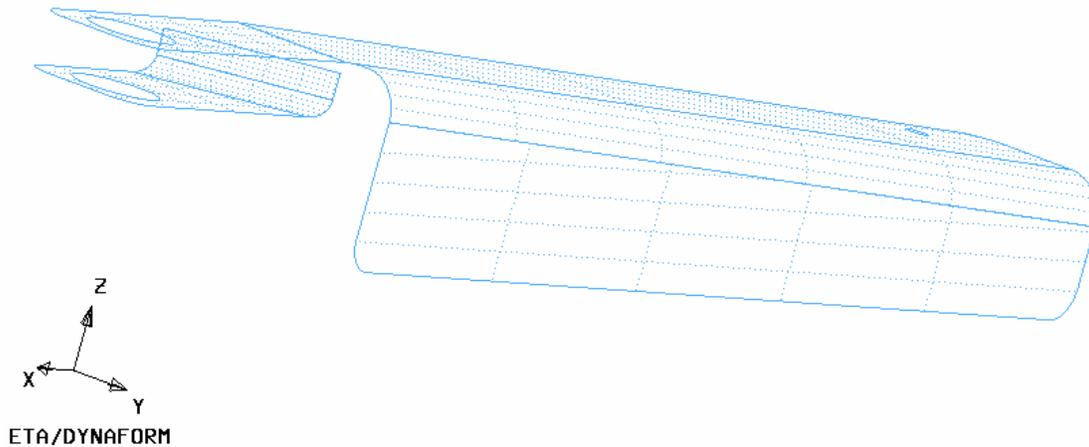


Figure 5.2.7 Middle surfaces display only

### TOGGLE ON/OFF OTHER PART

This function enables user to view only the MIDSRF 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 5.2.8.

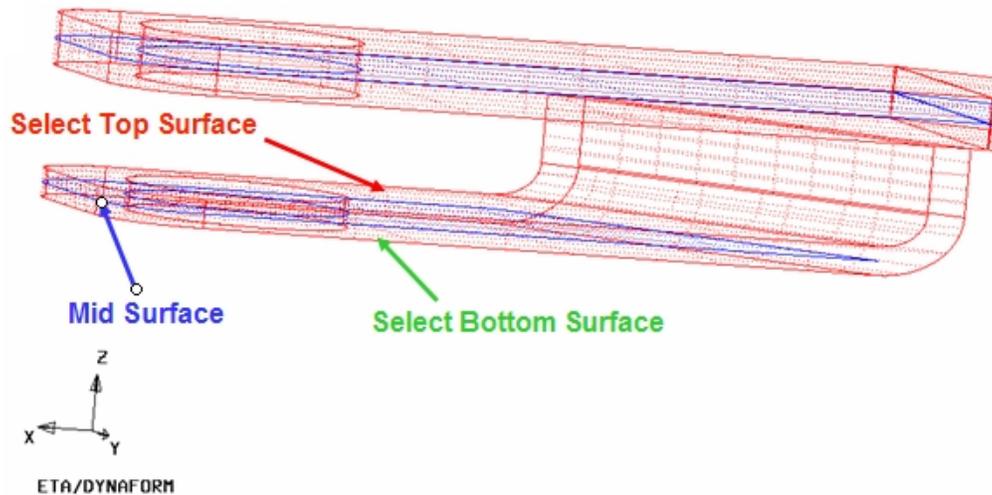


Figure 5.2.8 mid surface of two surfaces

### **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 5.2.9. The user can click the REVERSE option to reverse the offset direction or click the ACCEPT to accept the existing offset direction.



Figure 5.2.9 Select Offset direction dialog box

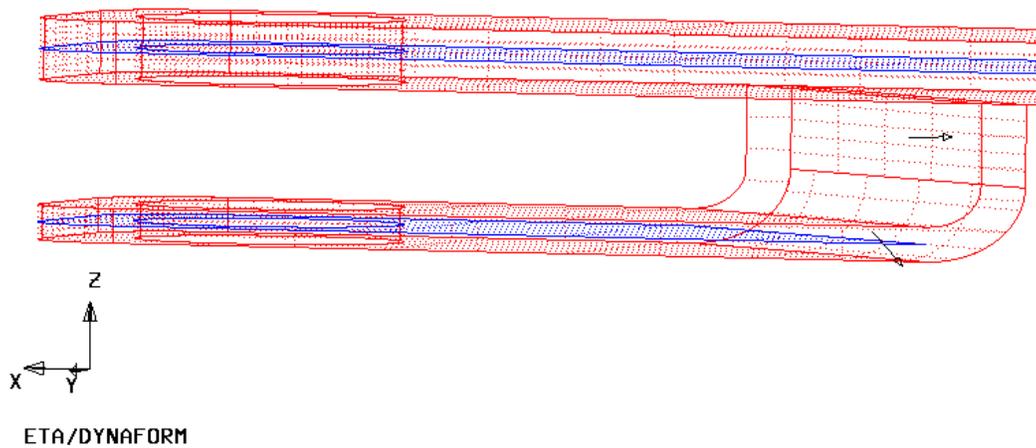


Figure 5.2.10 Surface Offset Direction

After selecting the offset direction, the new offset surface will be created. Please see the Figure 5.2.11.

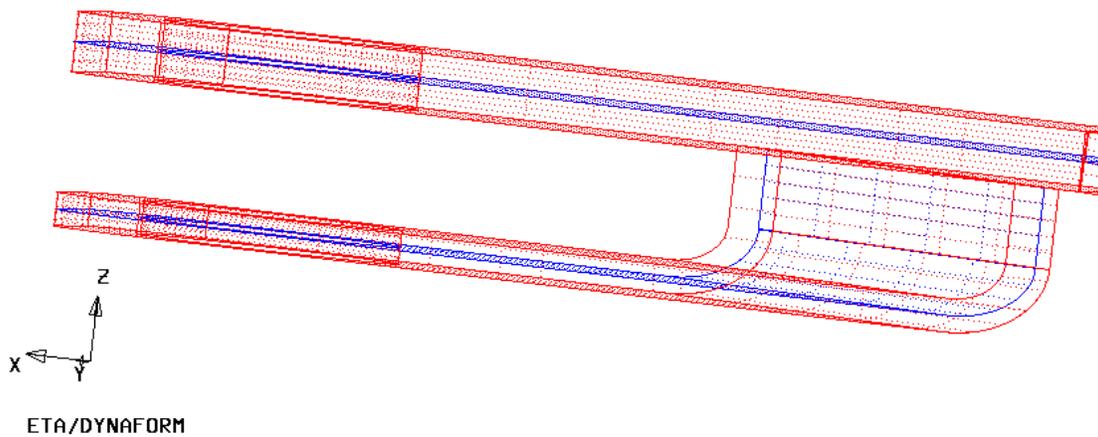


Figure 5.2.11 Create mid surface by offset

## 5.2.8 SEPARATE SURFACE



This function is used to separate the top and bottom surface layers of sheet metal parts. The SEPARATE 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 separated.
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 5.2.12.



Figure 5.2.12 Group Options dialog box

3. User can click GROUP TOP SURFACE to group all top surfaces as illustrated in Figure 5.2.13. The grouped surfaces are highlighted, while the SELECT OPTION dialog box as illustrated in Figure 5.2.17, is displayed.



Figure 5.2.13 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 5.2.14). At this point, Figure 5.2.15 is displayed.

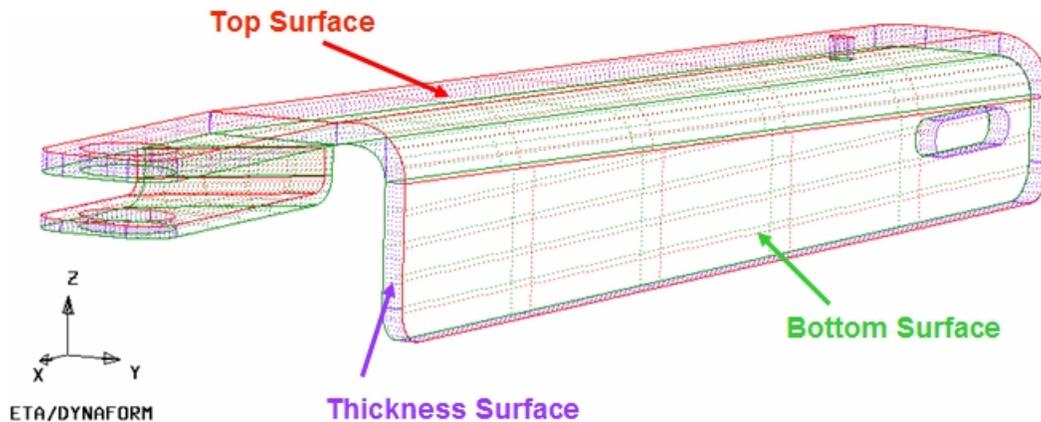


Figure 5.2.14 Group Bottom Surfaces

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 BOTS RF1. The number “1” will be

incremented by 1 for each subsequent grouping operation.

## 5.2.9 MESH TO SURFACE



This function allows the user to mesh to surface, which dialog box is shown in Figure 5.2.15.

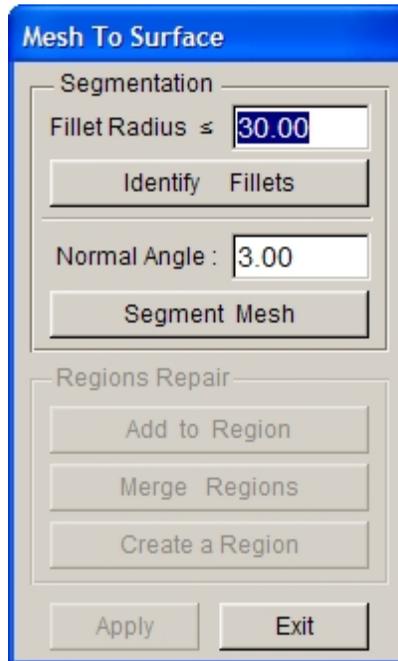


Figure 5.2.15 Mesh to Surface dialog box

- **FILLET RADIUS**

This function is used to identify fillets which radius is equal or less than the input value. The default is 30. The user can enter a different Fillet Radius and then click Identify Fillets to highlight the identified fillet, as shown in Figure 5.2.16. If mesh does not have this fillet, this function needs not to be used.

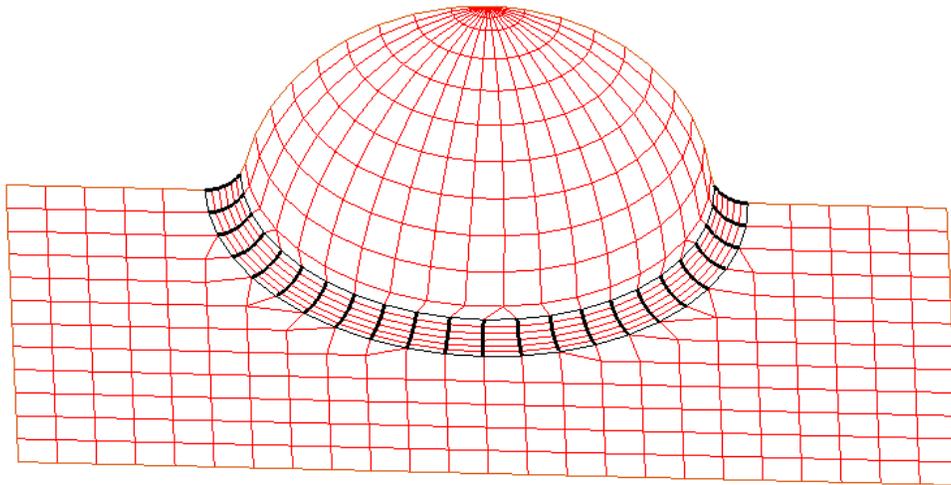


Figure 5.2.16 Identify Fillets

- NORMAL ANGLE

This function is used to control the size of mesh segment which determines the size of the generated surface. The default normal angle is 3 degrees. This normal angle is used to calculate the feature lines in the mesh. The user can enter a different angle value and then click Segment Mesh to segment mesh into regions, as shown in Figure 5.2.17. The program will generate a surface for each region.

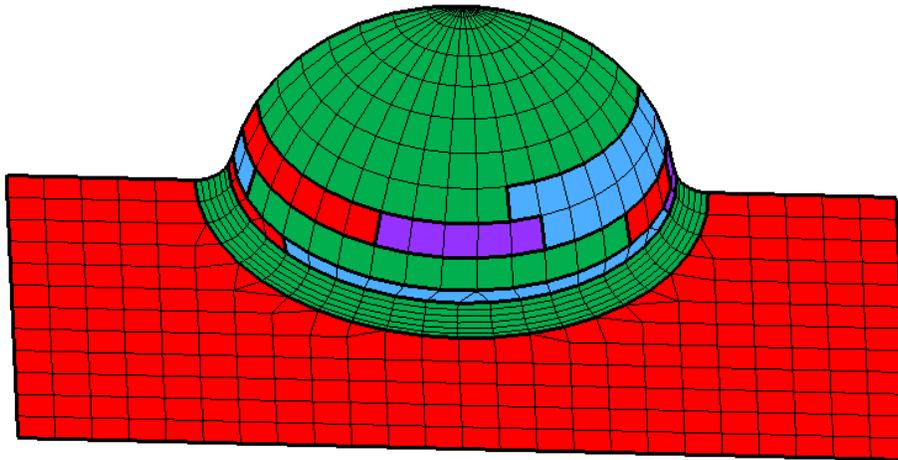


Figure 5.2.17 Segment Mesh

- ADD TO REGION

This function is used to add elements in one or multiple regions to another region. After clicking on this button, Select Element dialog box appears for the user to select elements to be added. Click OK or the middle mouse button to finish the element selection. Then select a region using left mouse click to add the elements. Click the right mouse button without selecting element to exit.

- MERGE REGIONS

This function is used to merge two or more adjacent regions into one. Click the left mouse button to select the regions to be merged and click the middle mouse button to merge. Then click on the right mouse button without selecting surface to exit.

- CREATE A REGION

This function is used to create a new region. After clicking on this button, a Select Element dialog box appears for the user to select the desired elements. Click OK or the middle mouse button to finish element selection. Click the right mouse button without selecting element to exit.

After editing regions, the generated surfaces will be placed in the new GENSURF0 which is created by the program automatically, as shown in Figure 5.2.18.

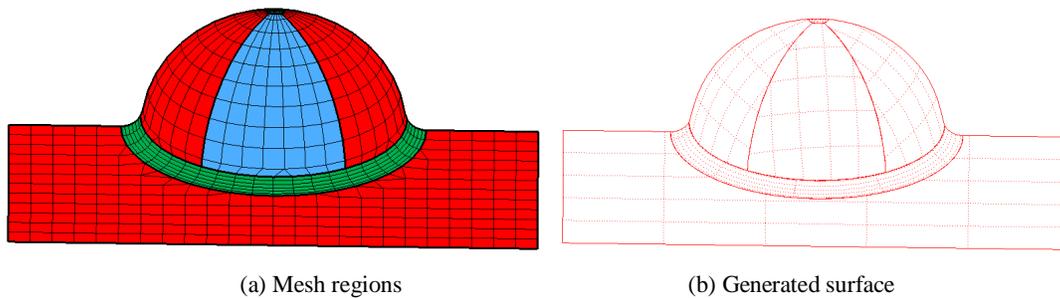


Figure 5.2.18 Mesh to Surface

### 5.2.10 SURFACE INTERSECT



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

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

### 5.2.11 SURFACE SPLIT



This function is used to split a selected surface into two surfaces using a specified line, as shown in Figure 5.2.19.

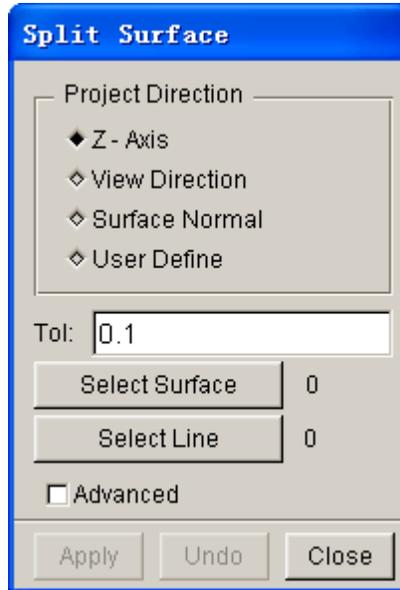


Figure 5.2.19 Split Surface

1. Select the split direction. There are four options for the user to define the direction of the split surface.

#### **Z-AXIS**

Define Z-axis as the surface split direction.

#### **VIEW DIR**

Define the current view direction as the surface split direction.

#### **SUR NORMAL**

Define the surface normal of the selected surface as the surface split direction.

#### **USER DEFINE**

The user can define the surface split direction. Once this option is selected, the program will pop up a dialog box for the user to define direction vector. Refer to Section 5.1.4, SHORTEST CURVE for more information.

2. Enter the TOL (gap tolerance) value. Gap tolerance is the allowable maximum gap between the adjacent surfaces. If the gap value between the adjacent surfaces is greater than gap tolerance, the adjacent surfaces are discontinuous.
3. Click SELECT SURFACES to select surfaces to be split.
4. Click SELECT LINE button to select the split line. If there is no split line, the user can select ADVANCED option to create a desired split line. Once the ADVANCED option is selected, the program will pop up a row of buttons to create line, as shown in Figure 5.2.20.

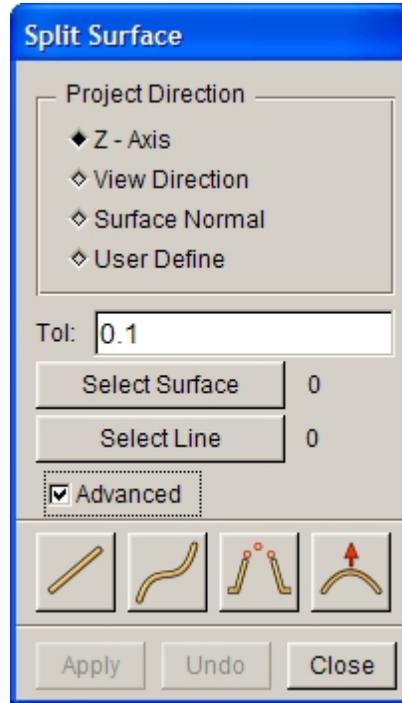


Figure 5.2.20 Create Split Line

-  — Create the auxiliary straight line.
-  — Create the auxiliary spline curve.
-  — Connect two lines smoothly.
-  — Edit line function.

Once one of the above functions is selected, the program will pop up the Create Line dialog box, as shown in Figure 5.2.21.

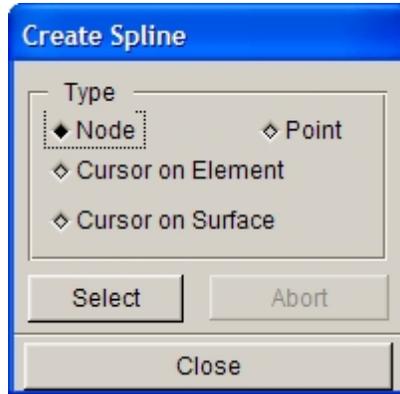


Figure 5.2.21 Create Spline dialog box

- **NODE:** Create line by selecting nodes on the screen.
- **POINT:** Create line by selecting points on the screen.
- **CURSOR ON ELEMENT:** Create line by mouse clicking on locations on element.
- **CURSOR ON SURFACE:** Create line by mouse clicking on locations on surface.

The user can also create spline line by using the combination of the above four methods. First, select one method and click Select button. Then select several points on the screen and choose another method. Finally, select some other points on the screen and click the middle mouse button to confirm the operation.

### 5.2.12 DELETE SURFACE



This function allows the user to remove the selected surface from the database. Refer to Section 4.3.3, PART/ADD SURFACE, for information about selecting surfaces.

### 5.2.13 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 trim.
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**.

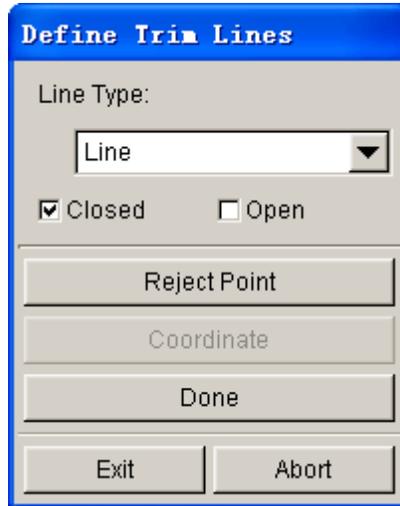


Figure 5.2.22 Define Trim Lines

4. **DEFINE TRIM LINE** as illustrated in Figure 5.2.22 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 5.2.23 appears.

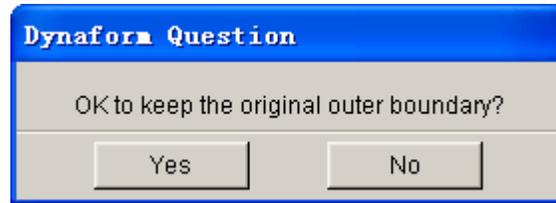


Figure 5.2.23 DYNAFORM Question dialog box

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.

## 5.2.14 UNTRIM SURFACE

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

## 5.2.15 EXTEND SURFACE

This function allows the user to extend surface. Clicking on this button, if multiple surfaces appear in display area, the program pops up a Select Surface dialog box. Once a surface is selected, the control mesh for the selected surface will be highlighted, and four highlighted points will appear in four directions of the surface boundary, as shown in Figure 5.2.24. If there is only one surface in the display area, the program will directly pop up the Surface Extend dialog box.

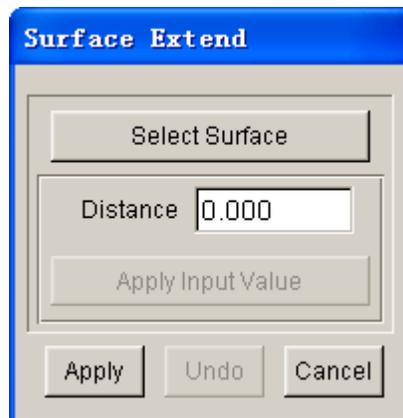


Figure 5.2.24 Surface Extend

### SELECT SURFACE

This function allows the user to select a surface to extend.

### DISTANCE

This function is used to display or enter the extended distance. The user can extend surface via the four highlighted points on boundary of the selected surface. Point the cursor to an arbitrary point to display a red arrow, as shown in Figure 5.2.25. Then hold and drag the left mouse button to extend the surface towards this direction. The extended distance of surface will be shown in the Distance field. The user can also input a specific distance manually in the Distance field. After entering a specific distance value, click **APPLY INPUT VALUE** to confirm.

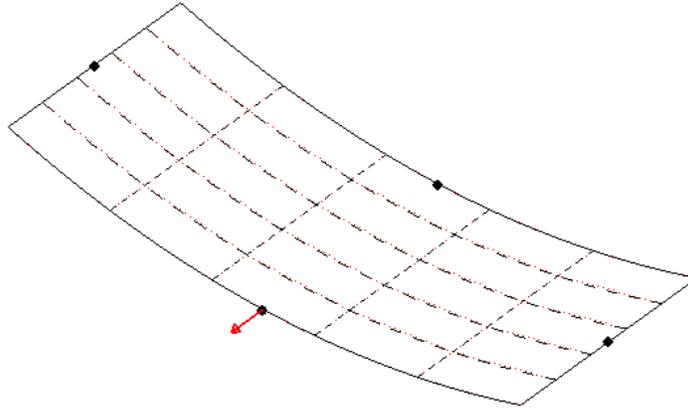


Figure 5.2.25 Surface Extend Direction

*Note: When using this function, the surfaces to be extended must be untrimmed surfaces for outer boundary.*

## 5.2.16 TRANSFORM SURFACE



This function allows the user to translate, rotate, mirror and scale the surface, and transform surface according to the coordinates. The interface is shown in Figure 5.2.26. The operation of this function is similar to those described in Section 5.1.18, TRANSFORM.

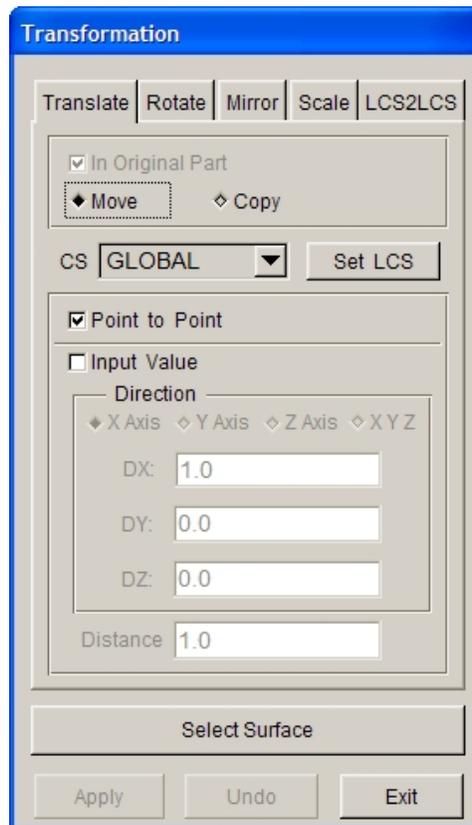


Figure 5.2.26 Transformation Interface

### 5.2.17 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 CREATE SECTION LINE.

### 5.2.18 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 automatically after a surface is selected.

### 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 **PRE-PROCESS/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. If the result after filling is not correct, please change the normal direction of the surface.

### 5.2.20 BOUNDARY LINE



This function allows the user to create line along the boundary of finite element mesh or surface. Refer to Section 5.1.5 for more information.

### 5.2.21 DUPLICATE SURFACE CHECK



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

### 5.2.22 CHECK SURFACE



This function allows the user to check TINY / SLIVER SURFACE, SURFACE BOUNDARY, CONTROL POINTS DISTRIBUTION, SPIKE / CUT SURFACE, SURFACE G0 CONTINUOUS and SURFACE G1 CONTINUOUS. The dialog box is illustrated in Figure 5.2.27.

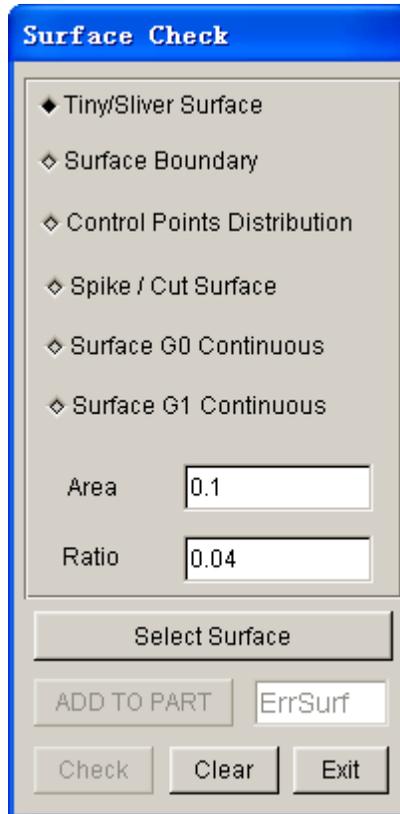


Figure 5.2.27 Tiny/Sliver Surface Check

- **TINY / SLIVER SURFACE**

This function allows the user to check for tiny/sliver surface. The user can enter values in AREA and RATIO data fields to define tiny surface and sliver surface, as shown in Figure 5.2.27. The surfaces with area less than 0.1 square millimeters are defaulted to be tiny surfaces. Surfaces which ratio between surface area and surface perimeter is less than 0.04 are sliver surfaces. The program will highlight the tiny/sliver surfaces and display their surface numbers in the message window.

- **SURFACE BOUNDARY**

This function allows the user to check whether there is self-intersection of surface boundary. The program will highlight the surfaces and display their surface numbers in the message window.

- **CONTROL POINTS DISTRIBUTION**

This function allows the user to check the distribution of control points in the surface. The program will highlight the surfaces and display their surface numbers in the message window.

- **SPIKE / CUT SURFACE**

This function allows the user to check for spike/cut surface. The user can enter DIST TOL and ANGLE TOL to define spike/cut surface. The defaults are 0.5 and 15, as shown in Figure 5.2.28. DIST TOL is the maximum distance between short edge and long edge. ANGLE TOL is the angle between two edges. The program will highlight the surfaces and display their surface numbers in the message window.

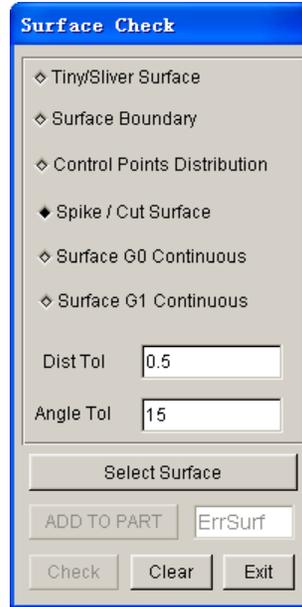


Figure 5.2.28 Spike / Cut Surface

- **SURFACE G0 CONTINUOUS**

This function allows the user to check the continuity of surface position. The user can enter GAP TOL to define the continuity surface position. The default is 0.5, as shown in Figure 5.2.29. The program will highlight the surfaces and display their surface numbers in the message window.

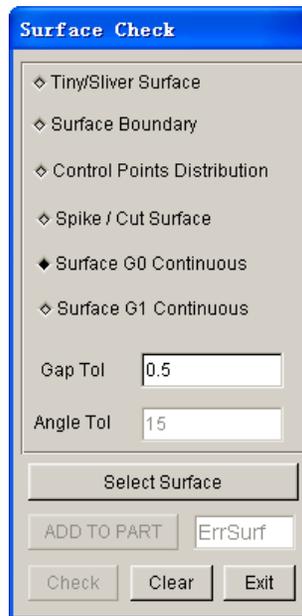


Figure 5.2.29 Surface G0 Continuous

- **SURFACE G1 CONTINUOUS**

This function allows the user to check Surface G1 Continuous. The user can enter GAP TOL and ANGLE TOL to define the continuity of surface G1. The defaults are 0.5 and 3, as shown in Figure 5.2.30. The ANGLE TOL is the angle between normal directions of surface at the common surface boundary. The program will highlight the surfaces and display their surface numbers in the message column.

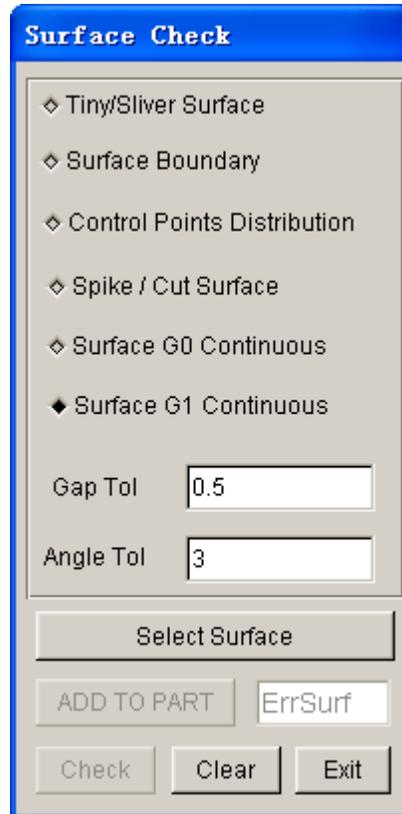


Figure 5.2.30 Surface G1 Continuous

- **ADD TO PART**

This function allows the user to add the error surfaces to ERRORSURF part created by the program automatically. The user can also change the default part name manually to create a new part.

- **CHECK**

Execute check operation.

- **CLEAR**

Clear the highlighted surfaces and surface numbers.

### 5.2.23 SHOW SURFACE



This function allows the user to highlight the selected surface and its UV lines. The dialog box is illustrated in Figure 5.2.31.



Figure 5.2.31 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).

### 5.3 ELEMENT (CTRL+E)

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

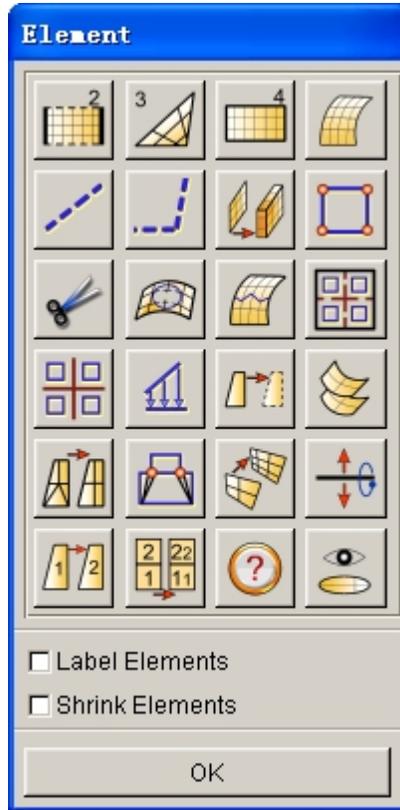


Figure 5.3.1 Element Options Menu

- **LABEL ELEMENTS**  
Toggle ON/OFF to show/hide the element labels.
- **SHRINK ELEMENTS**  
Reduce 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.

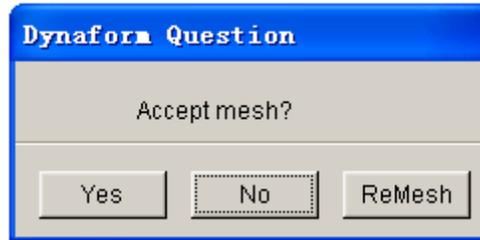


Figure 5.3.2 DYNIFORM Question dialog box

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 are  $N$  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 except it is required to select 4 lines in a sequence to form a closed region.

### 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.

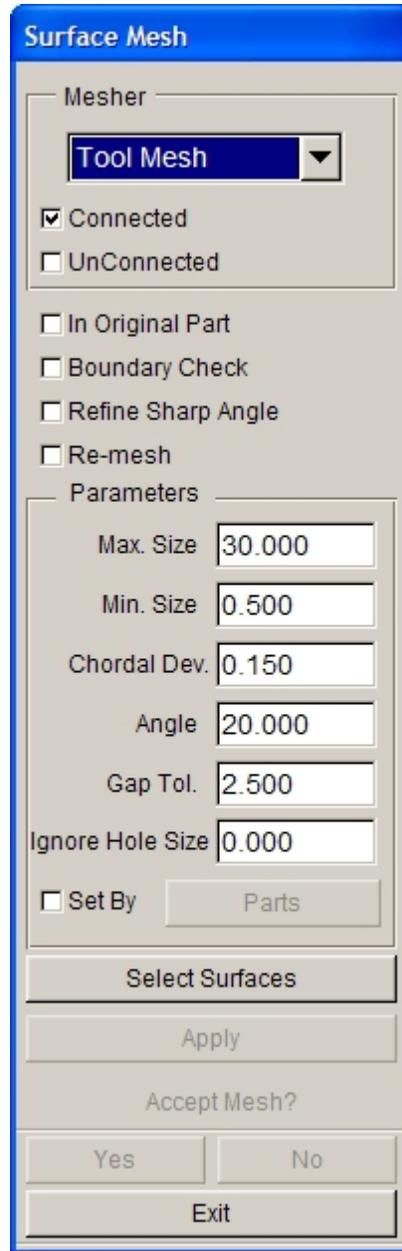


Figure 5.3.4 Surface Mesh dialog box

- Select a type of tool: CONNECTED or UNCONNECTED TOOL 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).

UNCONNECTED 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.
- Toggle on/off **RE-MESH**. Once this option is selected, if the user wants to re-mesh the corresponding surface of the meshed local region, the mesh boundary is still connected to the boundary nodes in the region which is not re-meshed. Therefore, when the surface of the local region needs to be modified after all the meshes have been meshed, the user does not have to re-mesh all the meshes.
- Enter the numbers of **MAXIMUM SIZE**, **MINIMUM SIZE**, **ANGLE** and **CHORDAL DEVIATION** in the data field.

**MAXIMUM SIZE**

Control the maximum element size.

**MINIMUM SIZE**

Control the minimum element size.

**ANGLE**

Control the inclination of the adjacent elements.

**CHORDAL**

Control the number of elements at the radius.

**GAP TOL**

Define the allowable maximum gap between the adjacent surfaces. If the gap between adjacent surfaces is greater than gap tolerance, these two adjacent surfaces are discontinuous.

**IGNORE HOLE SIZE**

Ignore the holes of the surface during meshing if the hole size is less than the input criteria.

- Click **SELECT SURFACES** button. Refer to Section 4.3.3, **PART/ADD SURFACE**, for more information.
- Click **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.

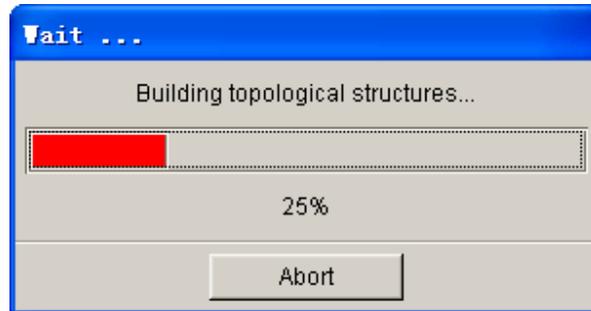


Figure 5.3.5 Building Topological Structures

- 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.

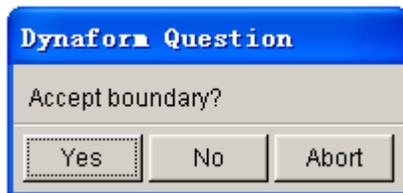


Figure 5.3.6 DYNAFORM Question

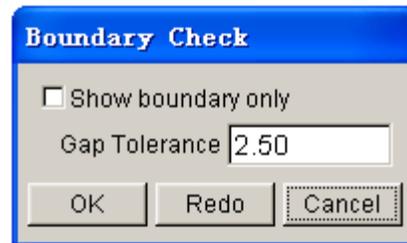


Figure 5.3.7 Boundary Check

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.

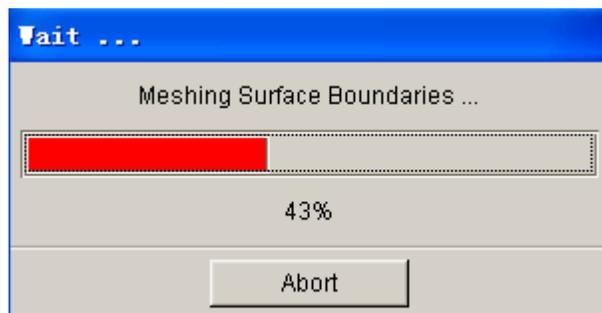


Figure 5.3.8 Meshing Surface Boundaries

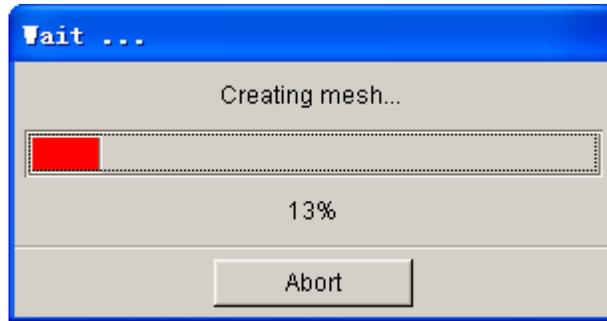


Figure 5.3.9 Creating Mesh

- 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  
Check 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 merge gaps between elements and removes free nodes from database.
- IGNORE HOLE SIZE  
Fill 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

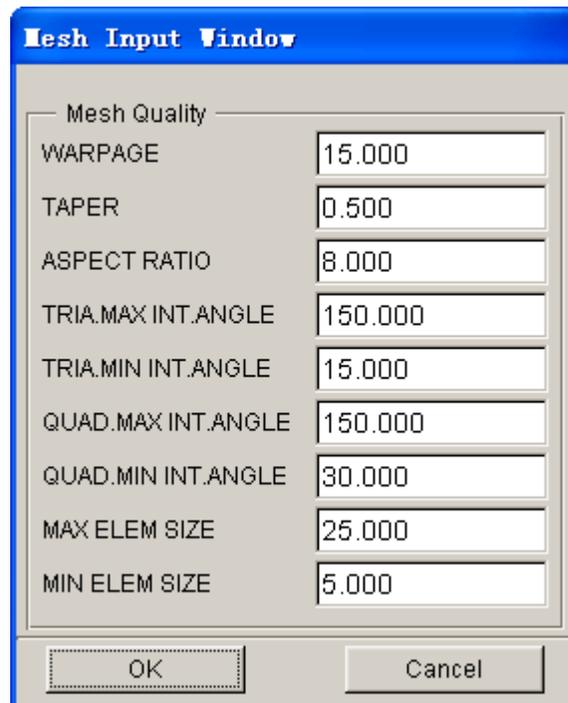


Figure 5.3.11 Mesh Quality dialog box

If TRIANGLE MESHER is selected:

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

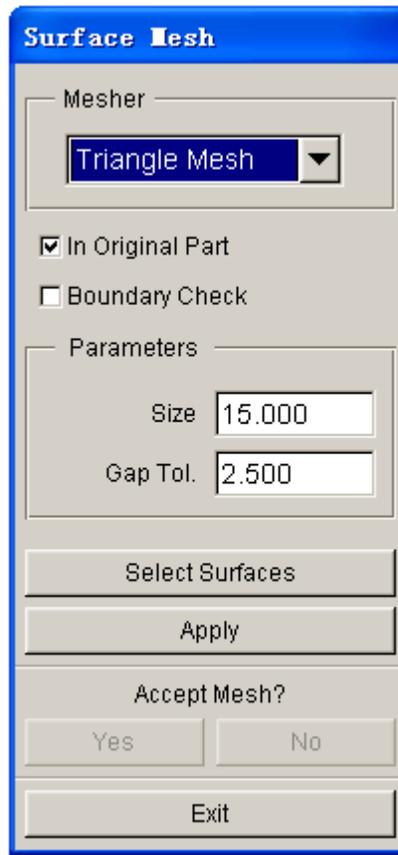


Figure 5.3.12 Triangle Mesh dialog box

### 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 Control

**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.

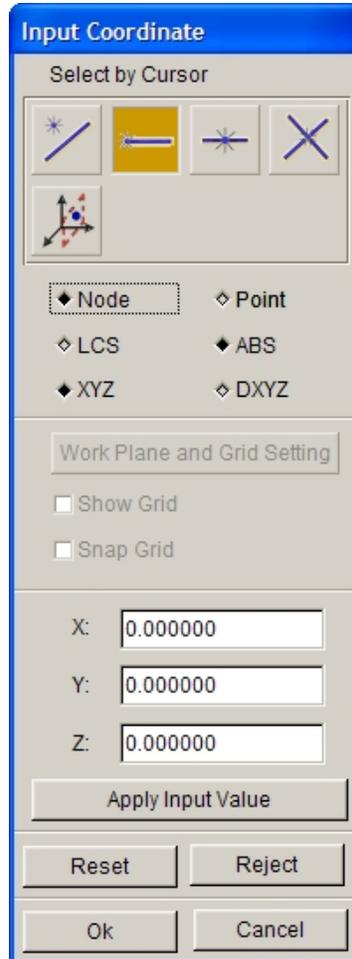


Figure 5.3.15 Input Coordinate dialog box

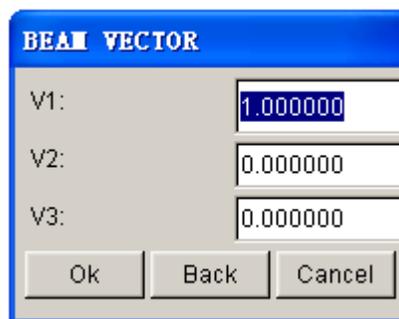


Figure 5.3.16 Beam Vector dialog box

### 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 thick shell or solid elements based on the selected shell elements. After selecting the shell elements, the program will pop up the Drag Mesh Option dialog box, as shown in Figure 5.3.17.



Figure 5.3.17 Drag Mesh Option dialog box

The user can select THICK SHELL to drag the plate (shell) elements to thick shell elements; select SOLID to drag the plate (shell) elements to solid elements. Select DELETE SOURCE ELEMENT to delete the source plate (shell) elements. The user should enter THICKNESS value to define thickness of the drag elements. Once SOLID option is selected, LAYERS is activated for the user to enter a value to define the number of layers of the generated solid elements.

**NOTE:** In eta/DYNAFORM, only some 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.

### 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 5.3.18.



Figure 5.3.18 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

The above 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 DONE, 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 DONE, 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 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.

### 5.3.10 TRIM ELEMENT



This function allows the user to trim elements using one or multiple closed or open trim lines. The trim process is performed in accordance with the defined vector / direction. The user can determine to keep the trimmed elements or not. The trimmed elements will be included in a new part named TRIMOUT. The Element Trimming dialog box is illustrated in Figure 5.3.19.

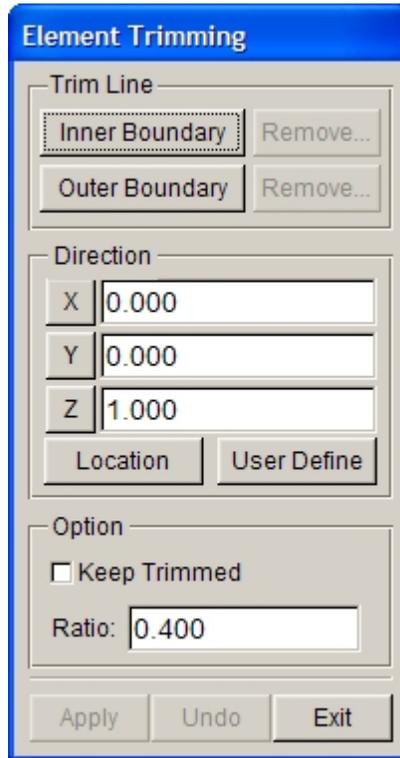


Figure 5.3.19 Element Trimming dialog box

- INNER BOUNDARY

This function allows the user to select one or multiple closed inner trim boundary line.

- OUTER BOUNDARY

This function allows the user to select one or multiple closed outer trim boundary line. If the open outer trim boundary line is selected, only one line can be trimmed each time.

- REMOVE

This function allows the user to remove the selected trim lines. Click this button to display a dialog box as illustrated in Figure 5.3.20. In the list of this dialog box, the user can remove the unnecessary trim lines.

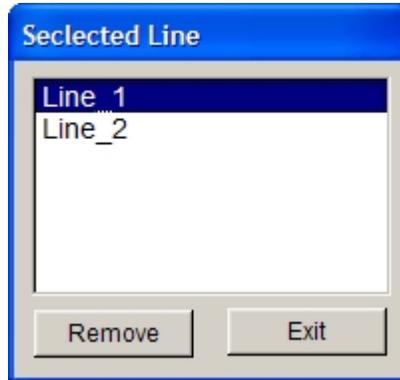


Figure 5.3.20 Selected Line List

- LOCATION

This function allows the user to move the location of local coordinates without changing the projection direction.

- USER DEFINE

This function allows the user to define the projection direction. The user must define a vector (projection direction) for trim line. Clicking on this button, the program will pop up a Define Vector dialog box. Detailed operation about defining vector refers to Section 6.1.1.5 Edit Surface.

- KEEP TRIMMED

This function allows the user to keep the trimmed elements. If this option is not selected, the program will delete the trimmed elements.

- RATIO

This function allows the user to set the trimming ratio. The range of trimming is from 0.2 to 0.45, which restricts the size of small elements generated during trimming. Large elements are generated with large ratio and small elements are generated with small ratio. The default is 0.4.

After selecting one or multiple trim lines for element trimming, when the user selects trim lines again, the program will pop up a DYNAFORM Question dialog box as shown in Figure 5.3.21. The user can click YES to continue the trimming of blank, but the previous trimming operation cannot be restored. If NO is selected, the user cannot trim the element unless re-entering the Element Trimming dialog box.

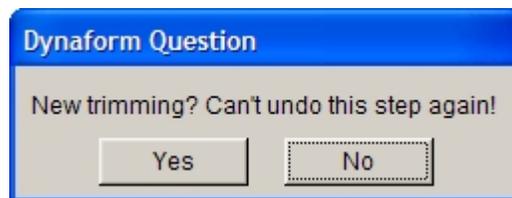


Figure 5.3.21 DYNAFORM Question dialog box

### 5.3.11 LANCE ELEMENT



This function allows the user to lance the selected elements along one or several specified path. The Element Lancing dialog box is illustrated in Figure 5.3.22.

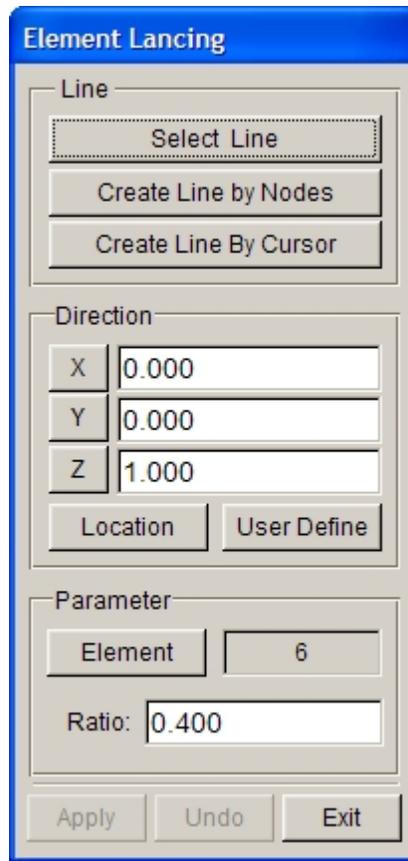


Figure 5.3.22 Element Lancing dialog box

- SELECT LINE

This function allows the user to select one or several existing lines as cutting lines.

- CREATE LINE BY NODES

This function allows the user to create a line segment by selecting nodes. The created line segment will be projected onto the selected element along the defined W axis and be regarded as the lancing path.

- CREATE LINE BY CURSOR

This function allows the user to create cutting lines by clicking the cursor on element.

- LOCATION

This function allows the user to move the location of local coordinates without changing the projection direction.

- USER DEFINE

This function allows the user to define the projection direction. The user must define a vector (projection direction) for trim line. Clicking on this button, the program will pop up a Define Vector dialog box. Detailed operation about defining vector refers to Section 6.1.1.5 Edit Surface.

- ELEMENT

This function allows the user to select lancing element area.

- RATIO

This function allows the user to set the trimming ratio. The range of trimming is from 0.2 to 0.45, which restricts the size of small elements generated during trimming. Large elements are generated with large

ratio and small elements are generated with small ratio. The default is 0.4.

### 5.3.12 COARSE ELEMENT



This function allows the user to coarsen two, three 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 two, three or four elements by using the options in the SELECT ELEMENT dialog box. Once the elements are 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.

### 5.3.13 SPLIT ELEMENT



This function divides any SHELL or SOLID elements into multiple elements based on the options described below. The dialog box is illustrated in Figure 5.3.23.

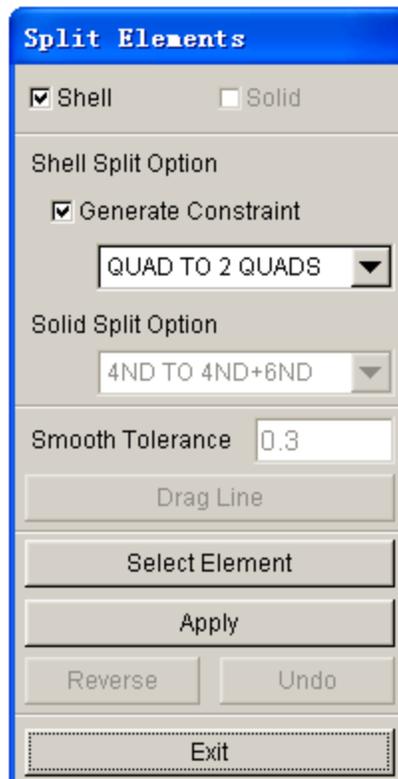


Figure 5.3.23 Split Elements dialog box

## 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 5.3.24.

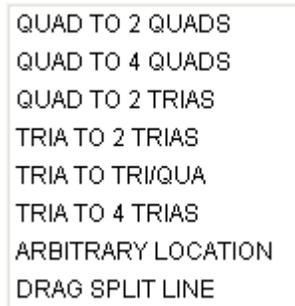


Figure 5.3.24 Shell Split Options

**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.

### 5.3.14 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 local 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.*

### 5.3.15 TRANSFORM ELEMENT



This function allows the user to translate, rotate, mirror and scale the element, and transform element according to the coordinates. The interface is shown in the Figure 5.3.25. The operation of this function is similar to those described in Section 5.1.18, TRANSFORM LINE.

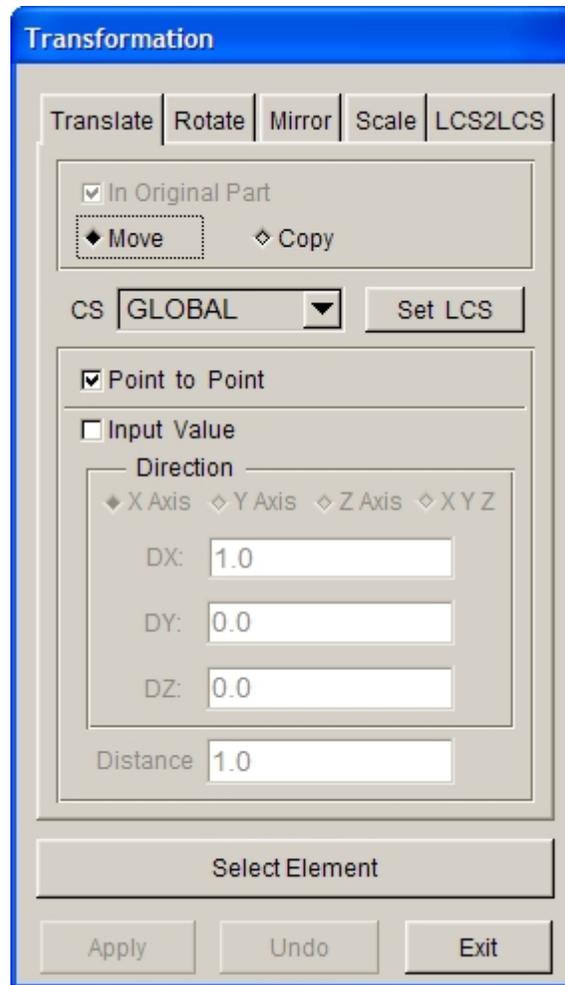


Figure 5.3.25 Transformation

### 5.3.16 OFFSET ELEMENT



This function allows the user to create a sequence of duplicate elements by offset (along normal direction).

### 5.3.17 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.

### 5.3.18 AUTO 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 is selected as it can change the

geometry of the tooling. Several user-defined control options are given for identifying and repairing the inferior mesh. After selecting elements, the REPAIR PARAMETERS dialog box as illustrated in Figure 5.3.26 is displayed.

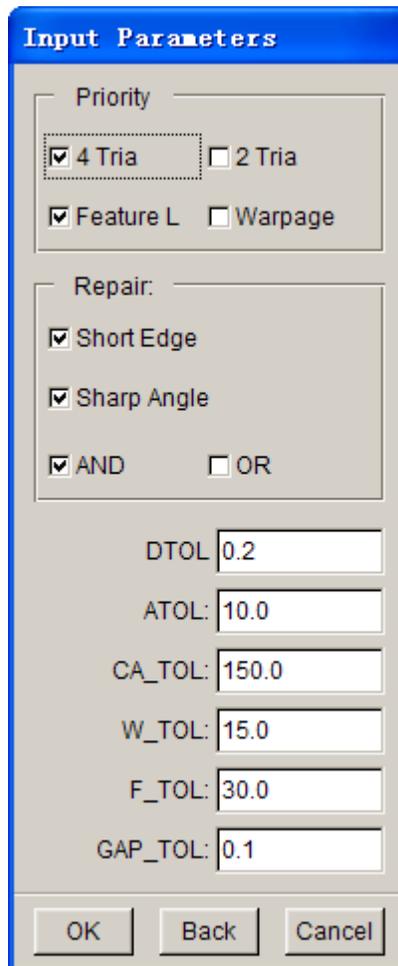


Figure 5.3.26 Input Parameters dialog box

### **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 SHARP 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.

### 5.3.19 AUTO REMESH



This function allows the user to change mesh parameters and re-mesh without original surfaces. The program will mesh the model according to the new mesh parameters. The Auto Re-mesh dialog box is illustrated in Figure 5.3.27.

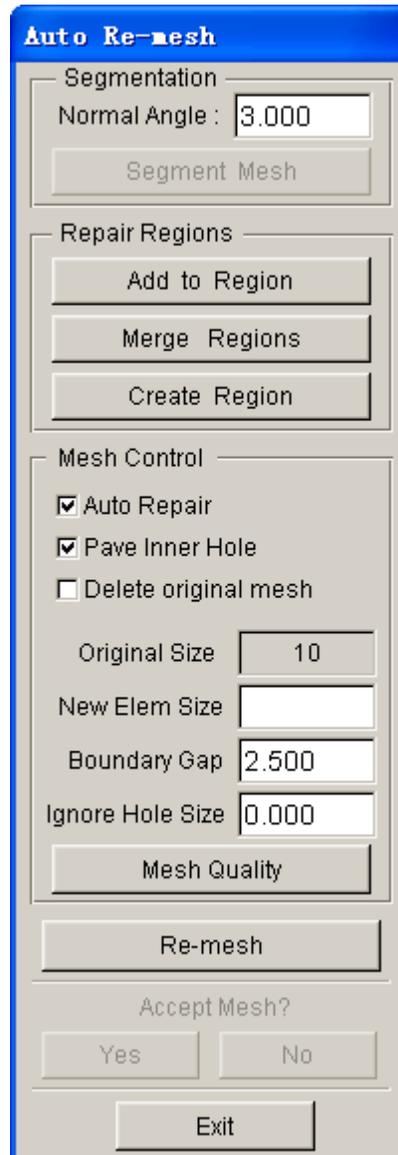


Figure 5.3.27 Auto Re-mesh dialog box

**NORMAL ANGLE:** normal angle between the adjacent elements.

**SEGMENT MESH:** segment mesh according to the normal angle and mark with different colors.

**REPAIR REGION:** Allow the user to add elements in a region to other regions, merge elements in several regions and select elements to create a new region.

**ADD TO REGION:** Allow the user to add elements in a region to other regions. Select elements by left mouse click and click middle mouse button to complete selection. Then select a region to add the elements to.

**MERGE REGIONS:** Allow the user to merge two or more adjacent regions. After selecting the regions to be merged, click on the middle mouse button to complete this operation.

**CREATE REGION:** Allow the user to select elements to create a new region. After selecting elements, click on the middle mouse button to complete this operation.

**MESH CONTROL:** Refer to Section 5.3.4 for detailed description.

**REMESH:** After the new mesh size is entered, click on this button to remesh.

### 5.3.20 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.

### 5.3.21 CHANGE ELEMENT NUMBER



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

### 5.3.22 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 5.3.28 appears.

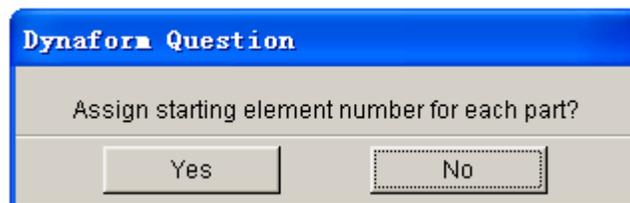


Figure 5.3.28 Starting Element Number Question

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).

### 5.3.23 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.

### 5.3.24 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 xxx NOT FOUND IN DATABASE***

## 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.

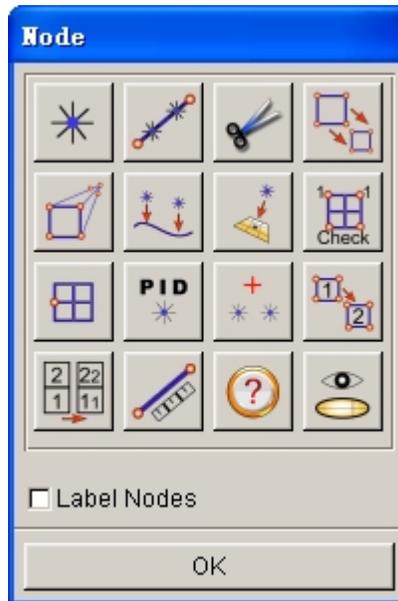


Figure 5.4.1 Node dialog box

- **LABEL NODES ON**  
Toggle 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 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.

### 5.4.4 TRANSFORM NODE



This function allows the user to translate, rotate, mirror and scale the node, and transform node according to the coordinates. The interface is shown in Figure 5.4.2. The operation of this function is similar to those described in Section 5.1.18, TRANSFORM LINE.

*Note: In node transform types, only Move can be activated, Copy is disabled.*

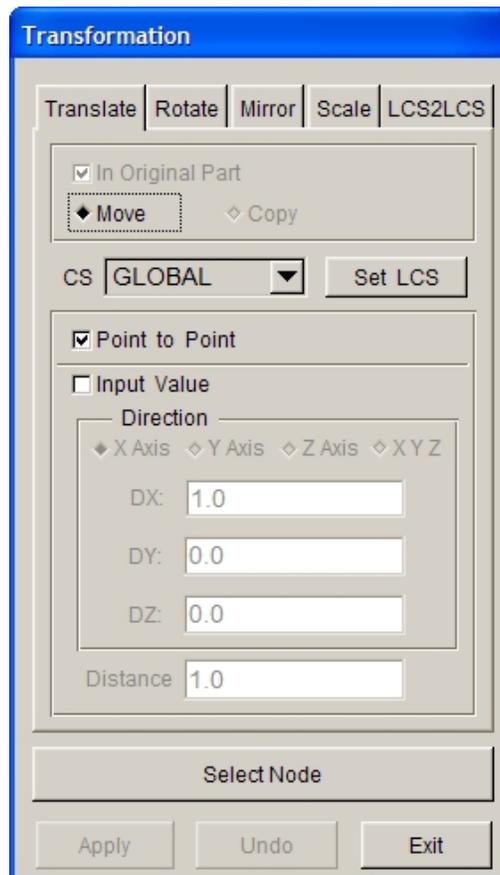


Figure 5.4.2 Transformation Interface

### 5.4.5 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.

### 5.4.6 ALIGN



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

### 5.4.7 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.
- An eta/DYNAFORM Question dialog box as illustrated in Figure 5.4.3 is displayed.

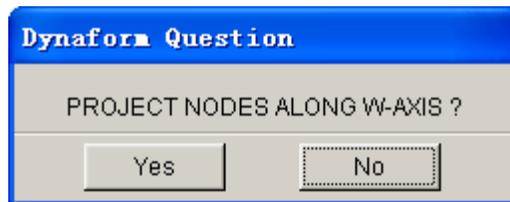


Figure 5.4.3 Project Node Question dialog box

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.

## 5.4.8 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 xxxx NODES FOR DUPLICATE NODES***

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

***xxxx 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 xxxx NODES FOR DUPLICATE NODES***  
***NO DUPLICATE NODE FOUND***

### 5.4.9 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.

### 5.4.10 PART CONNECT



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

### 5.4.11 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.

### 5.4.12 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.



### 5.4.13 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.

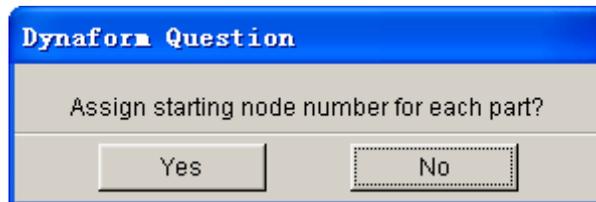


Figure 5.4.4 Starting Node Number Question dialog box

- An 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***

#### 5.4.14 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.

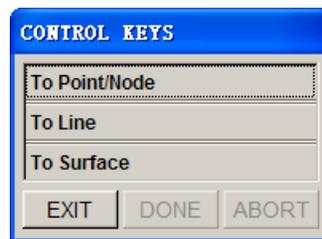


Figure 5.4.5 Measure Distance dialog box

#### 5.4.15 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.

### 5.4.16 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.

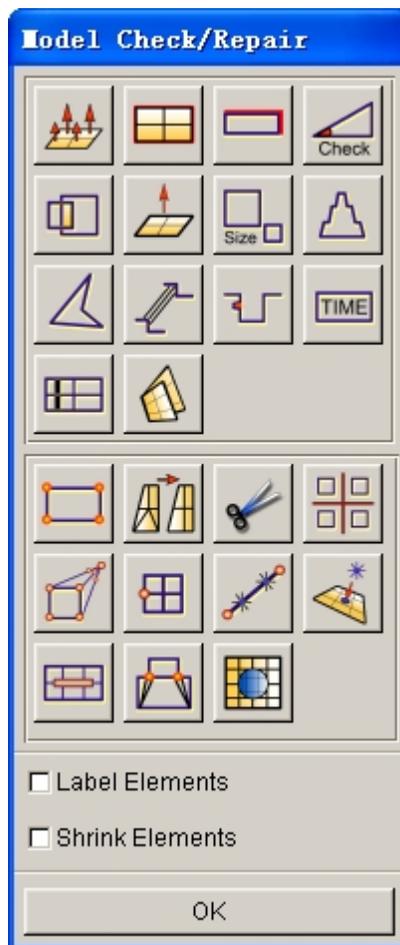


Figure 5.5.1 Model Check & Repair dialog box

## 5.5.1 MODEL CHECK

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

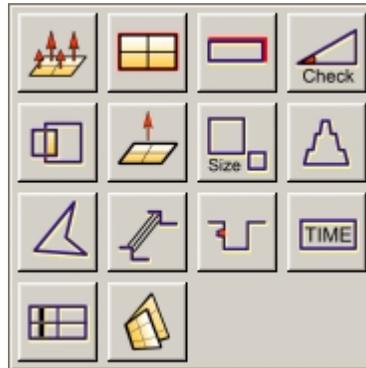


Figure 5.5.2 Model Check

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 ACTIVE PARTS and CURSOR PICK PART. eta/DYNAFORM will display the normal direction of the selected element.
- An eta/DYNAFORM QUESTION dialog box as illustrated in Figure 5.5.3 is displayed.

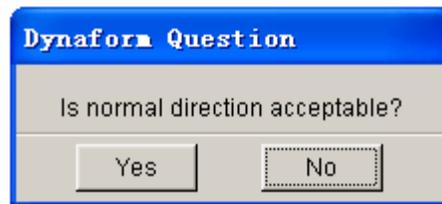


Figure 5.5.3 DYNAFORM Question dialog box

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.

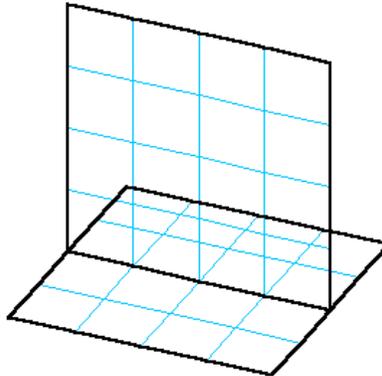


Figure 5.5.4 Surface Boundaries

**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. 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. An eta/DYNAFORM QUESTION dialog box as illustrated in Figure 5.5.5 is displayed.

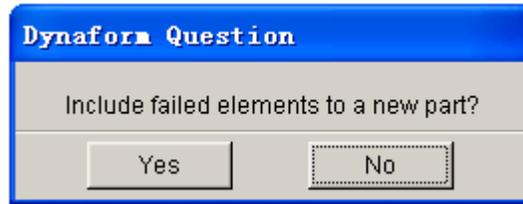


Figure 5.5.5 DYNAFORM Question dialog box

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

***xxxx ELEMENTS FAILED CHECK***

2. At same time, an 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.

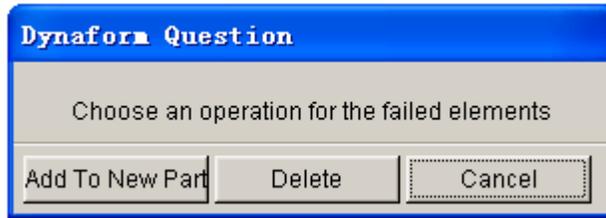


Figure 5.5.6 DYNAFORM Question dialog box

### 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, an eta/DYNAFORM QUESTION dialog box as illustrated in Figure 5.5.7 is displayed.

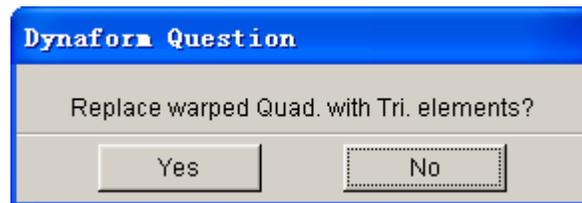


Figure 5.5.7 DYNAFORM Question dialog box

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 plot 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 described in Section 5.1.7.

### 5.5.1.14 MESH SURFACE DEVIATION



This function allows the user to measure the deviation between surface and element. The dialog box is illustrated in Figure 5.5.8.

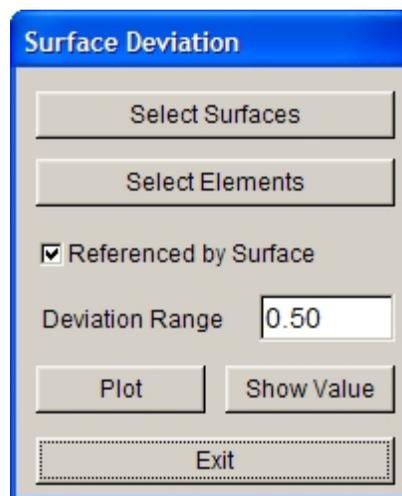


Figure 5.5.8 Surface Deviation dialog box

The user can select the surface and element to be measured and then define DEVIATION RANGE. Finally, click on PLOT to display the deviation plot, as shown in Figure 5.5.9. The user can also click on SHOW VALUE button and then display the deviation value by clicking nodes with cursor, as shown in Figure 5.5.10.

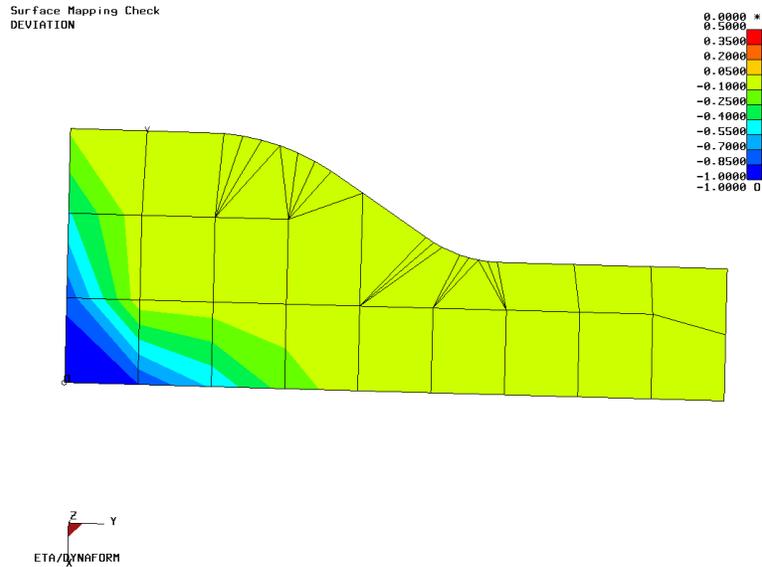


Figure 5.5.9 Deviation Plot

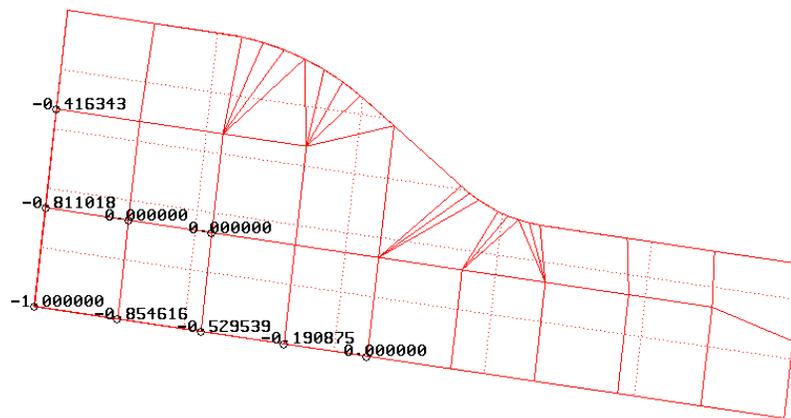


Figure 5.5.10 Display Deviation Value

### 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.

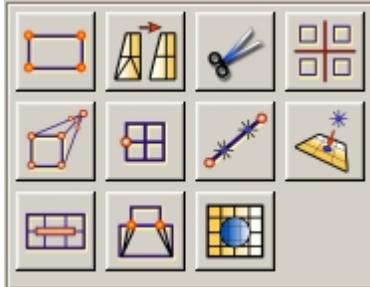


Figure 5.5.11 Model Repair

## 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.

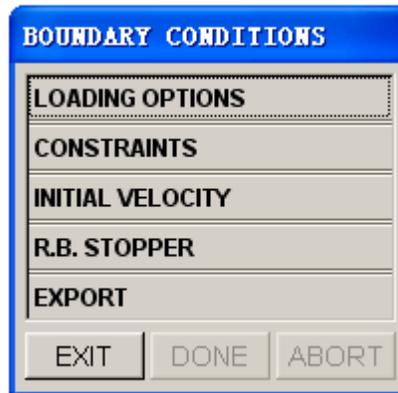


Figure 5.6.1 Boundary Conditions dialog box

### 5.6.1 LOADING OPTIONS

The LOAD SET dialog box is illustrated in Figure 5.6.2.

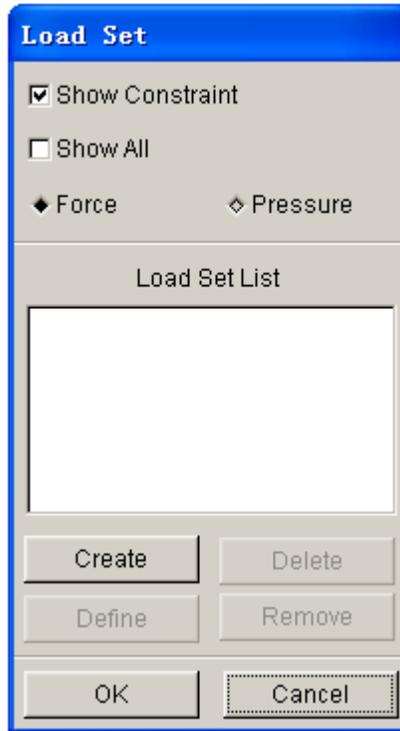


Figure 5.6.2 Load Set dialog box

**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.

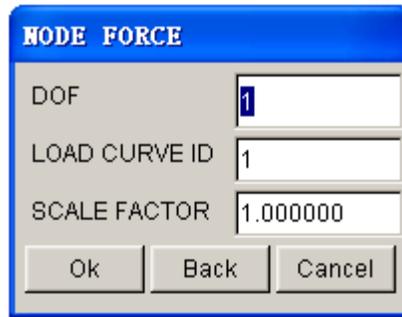


Figure 5.6.3 Define Node Force

**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 11.12.1, 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 5.6.3 to select nodes. Refer to Figure 5.4.2 for information about selecting nodes.
  - Select CANCEL in Figure 5.6.3 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.

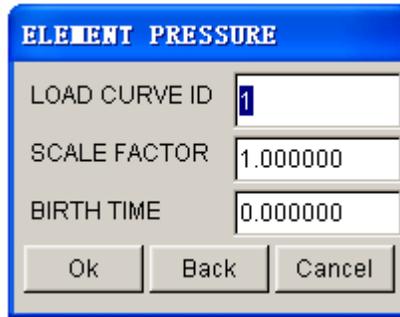


Figure 5.6.4 Element Pressure dialog box

### 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

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

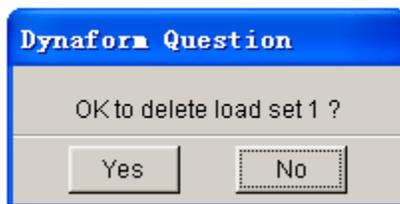


Figure 5.6.5 DYNAFORM Question dialog box

Select YES to delete the last selected load set.

Select NO to reject the choice.

### DEFINE

Include 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 CONSTRAINTS 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.

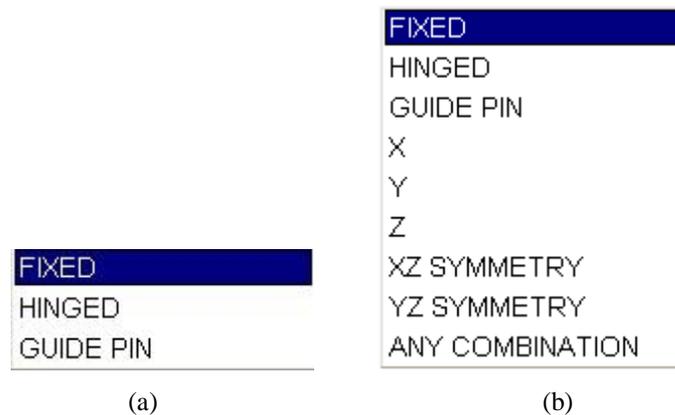


Figure 5.6.7 Type of Constraint (a) Basic (b) Advanced

- 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.

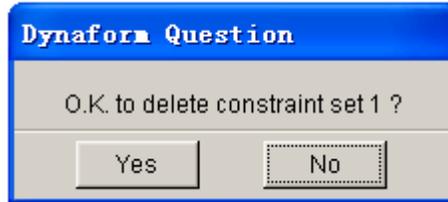


Figure 5.6.8 DYNAFORM Question dialog box

### 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.

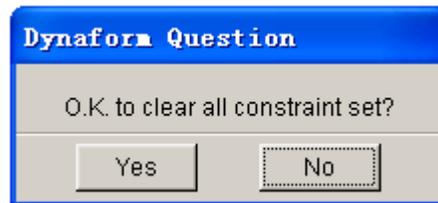


Figure 5.6.9 DYNAFORM Question dialog box

Select YES to remove all constraints.

Select NO to reject the choice.

**Note:** *the DOF of constraints are all in the current coordinate system.*

### 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.

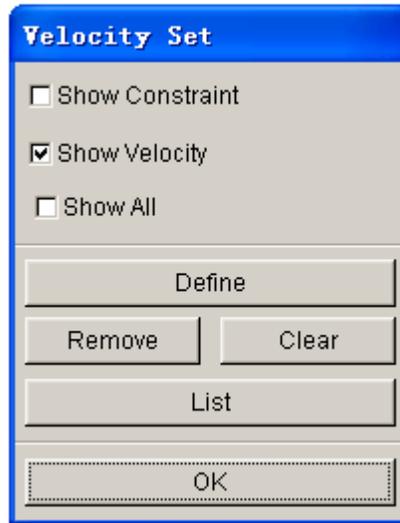


Figure 5.6.10 Velocity Set dialog box

#### 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 INITIAL VELOCITY dialog box as illustrated in Figure 5.6.11.
- Select nodes.

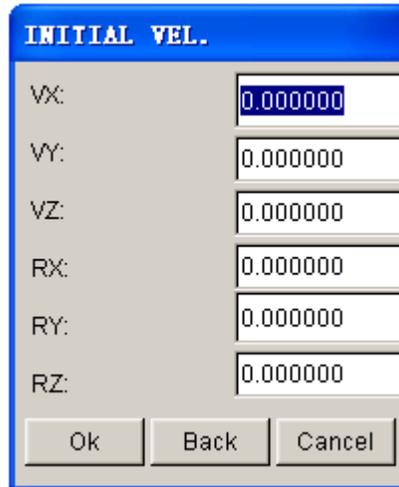


Figure 5.6.11 Initial Velocity dialog box

**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.

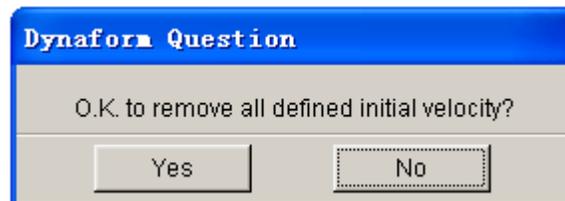


Figure 5.6.12 DYNAFORM Question dialog box

**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.

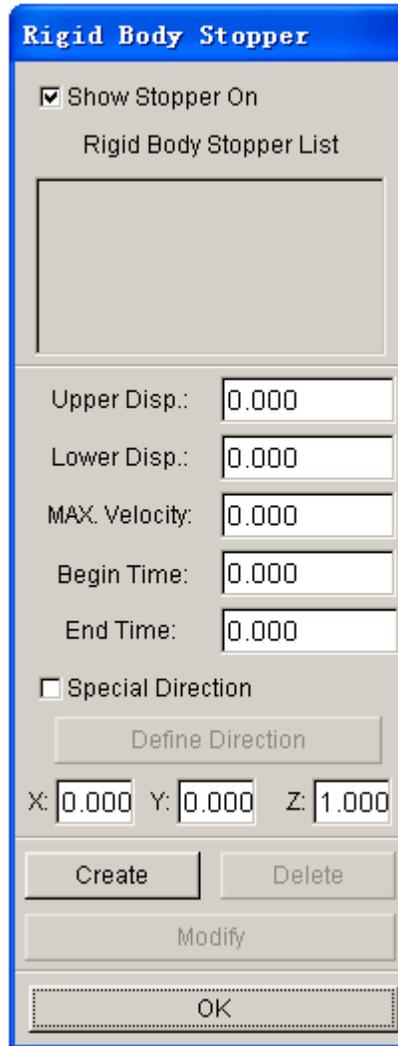


Figure 5.6.13 Rigid Body Stopper dialog box

- 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 5.6.14 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.

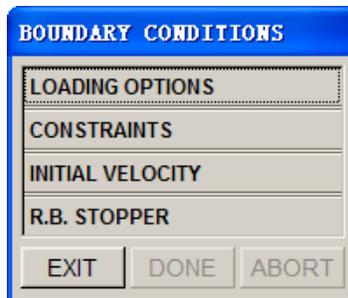


Figure 5.6.15 Boundary Conditions dialog box

## 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.

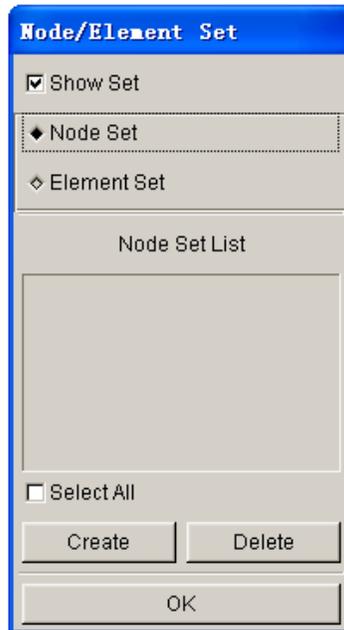


Figure 5.7.1 Node/Element Set dialog box

- 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. In eta/DYNAFORM 5.6, DFE module has been updated to provide a simpler GUI for the user to operate. In addition, with DFE module, the user can import the part model and complete the whole process design quickly.

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.

- Generate middle surface and separate top/bottom surfaces
- Flange Unfolding
- Edit and split surface
- Tool mesh and mesh check/repair
- Automatic filling of holes, boundary fill and outer boundary smooth
- Automatic and interactive generation of Side-Step
- Automatic and interactive generation of Symmetry/Double Attach
- Automatic tipping and adjusting of drawing direction
- 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
- Morphing surfaces and elements
- 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. In Dynaform 5.6, DFE Preparation menu is equipped with a new user interface that contains DEFINE, SYMMERY/DOUBLE ATTACH and TIPPING. 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.

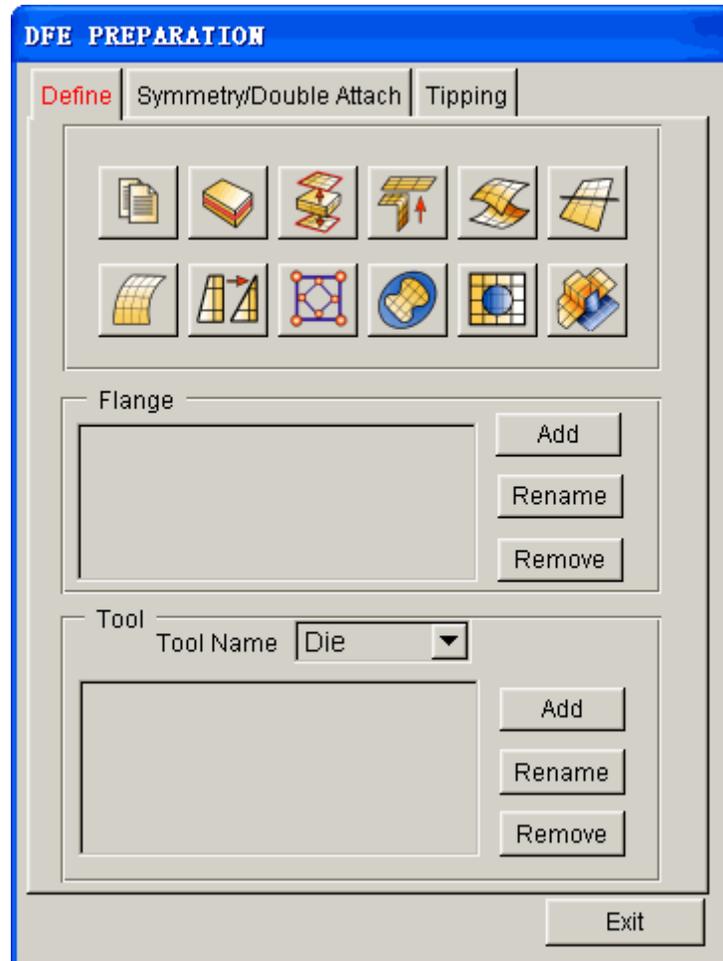


Figure 6.1.1 DFE Preparation Menu

### 6.1.1 DEFINE

This function allows the user to read in CAD model or finite element model. If the imported CAD data is a solid model with thickness information, the user can use MIDDLE SURFACE or SEPARATE SURFACE function to generate the desired surfaces. Then use UNFOLD FLANGE function to unfold the flange on the generated surfaces. In addition, the user can also use SURFACE EDIT or SURFACE SPLIT function to add the flange to a new part. Once the user clicks on the ADD button in FLANGE group, a dialog box as shown in Figure 6.1.2 appears. Click on SURFACE(S) to select flange surfaces, and then enter a part name

in the TO PART field, as shown in Figure 6.1.3. Click APPLY to create a new flange part. The user can also click  button to add the selected flange surfaces to another existing part. At the same time, the program will automatically add this part to the flange list, as shown in Figure 6.1.4. Besides, the user can also use RENAME and REMOVE button in FLANGE group to rename the flange part or delete the flange part.

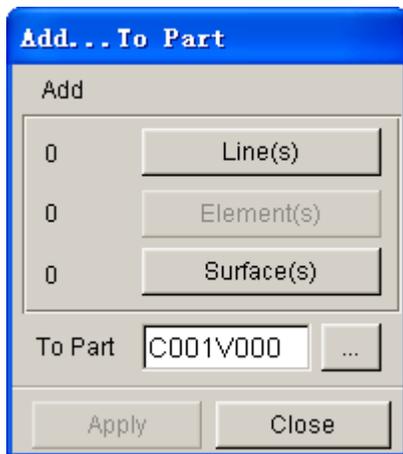


Figure 6.1.2 Select Flange Surface



Figure 6.1.3 Create Flange Part

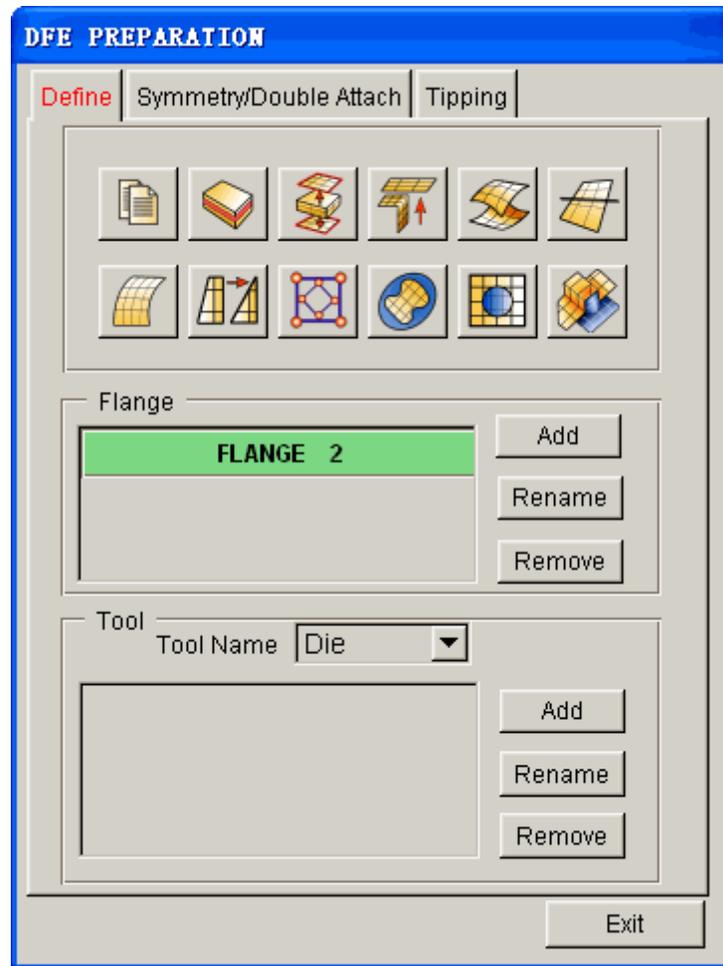


Figure 6.1.4 Add Flange Part

In the mean time, the user can click on ADD button in TOOL group to define some parts as punch and perform SURFACE MESH, INNER FILL, OUTER SMOOTH or SIDE-STEP on the defined punch. In addition, the user can also click on RENAME and REMOVE buttons to rename the punch part and delete the punch part. After defining the punch, the font color of DEFINE will change from red to black.

Details about functions in the interface are described below.

### 6.1.1.1 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.

### 6.1.1.2 MIDDLE SURFACE



This function allows user to create the mid surface from a solid part. For more information please refer to the Section 5.2.7.

### 6.1.1.3 SEPARATE SURFACE



This function is used to separate the top and bottom surface layers of sheet metal parts with the thickness information. For more information please refer to the Section 5.2.8.

### 6.1.1.4 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 6.1.5 for typical surfaces with the defined flange highlighted.

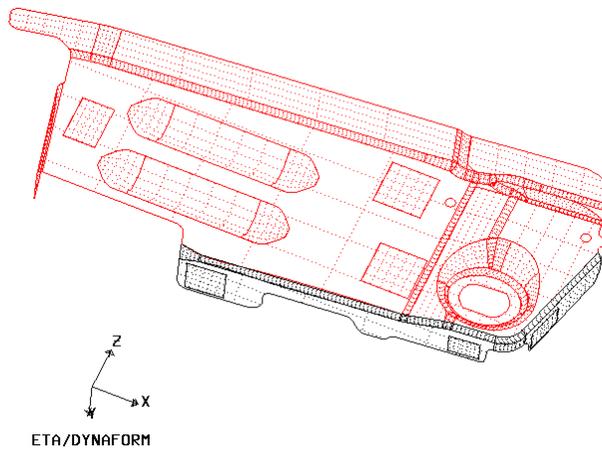


Figure 6.1.5 Select Flange Surface

- After selecting surfaces, the DFE module will determine and highlight the unfold line and the boundary line. As shown in Figure 6.1.6, 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 6.1.7. The user has to select an option to continue operation.

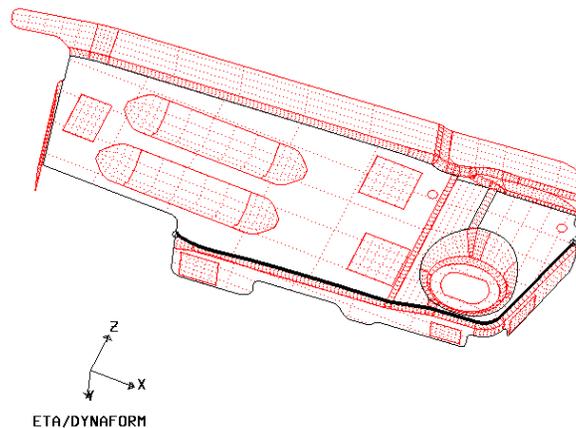


Figure 6.1.6 Unfold Line and Boundary Line



Figure 6.1.7 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 6.1.8 is displayed to allow the user to change the gap tolerance between surfaces.



Figure 6.1.8 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 6.1.9 is displayed. The bent angle is the angle between the unfolded surface and the base surface. The default angle ( $0^\circ$ ) indicates that unfolded surfaces are tangent to the base surfaces at the unfold line.

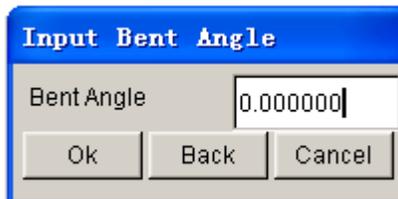


Figure 6.1.9 Input Bent Angle dialog box

- Select **OK** to accept the defined bent angle. The unfolded surfaces are shown in Figure 6.1.10 in yellow.

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

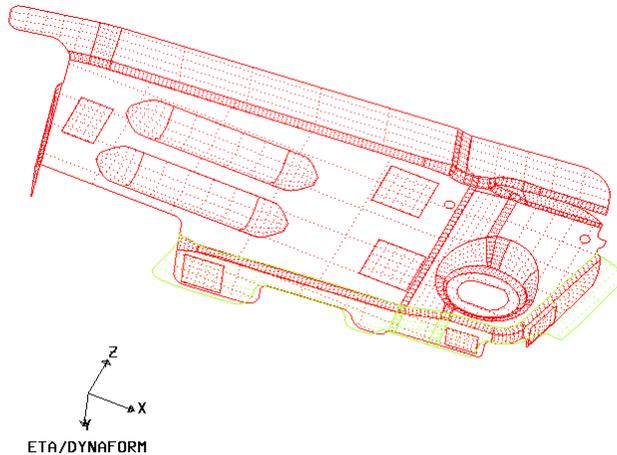


Figure 6.1.10 Unfolded Flange

- Select an option from the CONTROL KEYS dialog box as illustrated in Figure 6.1.11.



Figure 6.1.11 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 6.1.12. The CONTROL KEYS dialog box as illustrated in Figure 6.1.13 is displayed. The user has to select the next step from the options provided in the dialog box.

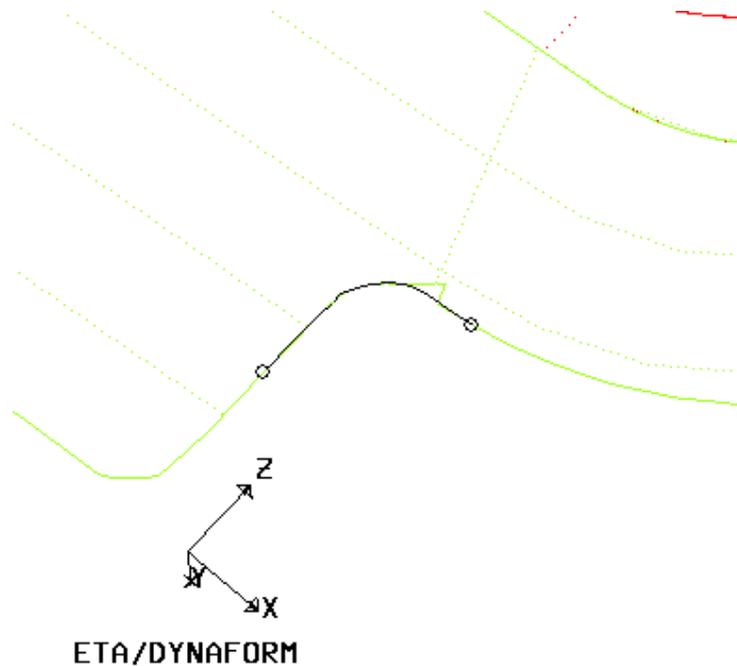


Figure 6.1.12 Smooth Edge



Figure 6.1.13 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.

### **6.1.1.5 SURFACE EDIT**



This function allows the user to edit surface by editing the control points on surface, as shown in Figure 6.1.14.



Figure 6.1.14 Surface Edit dialog box

After clicking on SELECT SURFACE button, the user can use cursor to select a surface on which the control mesh will be highlighted, as shown in Figure 6.1.15. Then the user can click DEFINE DIRECTION button to display the Define Vector dialog box. Please refer to Section 5.1.4 Shortest Curve for detailed operation.

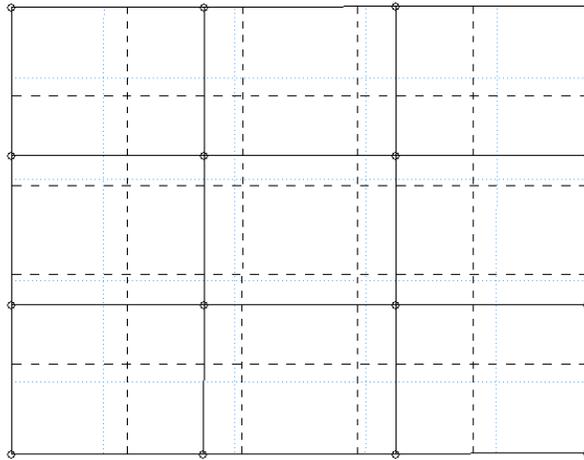


Figure 6.1.15 Display Surface Control Mesh

Once the surface morphing direction is defined, the user can select one or multiple (SHIFT + left mouse button) control points by cursor and then drag one of the selected control points to edit surface, as shown in Figure 6.1.16.

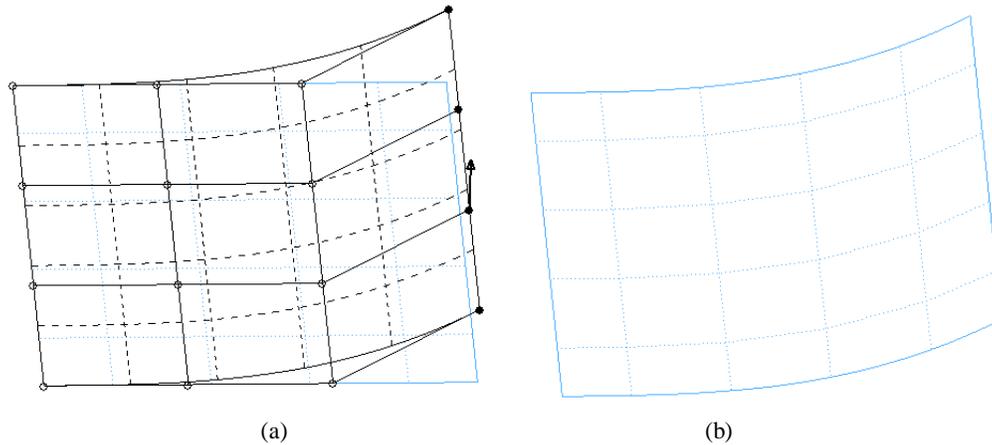


Figure 6.1.16 Edit Surface

*Note: The edited surface must be an non-trim surfaces.*

### 6.1.1.6 SURFACE SPLIT



This function allows the user to split the selected surface into two surfaces at the specified line. Refer to Section 5.2.11 for detailed description of surface split.

### 6.1.1.7 MESH TOOL



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

### 6.1.1.8 MODEL CHECK /REPAIR

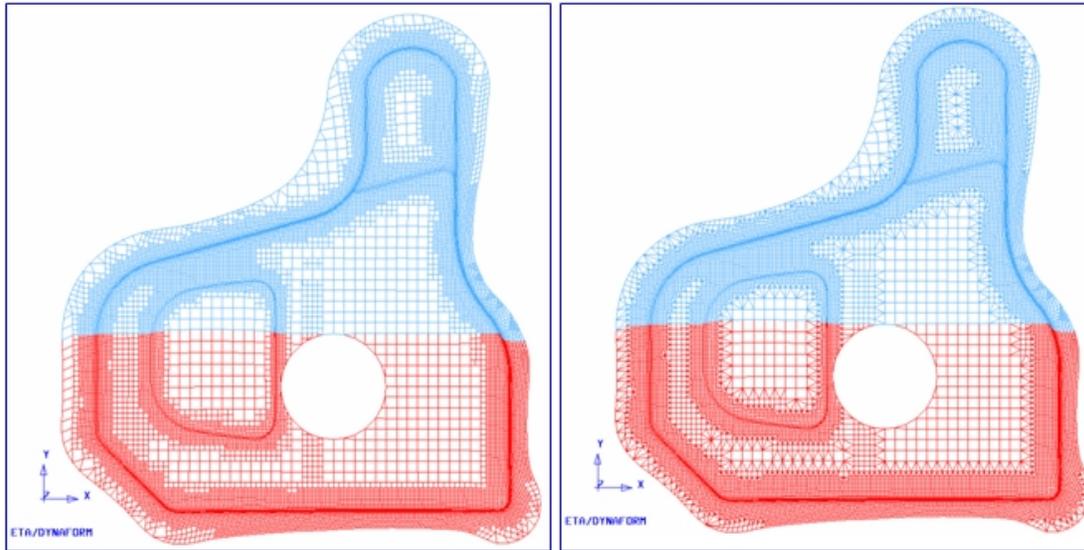


All functions in the MODEL CHECK /REPAIR dialog box are described in Sections 5.5. Please refer to the section for a detailed description of these functions.

### 6.1.1.9 TOPOLOGY REPAIR



This function allows the user to auto repair topological structure that is not connected. 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 6.1.17.



a) Original topology

b) Repaired topology

Figure 6.1.17 Auto topology repair

### 6.1.1.10 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 6.1.18, there are four functions in the dialog box (ROLLER, MORPHING, CORNER SMOOTH, EXPAND).

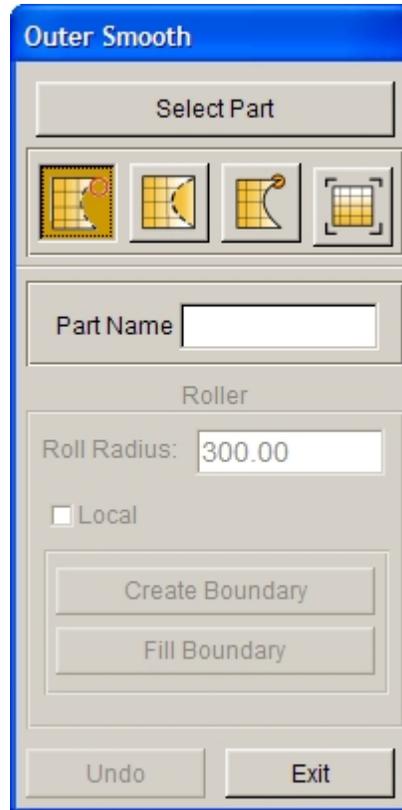


Figure 6.1.18 Outer Smooth

Before outer smooth, the user must specify a part by clicking on SELECT PART button.

- **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 6.1.19 is displayed. The default radius of the cylinder is 300 (mm) in the ROLL RADIUS input field.

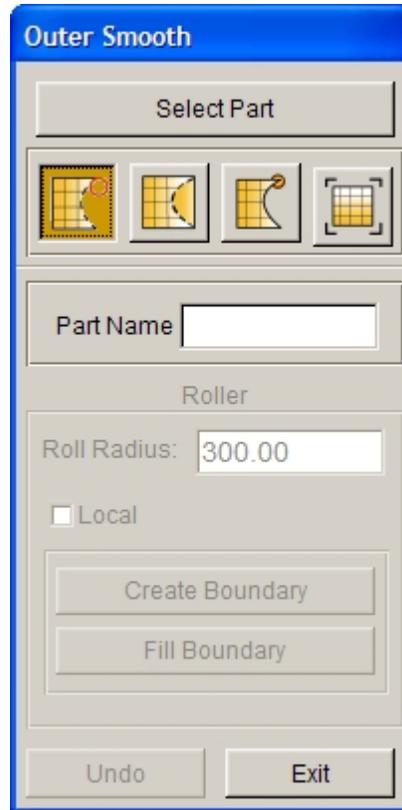


Figure 6.1.19 Roller dialog box

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 6.1.20 illustrates the result of a typical roller operation.

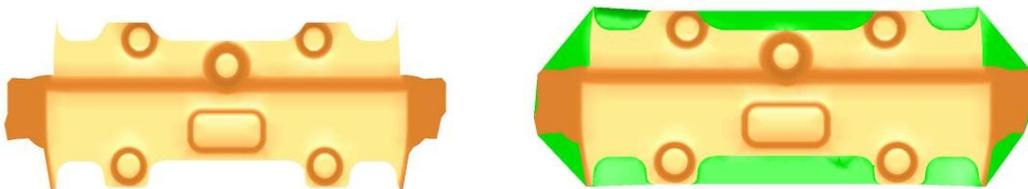


Figure 6.1.20 An illustration of a roller operation

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. 6.1.21 illustrates the result of a typical roller operation using the LOCAL option.

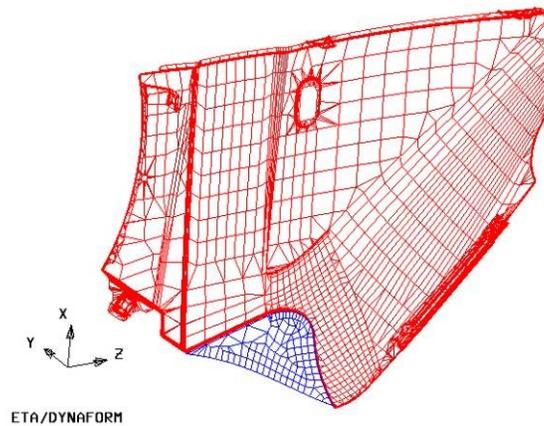


Fig. 6.1.21 An illustration of Outer Smooth/Roller using Local option

- MORPHING

This function allows the user to locally modify the part mesh by morphing the boundary line of the part. Figure 6.1.22 illustrates dialog box of the MORPHING function.



Figure 6.1.22 Morphing dialog box

1. Click on the MORPHING BOUNDARY button to highlight the boundary of the die. Eta/DYNAFORM requests the user to select two points on the part boundary to define a morphing region. Figure 6.1.23 illustrates the selected region on the part boundary for morphing operation.

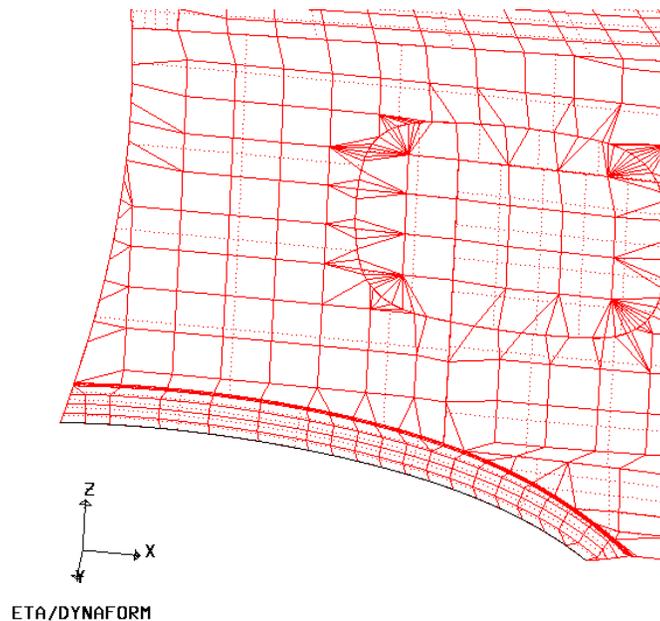


Figure 6.1.23 Select Morphing Boundary

2. After two points are selected, the CURVE EDIT dialog box as illustrated in Figure 6.1.24 is

displayed. About the detailed description of Curve Edit function, the user can refer to the Section 5.1.15.

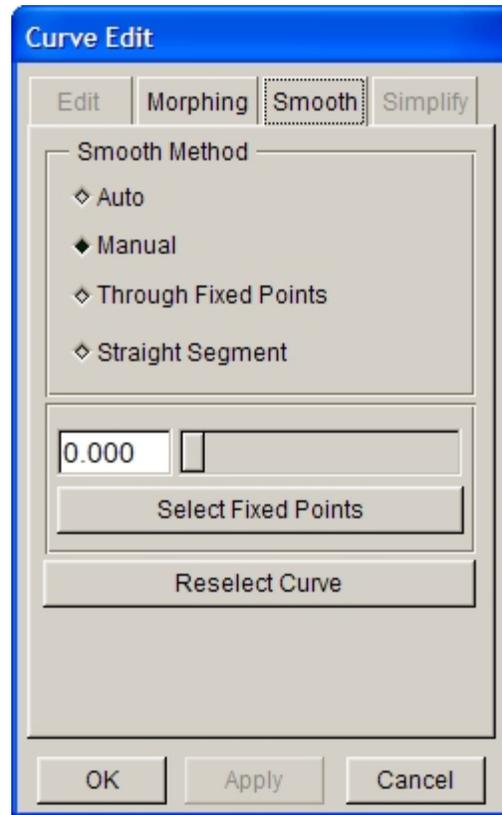


Figure 6.1.24 Curve Edit dialog box

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 6.1.25.

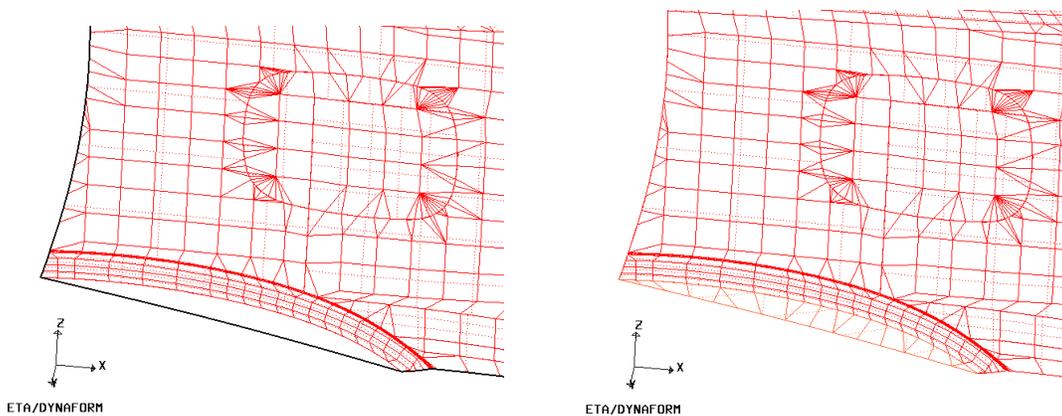


Figure 6.1.25 Morphing Result

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 EXIT button to exit the function. The generated elements, nodes or surfaces, etc. will be automatically saved in a part named IGS\_SMH. The user can modify the name of this part and the generated part will be automatically added to the punch.

- 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 6.1.26 is displayed. The default corner radius is 30 (mm) as displayed in the input field of CORNER RADIUS.

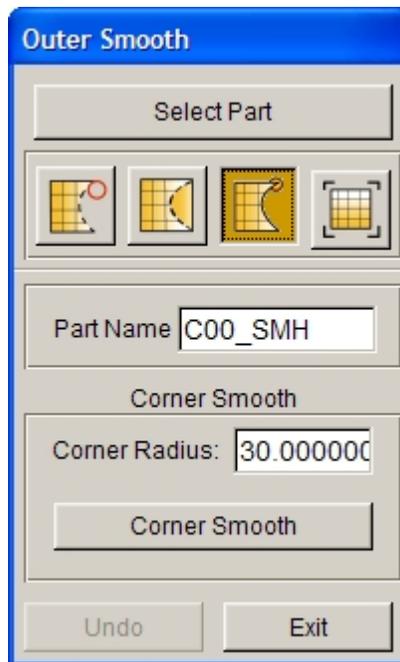


Figure 6.1.26 Corner Smooth dialog box

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 6.1.27.

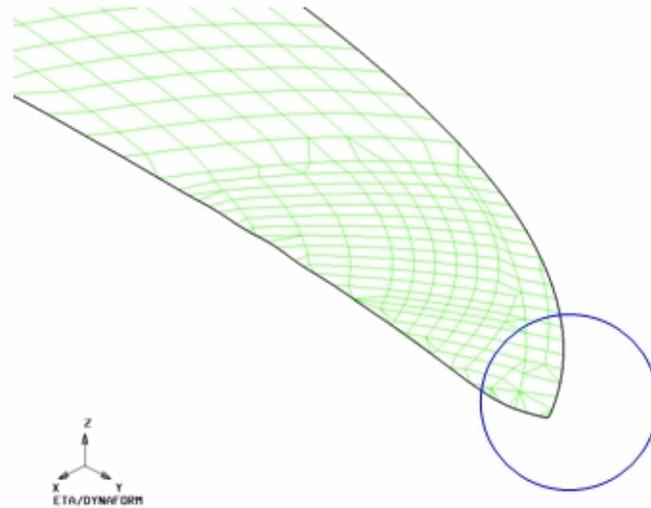


Figure 6.1.27 Sharp Corner

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 6.1.28 illustrates a typical result of a corner smooth operation.

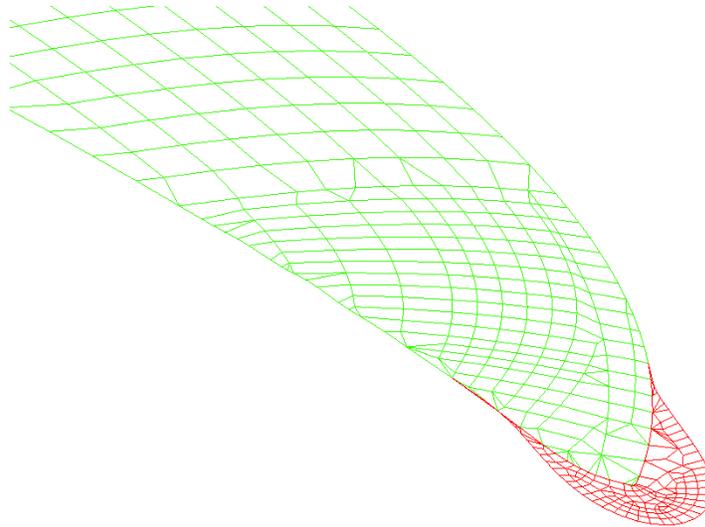


Figure 6.1.28 Corner Smooth

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.

- EXPAND

The EXPAND function is used to extend the part boundary with elements. When this function is selected, dialog box as illustrated in Figure 6.1.29 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.

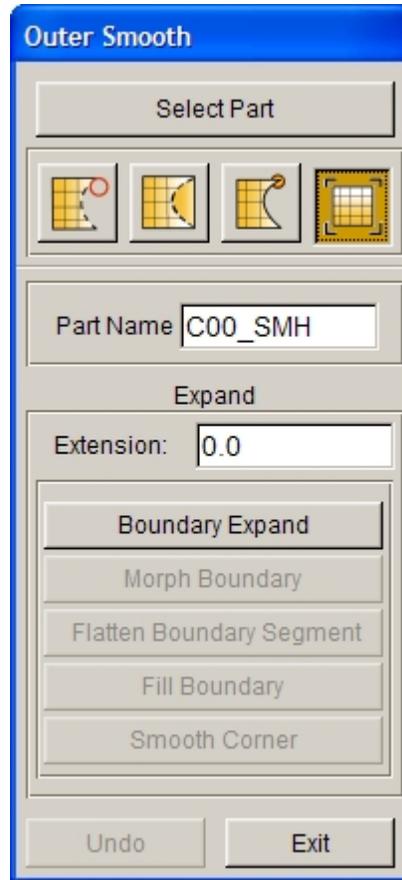


Figure 6.1.29 Boundary Expand dialog box

1. Click on the BOUNDARY EXPAND button to highlight the boundary of extension as illustrated in Figure 6.1.30a. 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 5.1.15 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.

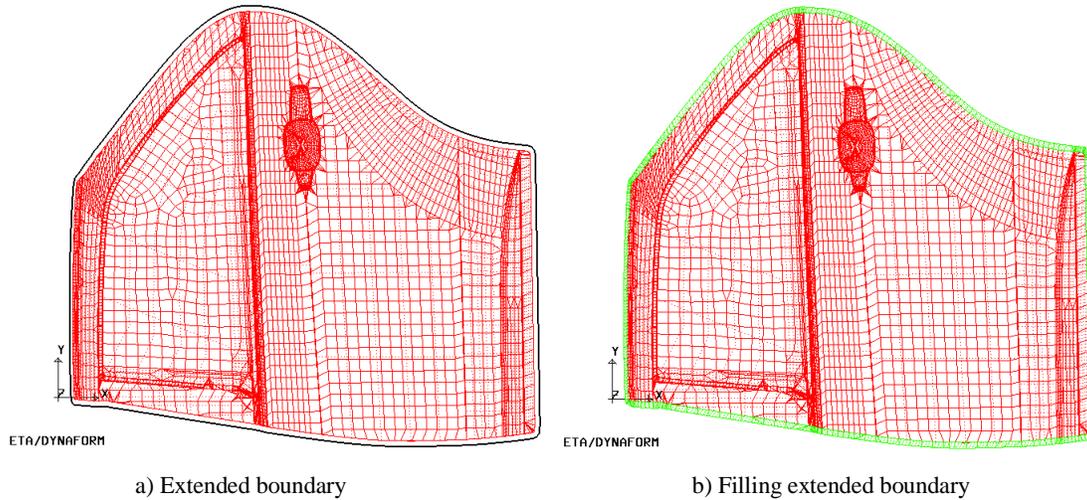


Figure 6.1.30 An illustration of the Boundary Expand operation

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 6.1.30b.
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  $30^\circ$ .

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



### 6.1.1.11 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 6.1.31. These functions are:

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

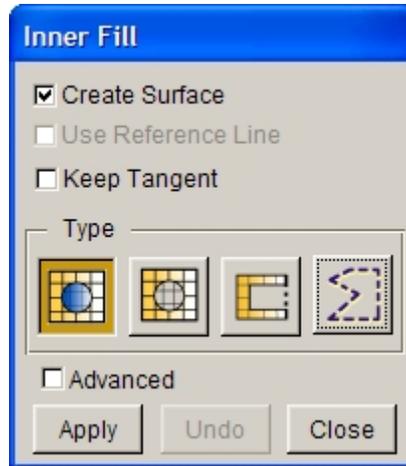


Figure 6.1.31 Inner Fill dialog box

**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.

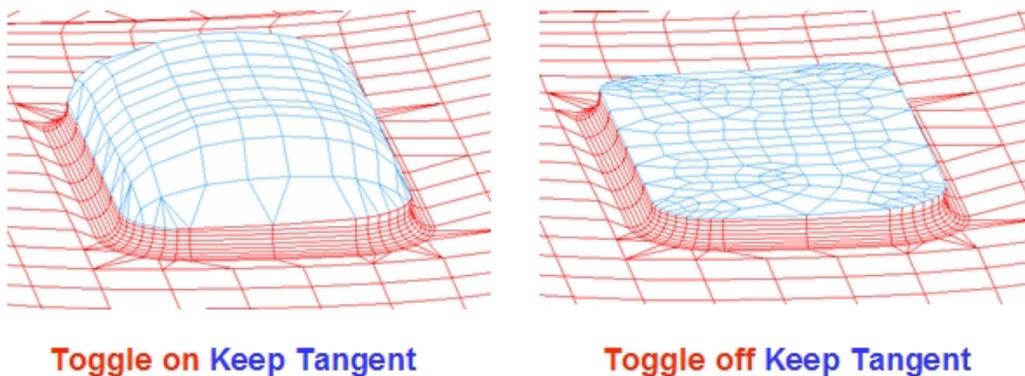


Figure 6.1.32 Toggle on/off Keep Tangent option

**ADVANCED**

The advanced option includes some auxiliary functions that are mainly applied on single hole fill, as shown in Figure 6.1.33. These functions are used for generation of reference lines, removal of curves, surfaces and elements. Detailed description about these functions refers to Section 5.2.11 Surface Split.

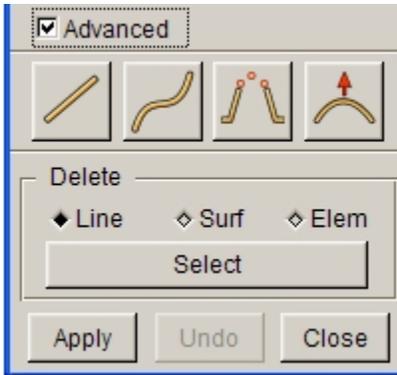


Figure 6.1.33 Advanced Option

**DELETE**

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

- **AUTO FILL**

After selecting this button and clicking APPLY, the program will automatically find all the inner holes and highlight their boundaries, as shown in Figure 6.1.34.

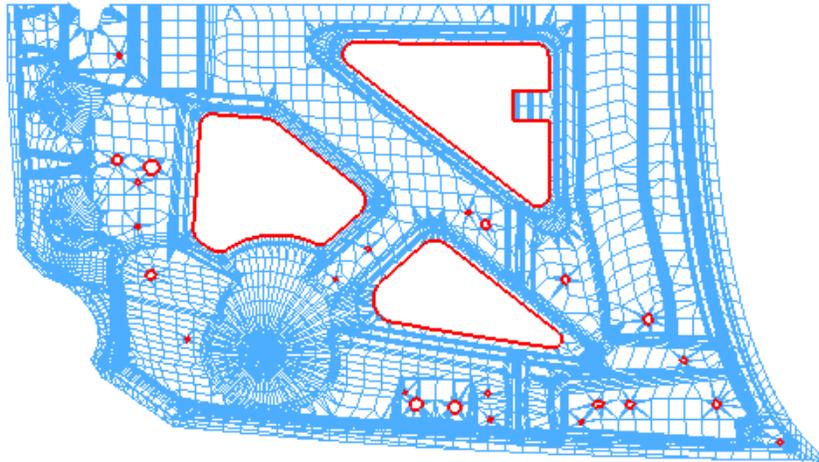


Figure 6.1.34 Before Inner Fill

Simultaneously, eta/DYNAFORM prompts:

***CLICK ON INNER BOUNDARYS TO DE-SELECT  
MIDDLE BUTTON – DONE, RIGHT BUTTON - CANCEL***

If there are holes which do not need to be filled, the user can select them by mouse cursor and click on the middle mouse button to confirm. The result is shown in Figure 6.2.35.

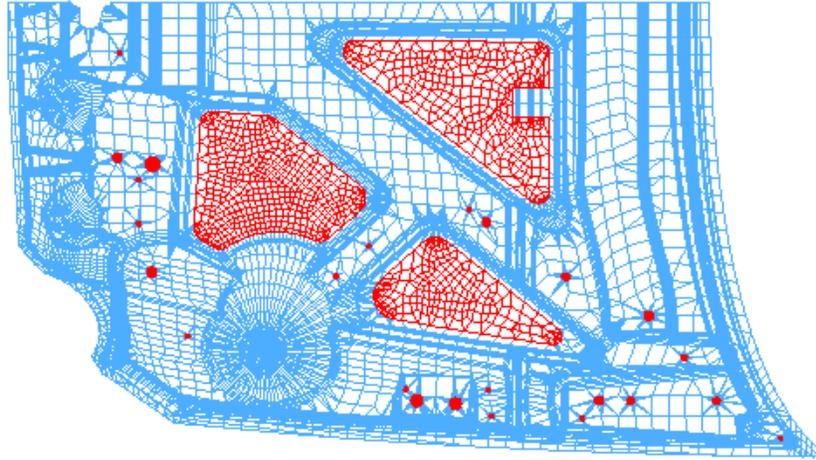


Figure 6.1.35 After Inner Fill

### UNDO

If the user is not satisfied with the result of Auto Fill, he/she can select UNDO button to cancel this result.

### EXIT

Exit the Inner Fill interface.

### ● HOLE FILL

This function allows the user to fill holes individually. Once this function is selected, the boundary of the inner hole on the current screen is highlighted. Then, the program prompts the user to select an inner boundary to be filled by using the left mouse button. After an inner boundary is selected, the program will automatically fill the hole, as illustrated in Figure 6.1.36.

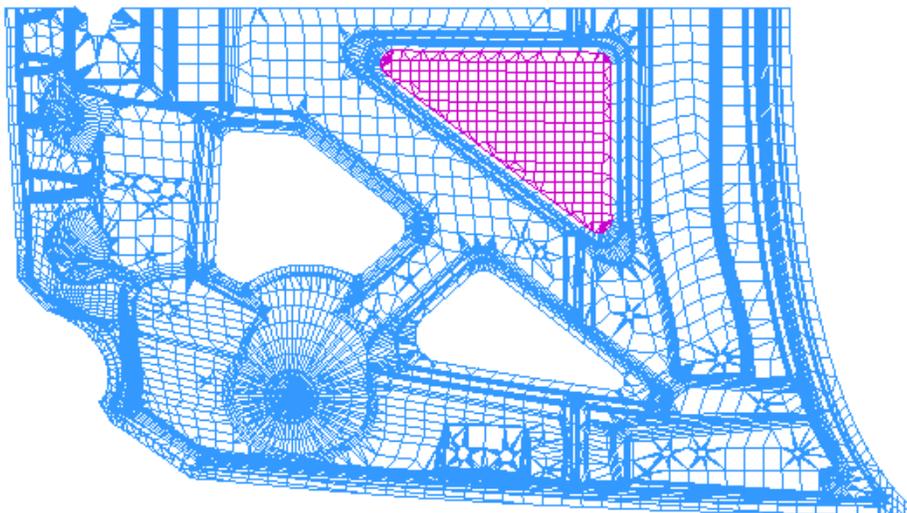


Figure 6.1.36 Boundary Lines of Inner Holes

If Hole Fill function is selected, User Reference Line option is activated for the user to create some reference lines for hole filling, as illustrated in Figure 6.1.37.

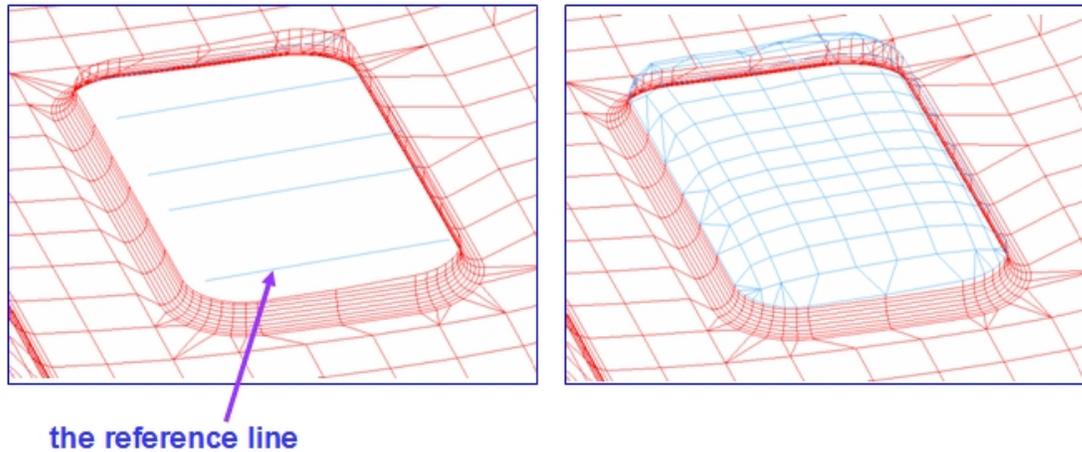


Figure 6.1.37 Filled Mesh within a Closed Curve

- **CURVE FILL**

This function allows the user to fill an open or cut-out area. Figure 6.1.38 illustrates an example of Curve Fill operation using a spline curve to close the cut-out area.

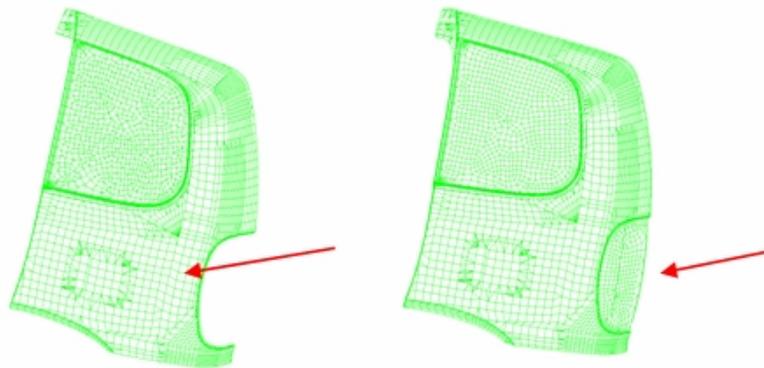


Figure 6.1.38 Filled Mesh with Open Curve

**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.

#### ● **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.

#### **6.1.1.12 SIDE-STEP**



This function allows the user to extend part boundary and provide side-step features. The dialog box is illustrated in Figure 6.1.39.

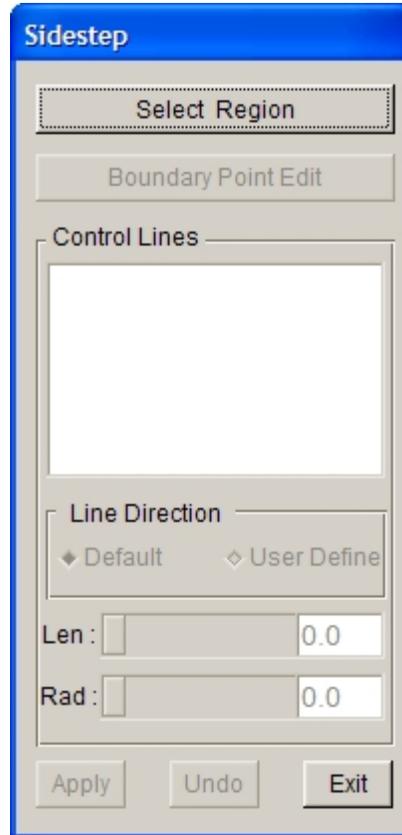


Figure 6.1.39 Sidestep dialog box

- **SELECT REGION**

This function allows the user to select the desired region. After clicking on this button, the program will highlight the part boundary. The user can first select two boundary points by cursor to define the region at the bottom of Sidestep, and then select another two boundary points to define the region at the top of Sidestep. The program will automatically generate the outline of the region, as shown in Figure 6.1.40.

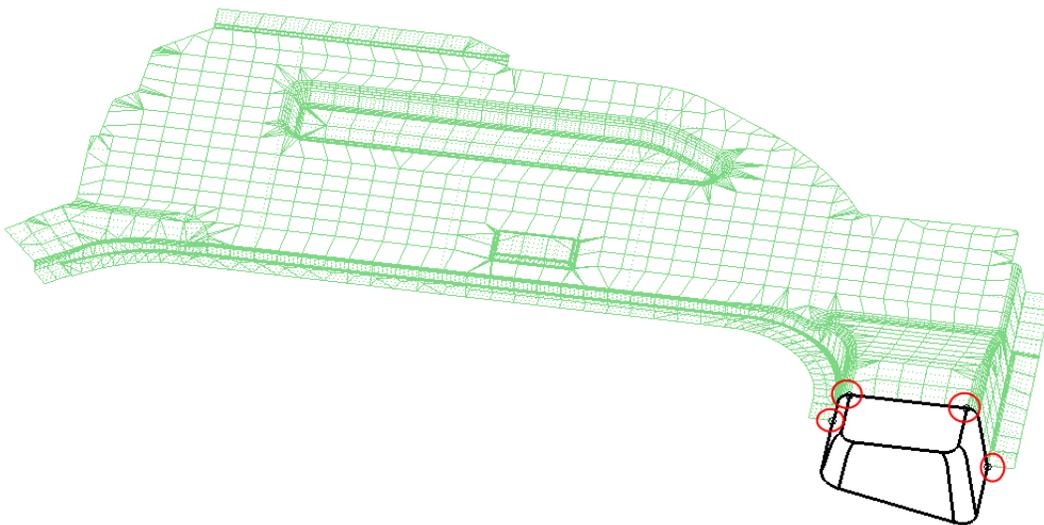


Figure 6.1.40 Select Region

- **BOUNDARY POINT EDIT**

This function allows the user to edit boundary point. After region is selected, this button is activated. The user can click on this button and move boundary point by cursor to edit the region.

- **CONTROL LINES**

This function allows the user to list the control lines. In Control Lines list, the user can select the control lines to be edited.

- **LINE DIRECTION**

This function allows the user to define line direction. The user can select DEFAULT or USER DEFINE. Once this option is selected, the program will pop up a Define Vector dialog box. Please refer to Section 5.1.4 Shortest Curve for detailed operation of this dialog box.

- **LENGTH**

This function allows the user to edit length of control lines. The user can select the desired control lines in Control Lines list and then drag slider to the desired length.

- **RADIUS**

This function allows the user to edit radius of control lines. The user can select the desired control lines in Control Lines list and then drag slider to the desired radius.

After completion of editing, click APPLY to generate Sidestep, as shown in Figure 6.1.41.

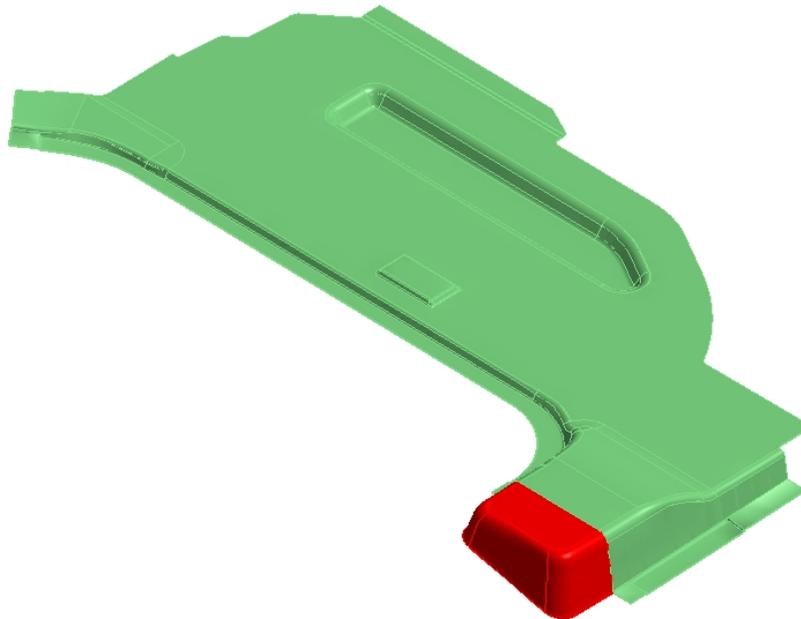


Figure 6.1.41 Generated Sidestep

## 6.1.2 SYMMETRY/DOUBLE ATTACH

The default interface of Symmetry/Double Attach is illustrated in Figure 6.1.42(a). The user can select any function in Double Attach to display the interface as shown in Figure 6.1.42(b). In Symmetry function, there are four types of part geometry for the user to input: NO SYMMETRY, SYMMETRY, HALF SYMMETRY INPUT and QUARTER SYMMETRY INPUT. In Double Attach function, there are three types of part geometry for the user to select: LEFT / RIGHT HALF INPUT, LEFT / RIGHT FULL INPUT and TWO PARTS. When using LEFT / RIGHT HALF INPUT, the user should first mirror another same part using Symmetry function, then design the Double Attach.

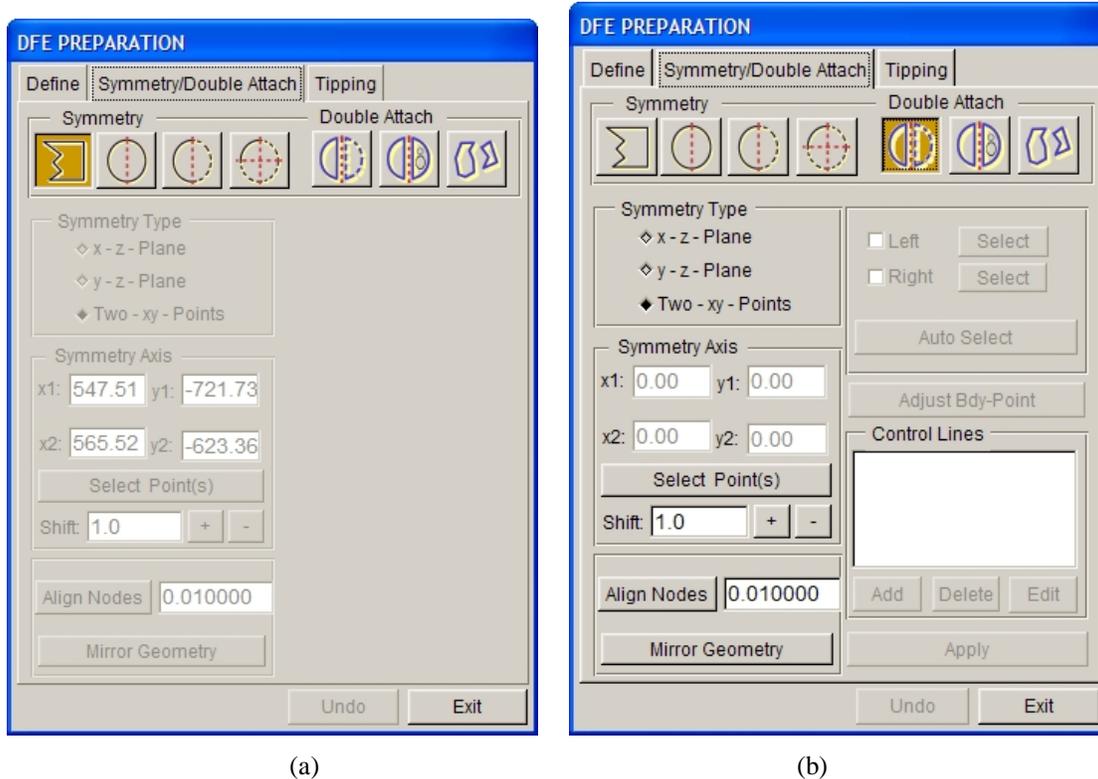


Figure 6.1.42 Symmetry / Double Attach

### 6.1.2.1 SYMMETRY

This function allows the user to define a symmetrical die by mirroring a half symmetry or quarter symmetry plane.

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 SYSSMETRY 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.

The user can enter the SHIFT value and click on  or  to move the symmetry axis. 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. After clicking on this button, the program will pop up a DYNAFORM Question dialog box, as shown in Figure 6.1.43, and highlight the result from the operation. The user can select YES to accept the result or select NO to correct the inconsistency between some nodes on symmetry plane and symmetry plane direction within the offset range via adding align nodes by cursor. The user can also select ABORT to redefine the offset range.

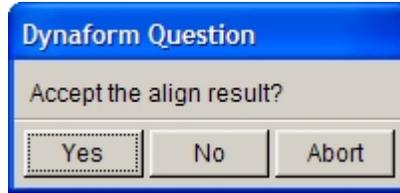


Figure 6.1.43 DYNAFORM Question dialog box

Note: If the symmetry plane is not defined to be die, when clicking on ALIGN NODES, the message window prompts:

DIE NOT DEFINED IN DATABASE

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.

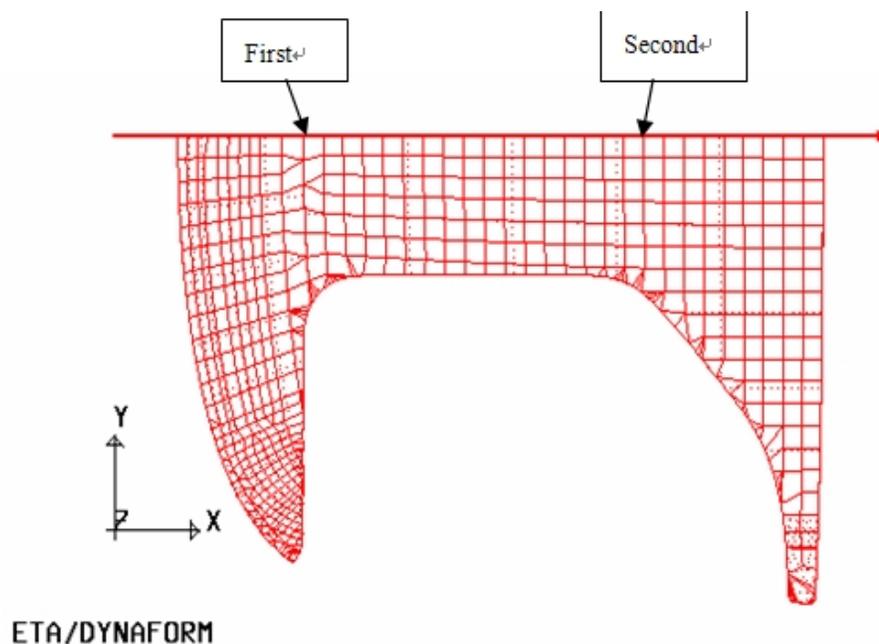


Figure 6.1.44 Half Symmetry Part With X-Z Plane

5. Click on the MIRROR GEOMETRY SYMMETRY button, following by selecting the part as illustrated in Figure 6.1.44. The mirror result is illustrated in Figure 6.1.45.

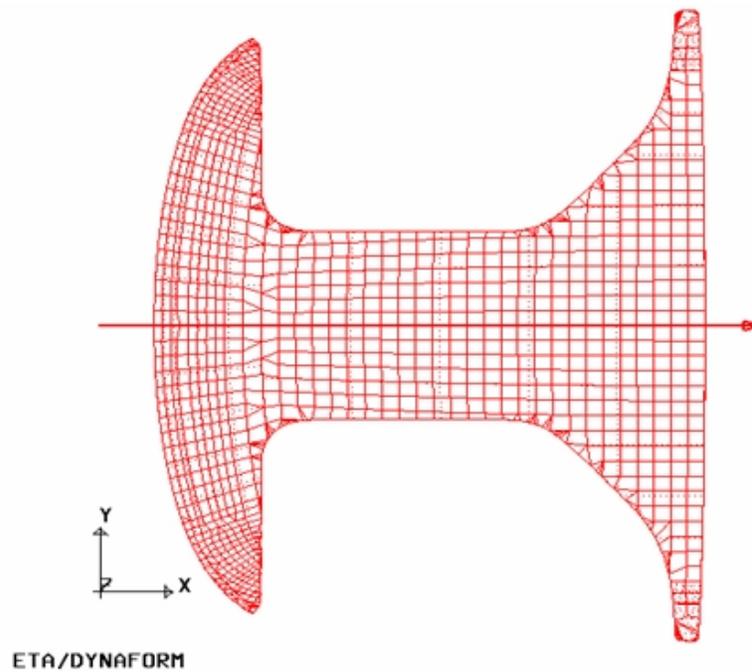


Figure 6.1.45 The Mirroring Result

### 6.1.2.2 DOUBLE ATTACH

This function allows the user to design Double Attach by entering the part geometry. There are three types of part geometry listed below.



#### LEFT / RIGHT HALF INPUT

This function allows the user to input one part geometry, then mirror another same part by using Symmetry function. Once this button is selected, the Symmetry interface is activated, as shown in Figure 6.1.46.

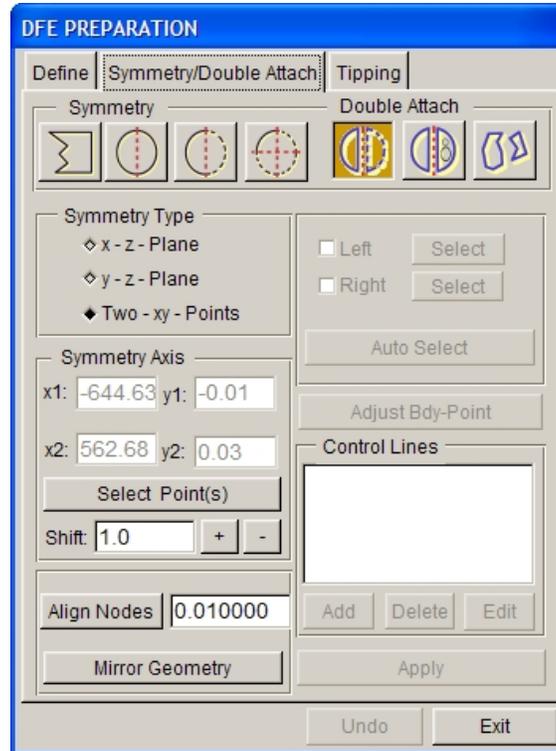


Figure 6.1.46 Left / Right Half Input

**LEFT / RIGHT FULL INPUT**

This function allows the user to input two part geometries with the same outline and different interior. Once this button is selected, the Symmetry Type is disabled, but Select buttons for Left and Right are activated, as shown in Figure 6.1.47.

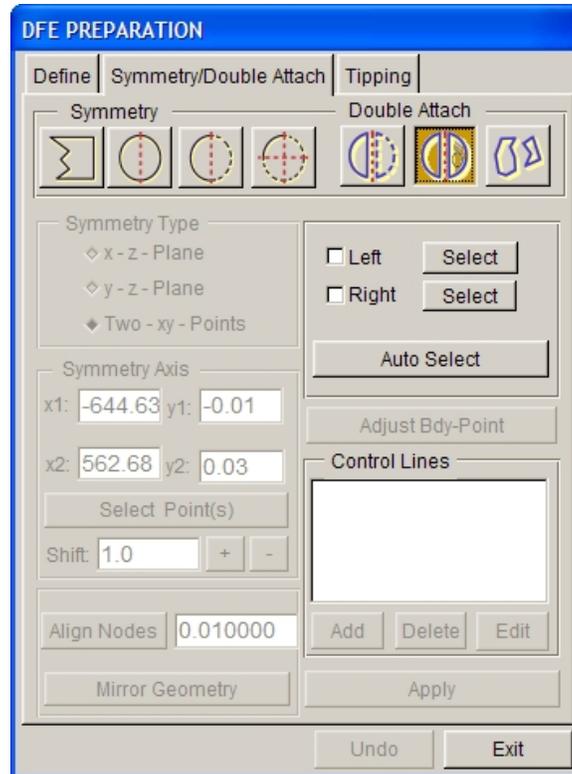


Figure 6.1.47 Left /Right Full Input

**TWO PARTS**

This function allows the user to input two totally different part geometries. Once this button is selected, the Symmetry Type is disabled, but Select buttons for Left and Right are activated. The interface is similar to the one in Figure 6.1.47.

The user can select LEFT and click on SELECT to select the left part; select RIGHT and click on SELECT to select the right part; or click on AUTO SELECT to select left / right part automatically. After selection of part, the program will auto generate outline between two parts and activate ADJUST BDY-POINT for the user to adjust the location of boundary point. Once the outline is generated, the user can modify the shape of it by using ADD, DELETE and EDIT buttons.

If LEFT / RIGHT HALF INPUT option is selected, the Symmetry Type is activated. After mirroring the part via symmetry function, the program will select left / right part automatically and auto generate outline between two parts, as shown in Figure 6.1.48. Click APPLY to generate region, as illustrated in Figure 6.1.49.

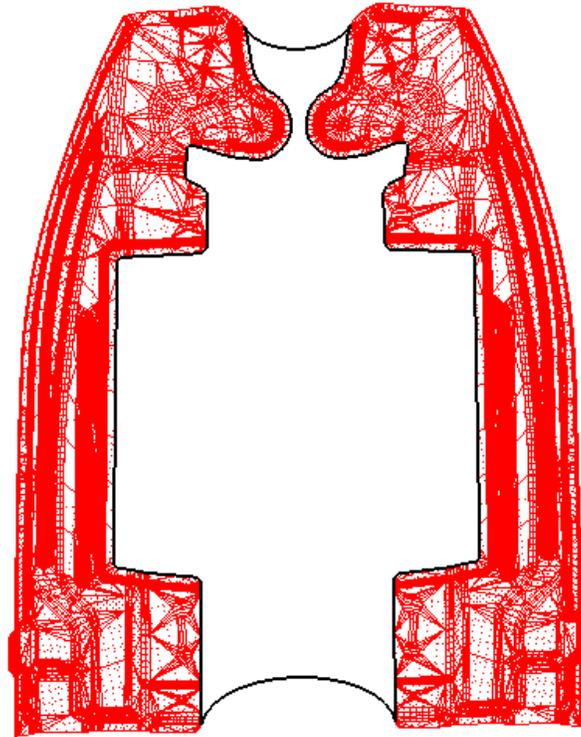


Figure 6.1.48 Generate Outline

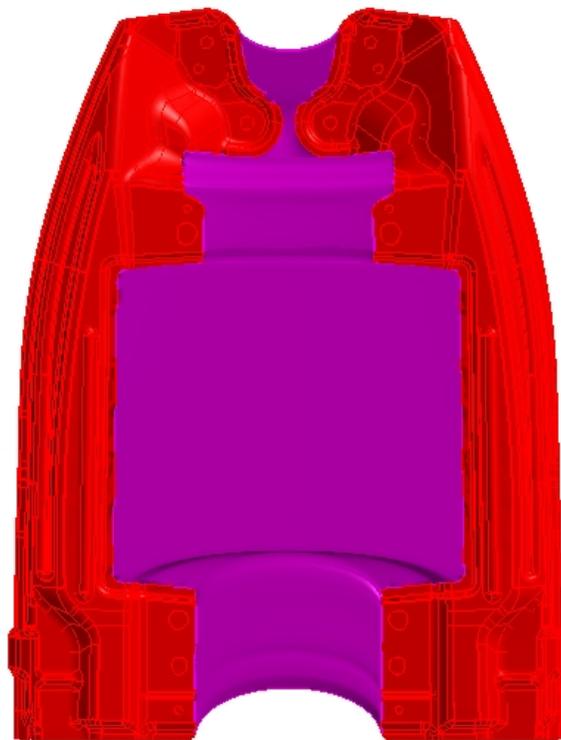


Figure 6.1.49 Generate Middle Region

If the generated region needs to be modified, the user can select ADD to add control lines. Click the left mouse button to add landscape control lines, as shown in Figure 6.1.50, and click the right mouse button to add portrait control lines, as shown in Figure 6.1.51. After control lines are added, select a control line in Control Lines list and click on EDIT to edit the control line in the displayed Edit Curve dialog box. The generated region after editing is illustrated in Figure 6.1.52. Refer to Section 5.1.15 Edit Line for detailed description.

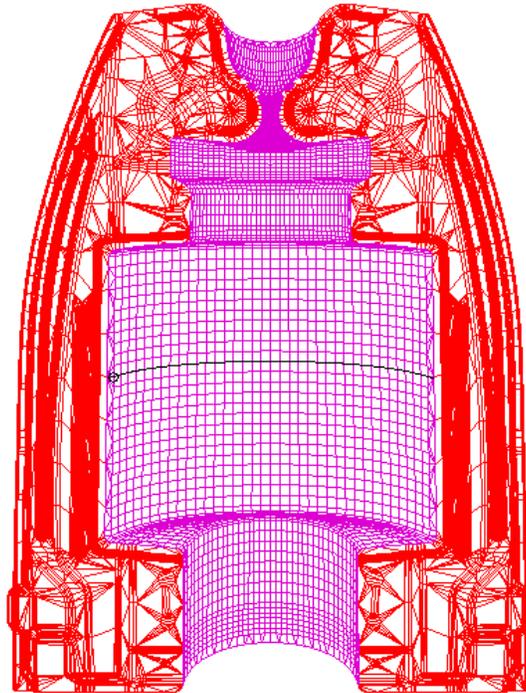


Figure 6.1.50 Add Landscape Control Line

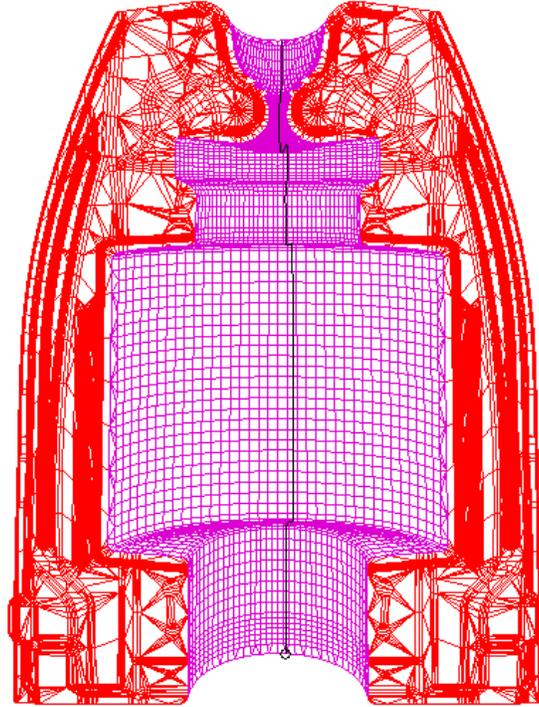


Figure 6.1.51 Add Portrait Control Line



Figure 6.1.52 Generated Region after Editing

If the generated region is easy to form undercut or the part boundary is not smooth, the user can first smooth the boundary by OUTER SMOOTH function to avoid undercut or poor quality. Refer to Section 6.1.1.10 OUTER SMOOTH for detailed description.

### 6.1.3 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 6.1.53.

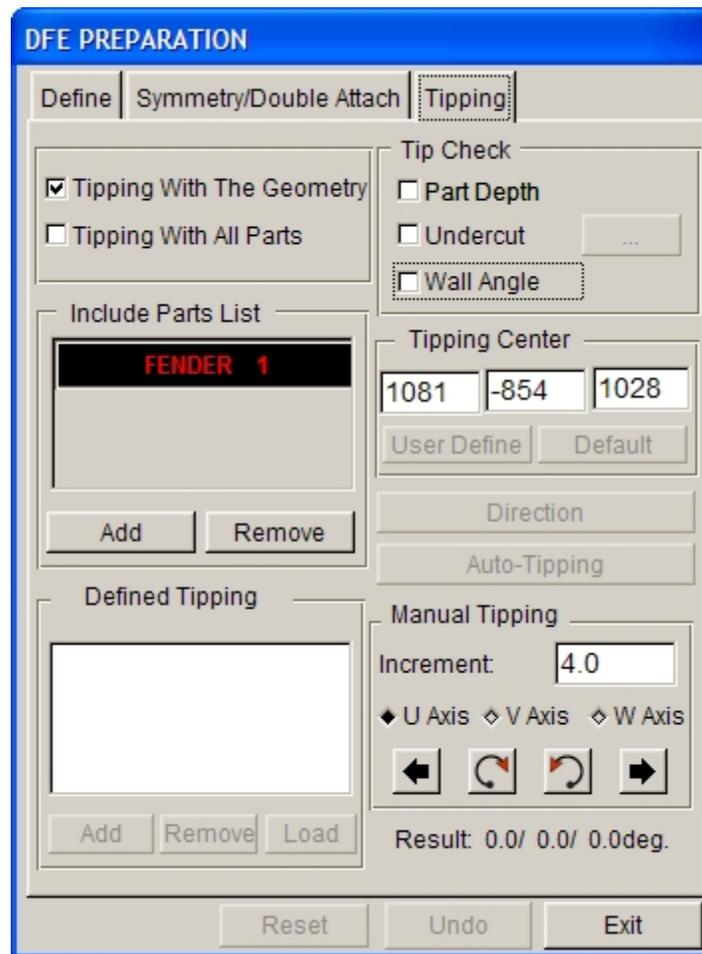


Figure 6.1.53 Tipping interface

Before tipping, there are some parts already existing in the part list, as shown in Figure 6.1.53. When defining some parts as DIE in the previous DEFINE interface, the program automatically adds these parts into the Include Part List on Tipping interface.

#### 6.1.3.1 TIPPING WITH THE GEOMETRY

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.

### 6.1.3.2 TIPPING WITH ALL PARTS

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.

### 6.1.3.3 INCLUDE PARTS LIST

This function allows the user to add the parts that need tipping into the list using ADD button. The user can also delete the unnecessary parts using REMOVE button. After tipping, the ADD and DELETE buttons are disabled so that the user cannot add or delete parts.

### 6.1.3.4 DEFINED TIPPING

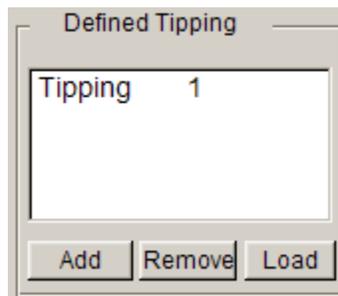


Figure 6.1.54 Record and Restore the Defined Result

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.

### 6.1.3.5 TIPPING CHECK

This function allows the user to check the result after tipping operation, as shown in Figure 6.1.55.

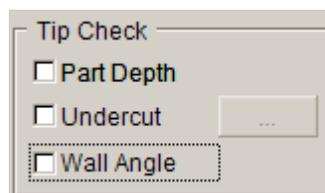


Figure 6.1.55 Check tipping

- PART DEPTH

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

This function displays a contour plot showing regions in the die with undercut. Red indicates severe undercut where the backdraft is less than 0°. Yellow indicates a marginal area with a draw angle of approximately

0~3°. 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. Click  button to display a dialog box as illustrated in Figure 6.1.56.



Figure 6.1.56 Undercut Limit dialog box

- **WALL ANGLE**

Once this option is selected, the program displays a contour plot showing wall angle on the die. Different wall angles are displayed in different colors. The user can point the cursor to any node on the selected part and click on the left mouse button to show the wall angle at this node, as illustrated in Figure 6.1.57.

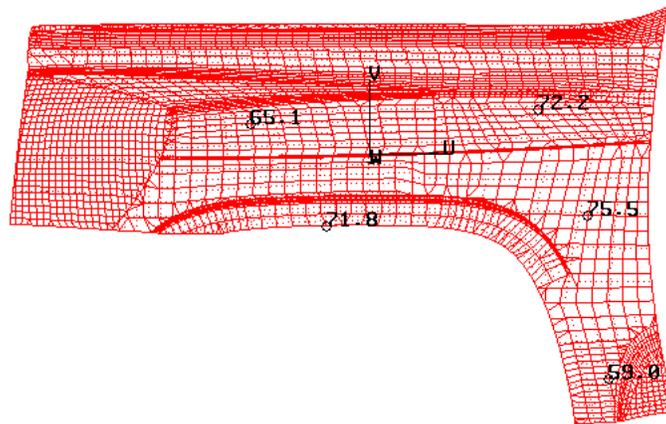


Figure 6.1.57 Display Wall Angle

### 6.1.3.6 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 6.1.58.

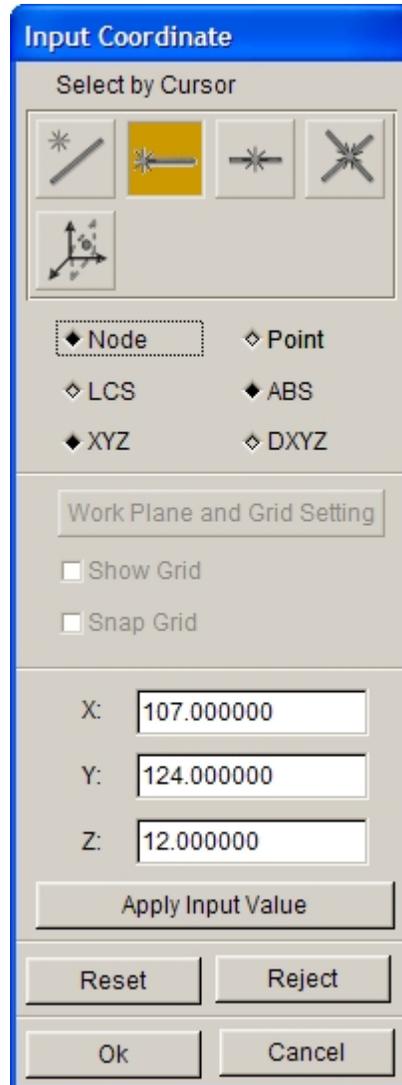


Figure 6.1.58 Selection of Tipping Center using INPUT COORDINATE dialog box

### 6.1.3.7 DIRECTION

This function allows the user to define tipping direction, click on this button to display Define Direction dialog box, as shown in Figure 6.1.59.

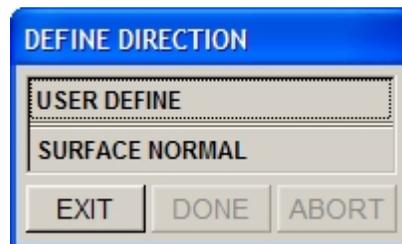


Figure 6.1.59 Define Direction dialog box

- USER DEFINE

This function allows the user to define tipping direction by defining vector. Refer to Section 5.1.4 Shortest Curve for detailed description on defining vector.

- SURFACE NORMAL

This function allows the user to define tipping direction via the normal of the selected surface.

### 6.1.3.8 AUTO-TIPPING

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

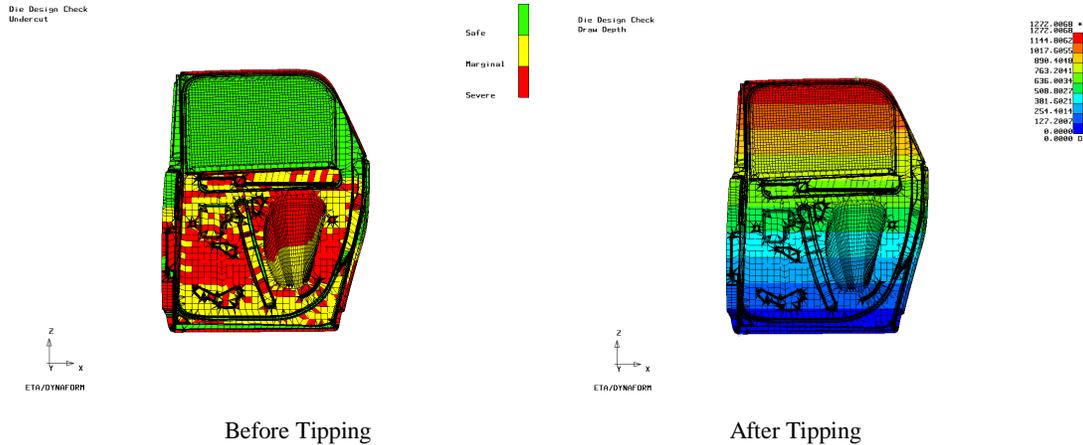


Figure 6.1.60 Tipping Check

**NOTE:** eta/DYNAFORM will automatically compute the tipping center. A default local coordinate system is created at the tip center with the UVW direction that is consistent with XYZ of the global coordinate system. The W axis is always set as the drawing direction. The functions on the tipping dialog box rotate the die so that the W axis is set as the drawing direction.

### 6.1.3.9 MANUALTIPPING

In addition to the automatic tipping function described in Section 6.1.3.8, the user can manually tip the die about U, V, and W axes to a better draw direction. This function is very useful when auto-tipping does not provide a satisfactory tip result due to unbalanced mesh pattern or part shape.

In manual tipping operation, the DIE can be rotated/translated about/along the U-, V-, or W-axis. Using the U+/-, V+/-, W+/- buttons to perform the tipping operation according to the increment defined in the dialog box, as shown in Figure 6.1.61. The tipping check will be visible if the checkbox for UNDERCUT or DRAW DEPTH button is toggled on.

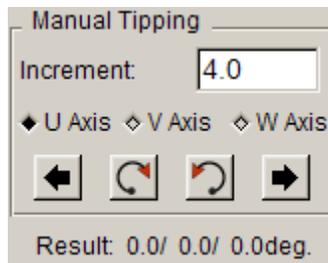


Figure 6.1.61 Manual Tipping

- **ROTATE**

Select ROTATE and key in any absolute rotation angle in the input field. Click on the U+(-), V+(-), and W+(-) buttons that represent the rotation axis to rotate the model with defined angle about the selected axis.

- **TRANSLATION**

If TRANSLATION option is selected, the U+(-), V+(-), W+(-) buttons represent translation direction. The value in the input field of TRANSLATION is the incremental displacement along the selected axis.

### **6.1.3.10 UNDO AND RESET TIPPING OPERATION**

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

## 6.2 BINDER

Binder module is a brand new module in DYNAFORM 5.6 for the user to generate various types of binder shapes.

There are 5 main functions on the new Binder interface: CREATE, MODIFY, FLANGE ON BINDER, BUTTERFLY and ADVANCED, as shown in Figure 6.2.1.

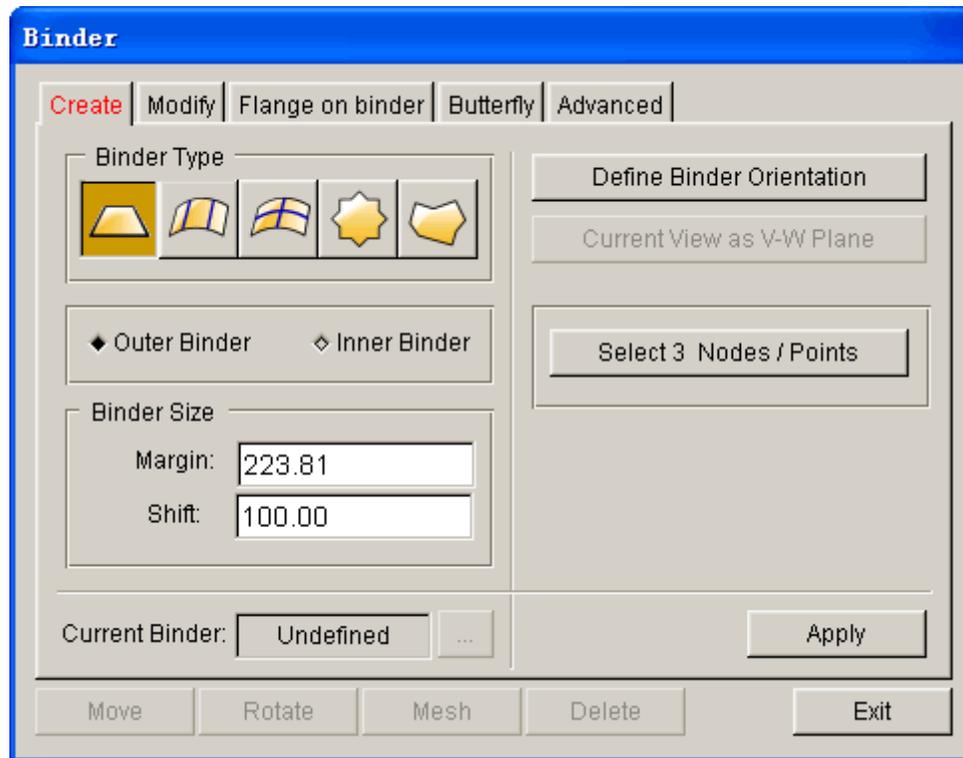


Figure 6.2.1 Binder Type

### 6.2.1 CREATE

This function allows the user to create single or multiple binders. After the binder is created, the color of CREATE button changes from red to black. The user can create several types of binder described below, as illustrated in Figure 6.2.2.



Figure 6.2.2 Binder Type

1. FLAT BINDER

Use a flat rectangular surface to define the binder shape.

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

A detailed description of each function in BINDER dialog box is provided in the following sections.

### 6.2.1.1 FLAT BINDER

This function is used to create a flat binder, as illustrated in Figure 6.2.1. 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.

#### **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.

The following two options are used to define the orientation of a flat binder.

#### **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 6.2.3, 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.

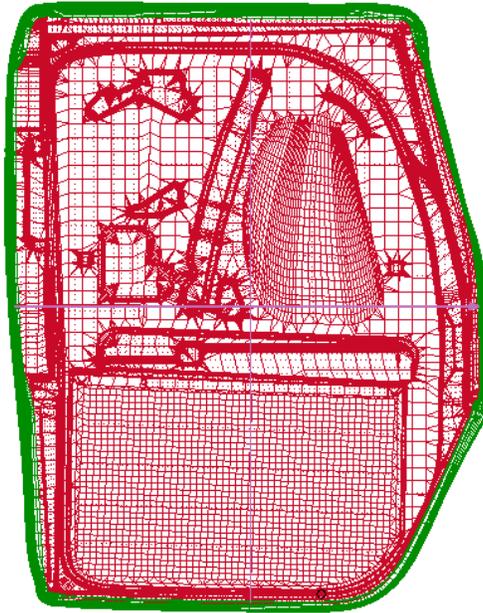


Figure 6.2.3 Two Control Lines

### 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.

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. Click the CREATE button at the lower right corner 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 10.2.1 for a detailed explanation on the TOOL part.
4. If a binder has already been created in the C\_BINDER part, a DYNAFORM QUESTION dialog box as illustrated in Figure 6.2.4 is displayed.

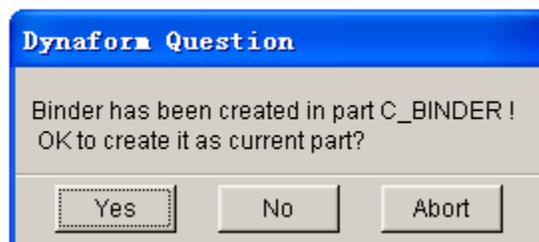


Figure 6.2.4 Dynaform Question dialog box

**YES** – Override previously created binder with the current definition.

**NO** – Keeps the previously created binder.

### CURRENT BINDER

This function is used to display the current binder part. If there are multiple binder parts existing in the current project, the user can click on  button to select the binder part to be modified or operated. Once this button is selected, a dialog box as illustrated in Figure 6.2.5 appears to prompt the user to select a binder part.

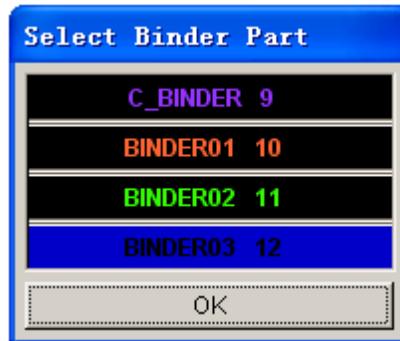


Figure 6.2.5 Select Binder Part dialog box

Once the binder is created, the edit functions are activated. The edit function are used to modify the binder, as shown in Figure 6.2.6.

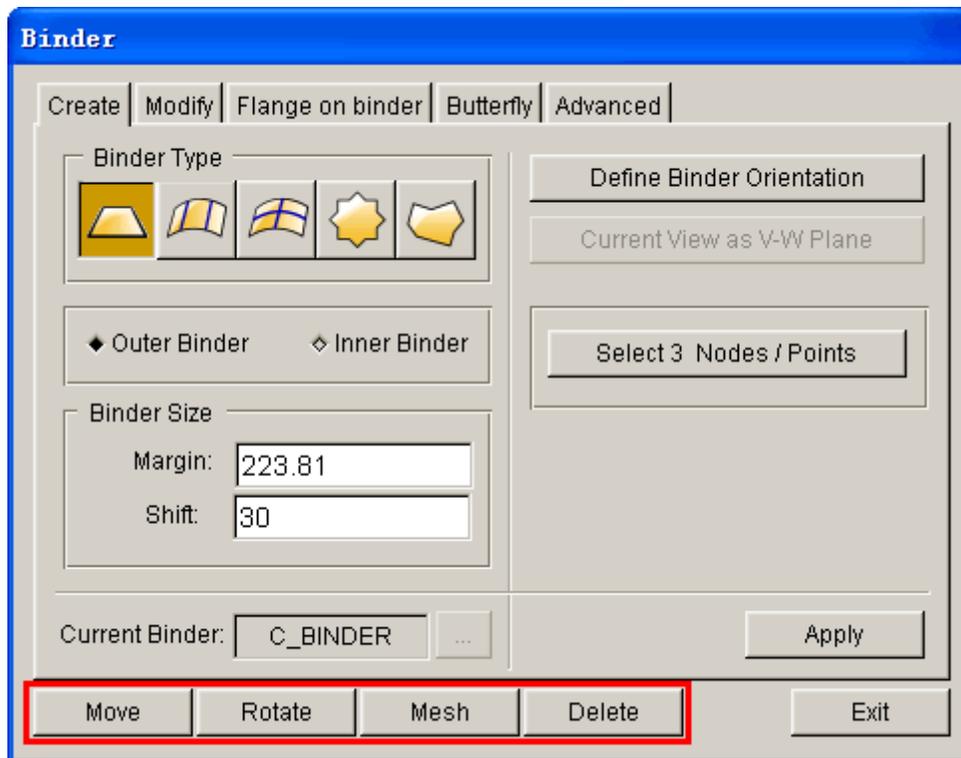


Figure 6.2.6 Binder Edit options

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 6.2.7.

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.

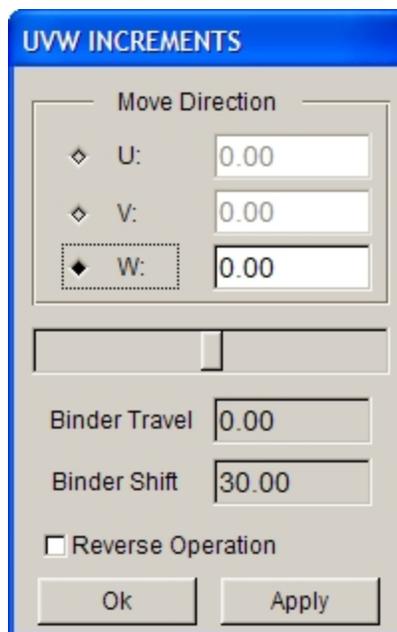


Figure 6.2.7 Move Binder dialog box

### 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 6.2.8. 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 click CREATE button to define the local coordinate system. The user can define the center of rotation by click on the CENTER OF ROTATION button, following by picking a location using mouse cursor.

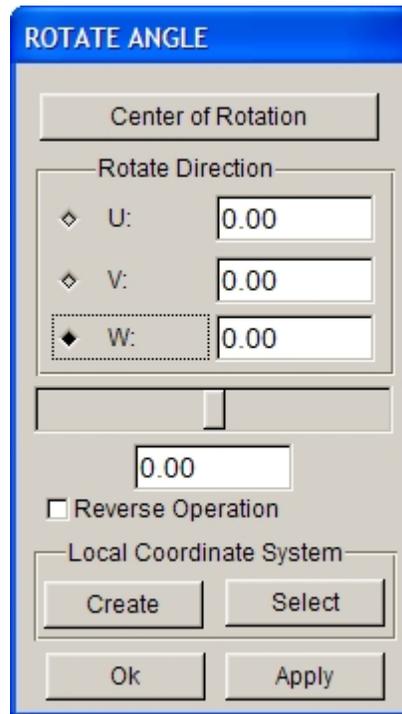


Figure 6.2.8 Rotate Binder dialog box

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 in the box right below the SLIDER bar as illustrated in Figure 6.2.8.
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 6.2.9 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.

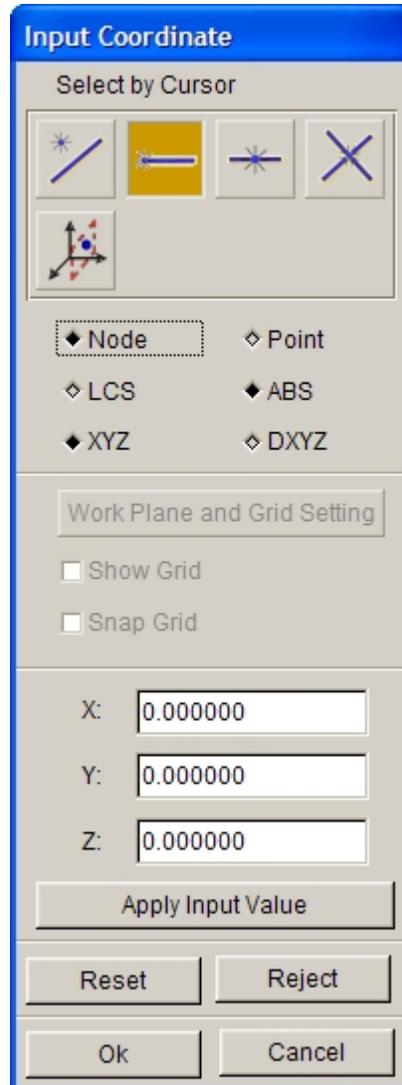


Figure 6.2.9 INPUT COORDINATE dialog box for selecting origin of the LCS

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.

Click on the OK button to dismiss the dialog box.

## MESH

This function is used to generate the binder mesh using the binder surface. A dialog box as illustrated in Figure 6.2.10 is displayed.

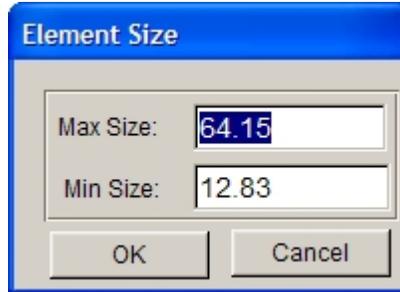


Figure 6.2.10 Element Size dialog box

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.

### 6.2.1.2 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 6.2.11, are activated.

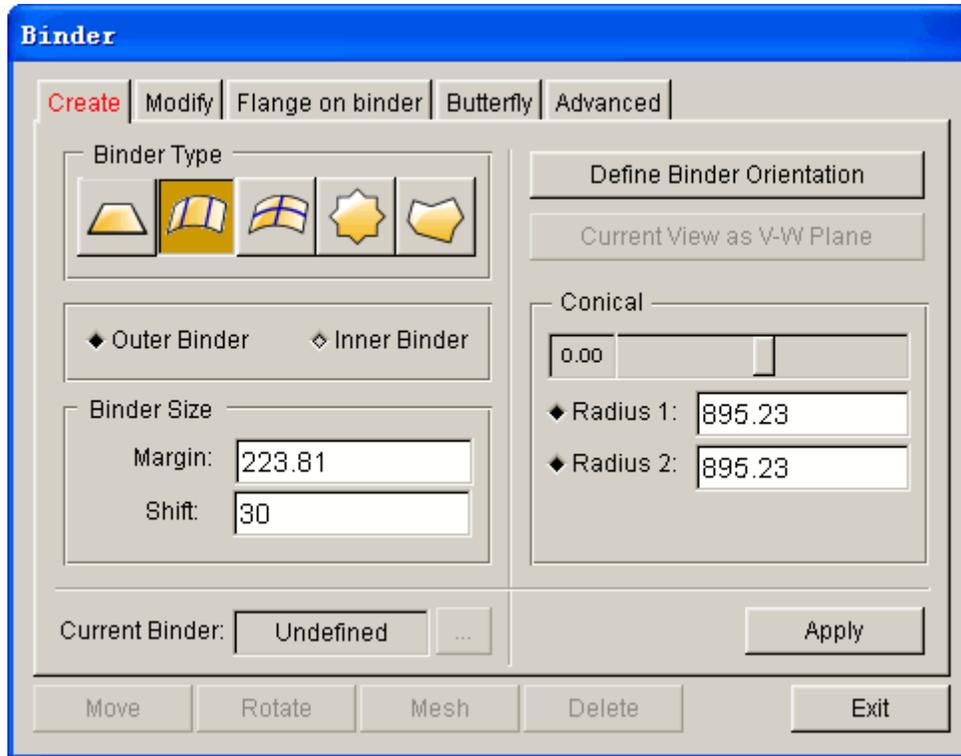


Figure 6.2.11 Conical Binder dialog box

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.

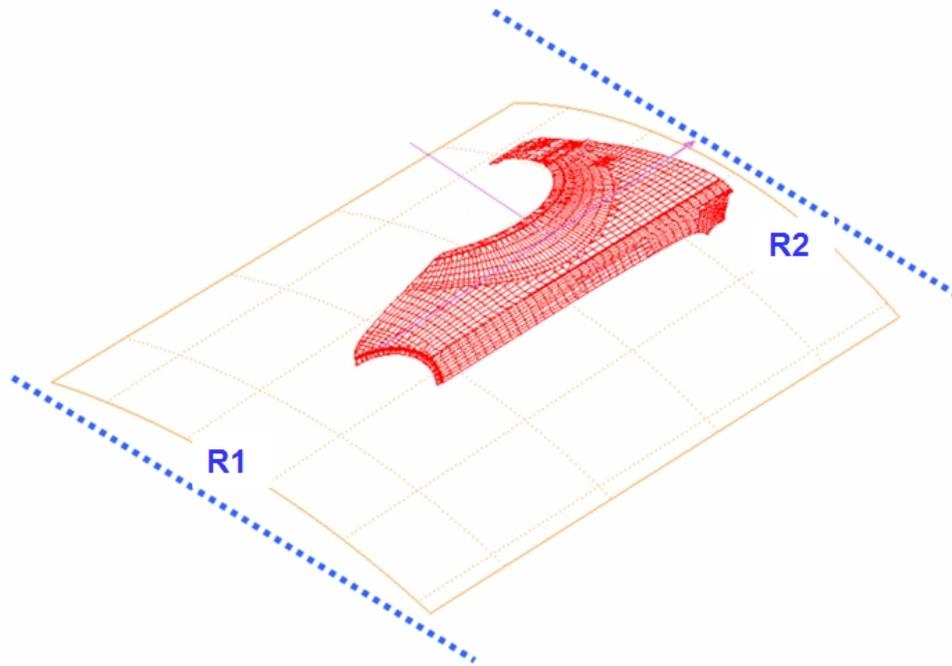


Figure 6.2.12 An illustration of generation of conical binder surface using two arcs

*NOTE: eta/DYNAFORM calculates the default conical Radius 1 and conical Radius 2 based on the size and shape of the part.*

### 6.2.1.3 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 6.2.13.

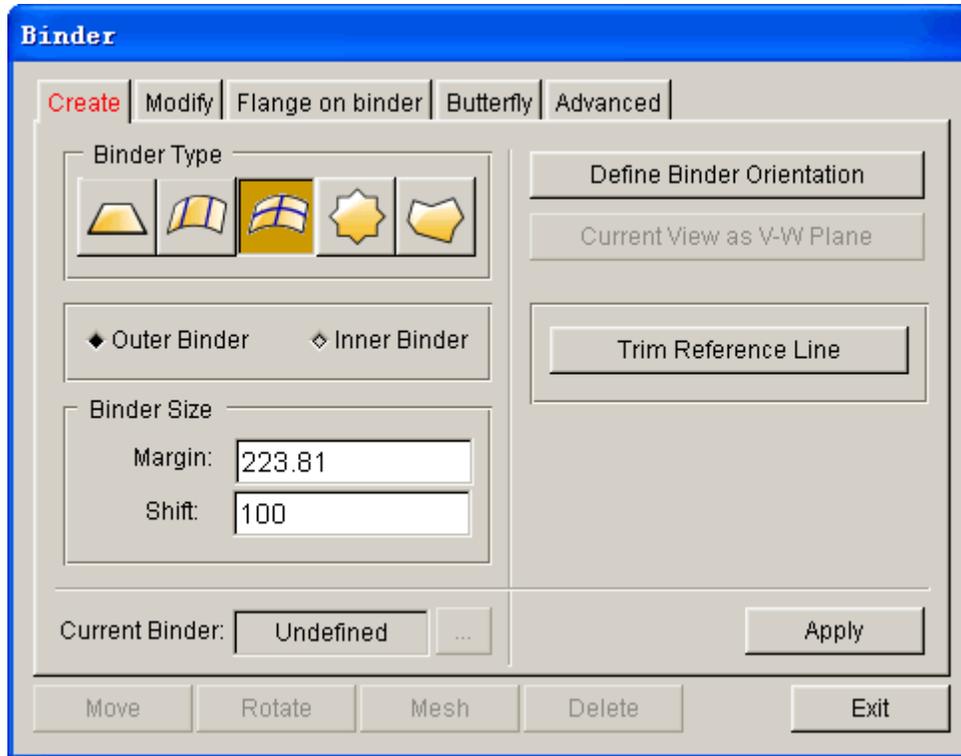


Figure 6.2.13 Two-line Binder dialog box

- **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 6.2.14, 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 6.2.15-6.2.17 illustrates binders created using different lengths of U-V axes.

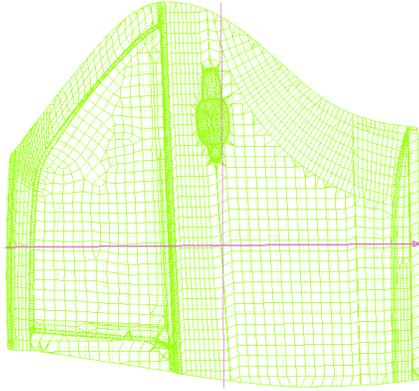


Figure 6.2.14 Two Control Lines



Figure 6.2.15 Two Line binder (before trimming)

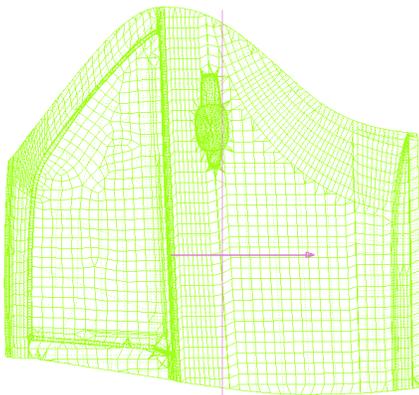


Figure 6.2.16 Trim Reference Line



Figure 6.2.17 Two Line Binder From Trimmed Reference Line

#### 6.2.1.4 BOUNDARY LINE BINDER

This function generates a binder surface using the boundary of part. If a region of part can be a region of binder, the user can select this function. The dialog box is illustrated in Figure 6.2.18.

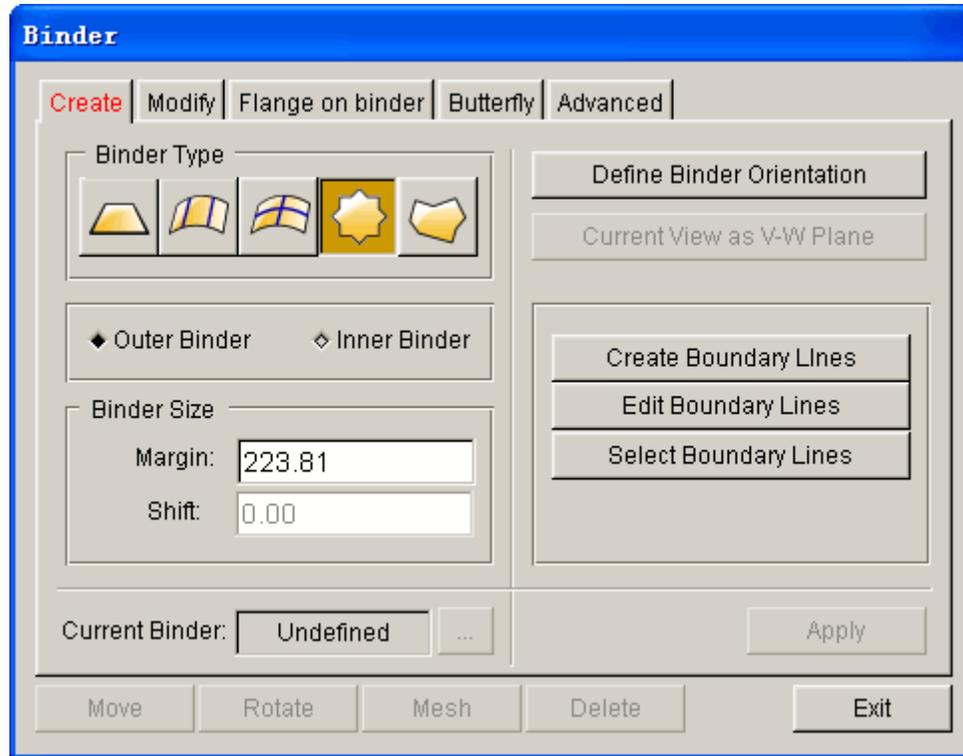


Figure 6.2.18 Boundary Line Binder

### 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 6.2.19 shows a typical orientation of the U-V axes relative to the part.

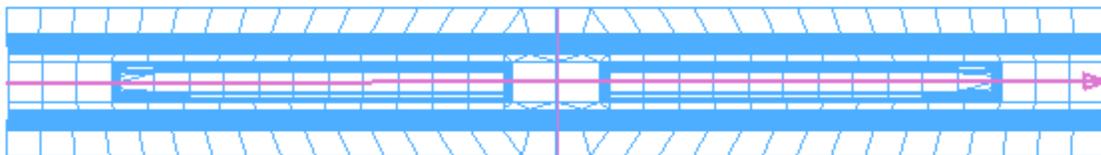


Figure 6.2.19 Define Binder Orientation

### CREATE BOUNDARY LINES

This function generates boundary lines of the part. The boundary lines are kept in the current part. Figure 6.2.20 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 6.2.21. The description of edit line functions is discussed in Section 5.1.

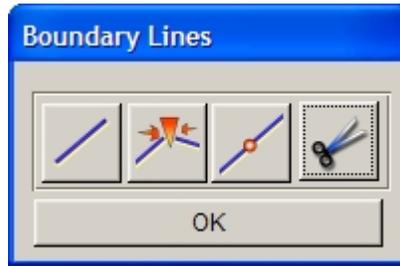


Figure 6.2.20 Edit Boundary Line dialog box

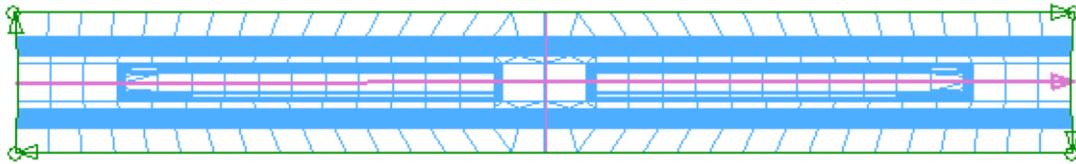


Figure 6.2.21 Created Boundary Line

### 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 6.2.22. Click on the CREATE button to generate the binder surface. The generated binder is illustrated in Figure 6.2.23.

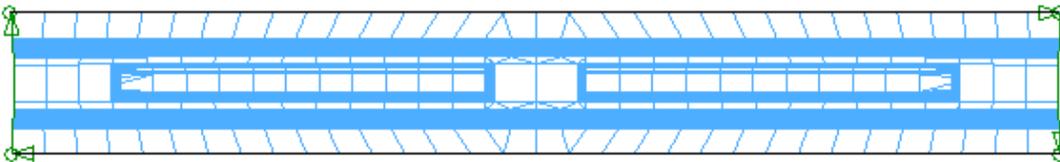


Figure 6.2.22 Selected Boundary Line

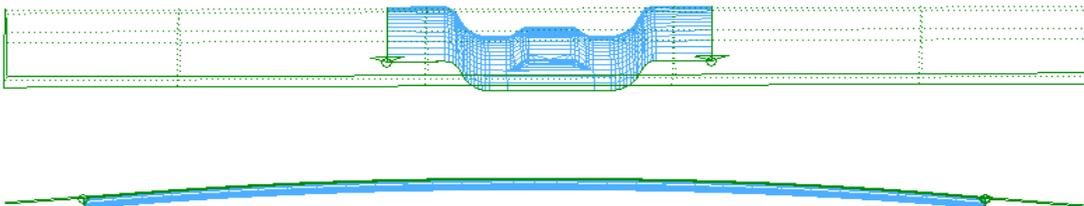


Figure 6.2.31 Created Binder

If the binder is not acceptable, the user may create another binder by editing the boundary.

### 6.2.1.5 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 6.2.24.

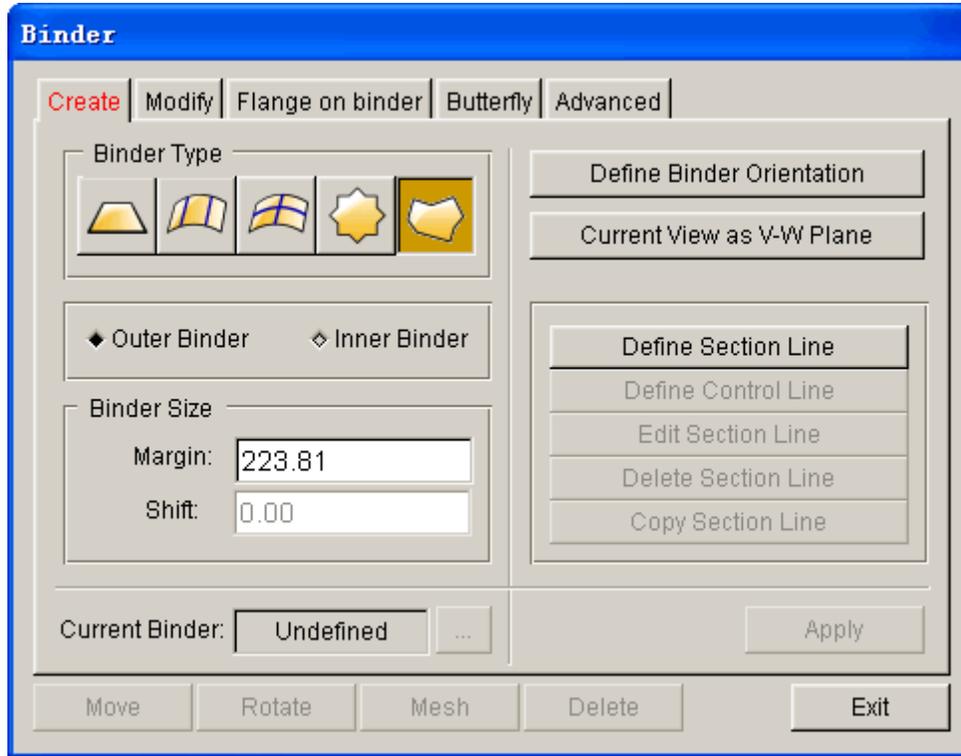


Figure 6.2.24 Free Form Binder dialog box

#### 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 6.2.19 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 6.2.25. 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***

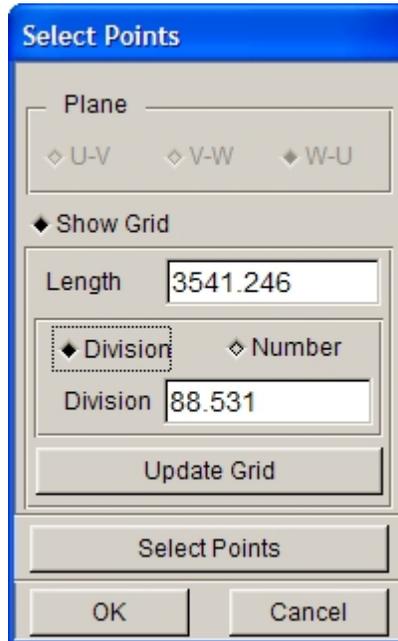


Figure 6.2.25 Selected points dialog box

**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 6.2.26 illustrates a typical section line defined on the screen.

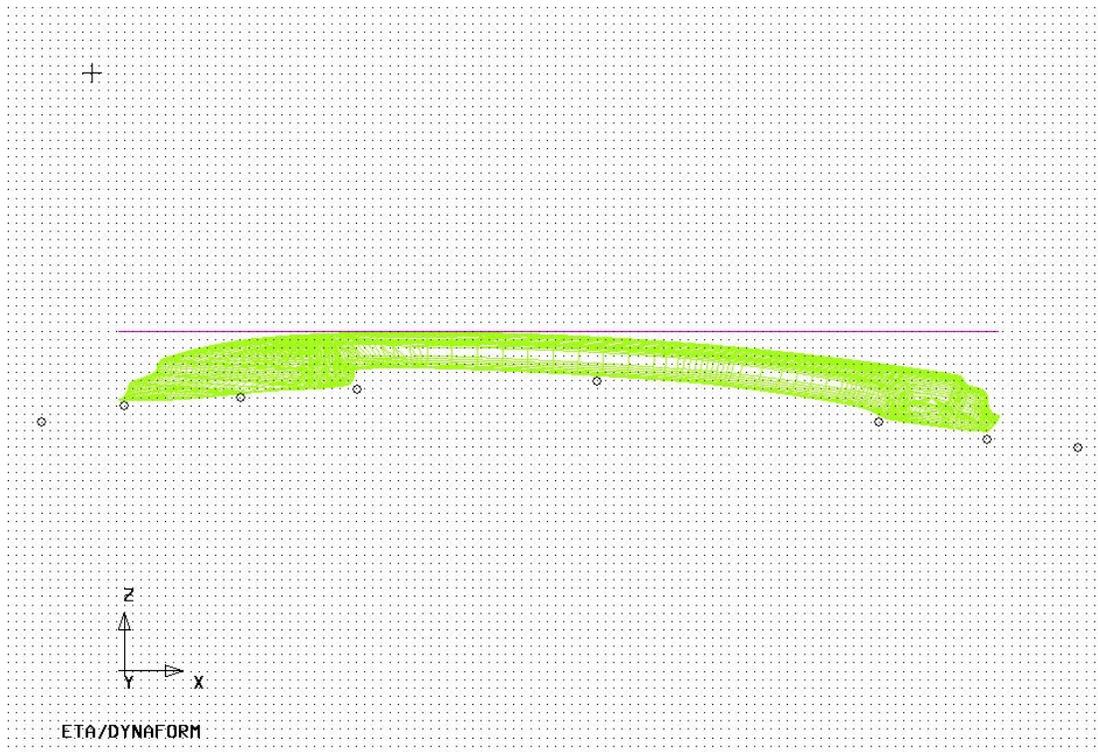


Figure 6.2.26 Typical Section Line

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 6.2.27 illustrates a typical control line joining the section lines.

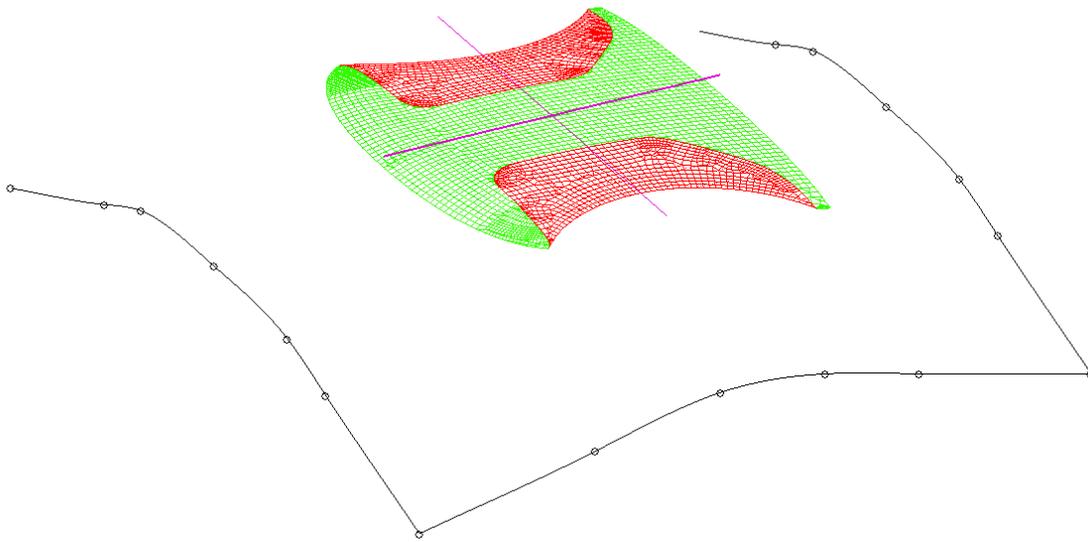


Figure 6.2.27 Typical Control Line Joining Section Lines

### **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 5.1.15.

### **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 that 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 6.2.28 illustrates a typical free form binder with a curved control line.

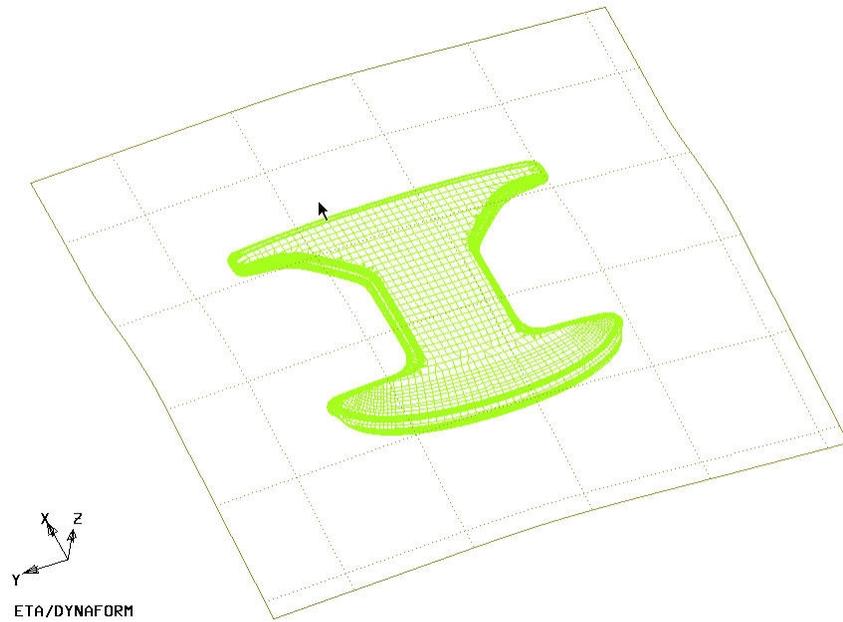


Figure 6.2.28 A Typical Free Form Binder with Control Line

## 6.2.2 MODIFY

On Modify binder interface, as illustrated in Figure 6.2.29, the user can easily modify the binder by adding control lines to change the binder to a desired shape.

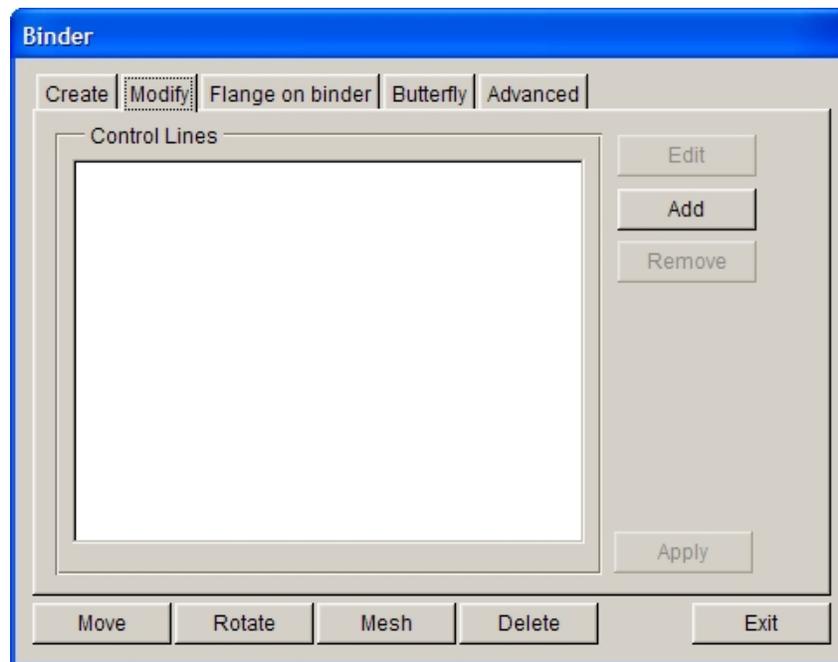


Figure 6.2.29 Modify Binder

- CONTROL LINES

This function is used to list the added control lines, as shown in Figure 6.2.31. In the list, point the cursor to any control line in the list and double click the left mouse button, the program will automatically display the corresponding serial number for all control lines for the user to identify. Therefore, the user can select the serial number to perform edit and delete operations.

- **ADD**

This function allows the user to add control lines on binder, as shown in Figure 6.2.29. Once this button is selected, three methods for adding control lines appear on the interface, as shown in Figure 6.2.30. When using the first method, the user can directly use the cursor to select a point on binder and then drag the mouse cursor to generate a control line. The cursor should be on the binder when selecting point. After the control line is added, EDIT and REMOVE buttons are activated, as shown in Figure 6.2.31. Another method is to select BY INPUT and activate DX, DY and VECTOR. DX indicates the X coordinate for distance with stamping center. DY indicates the Y coordinate for distance with stamping center. The user can input DX value, DY value or DX, DY value to accurately control the location of control line. Then, select VECTOR to define the vector direction of control line. In the end, select CREATE to generate a control line with user defined location, as shown in Figure 6.2.32.

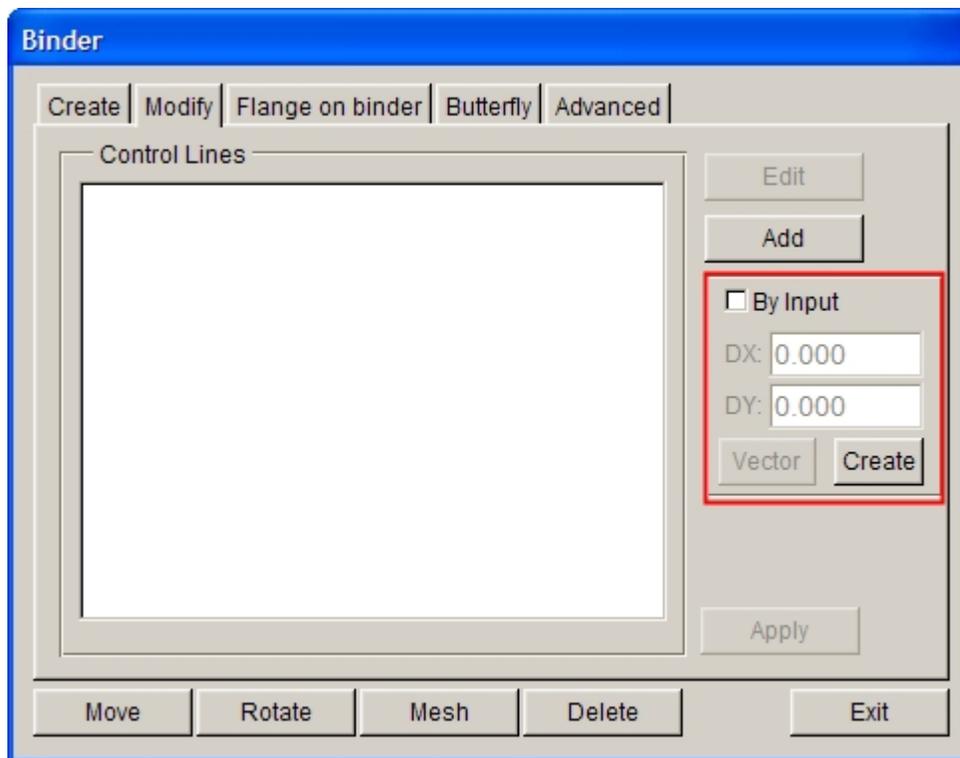


Figure 6.2.30 Add Control Line

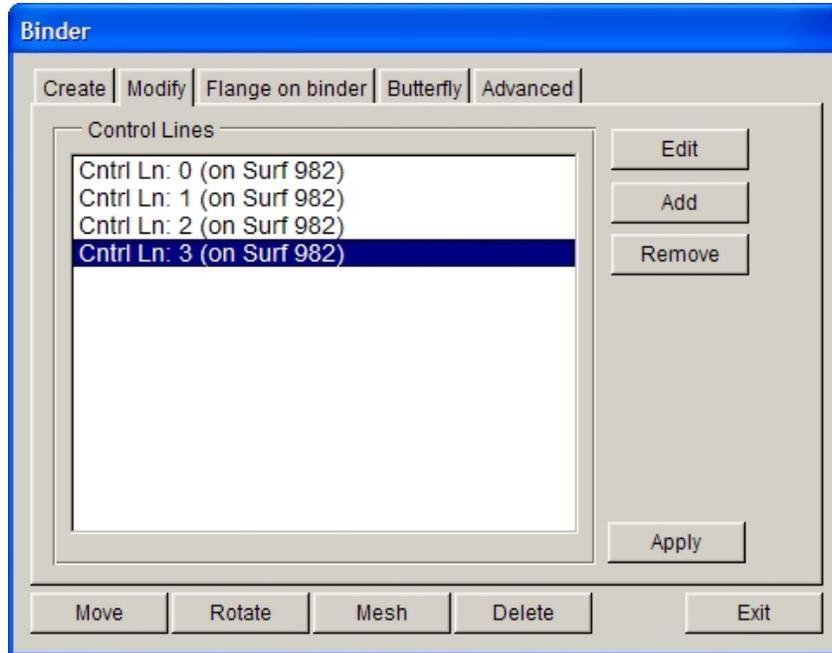


Figure 6.2.31 Control Line List

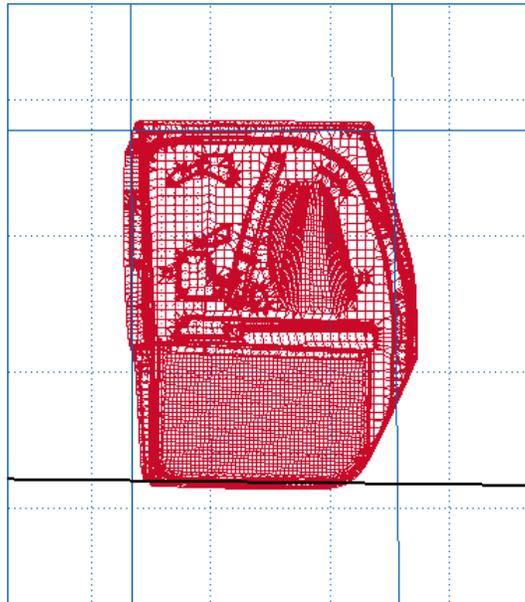


Figure 6.2.32 Add Control Line

*Note: When adding the control lines, it is suggested to use the top view. The location of control lines had better to be at the boundary of part so that the user can modify binder conveniently. In addition, APPLY can only be activated when there are intersecting control lines.*

- EDIT

This function allows the user to modify control lines. Select a control line in the Control Lines list and click

on this button to display a 2D window, as shown in Figure 6.2.33. In the 2D window, the control lines and part section lines for their locations are displayed. The user can change the binder by editing control lines so that the binder shape is more suitable for the part. After editing a control line, the user can directly select another control line in 3D window by cursor for editing in the 2D window without exiting the 2D window.

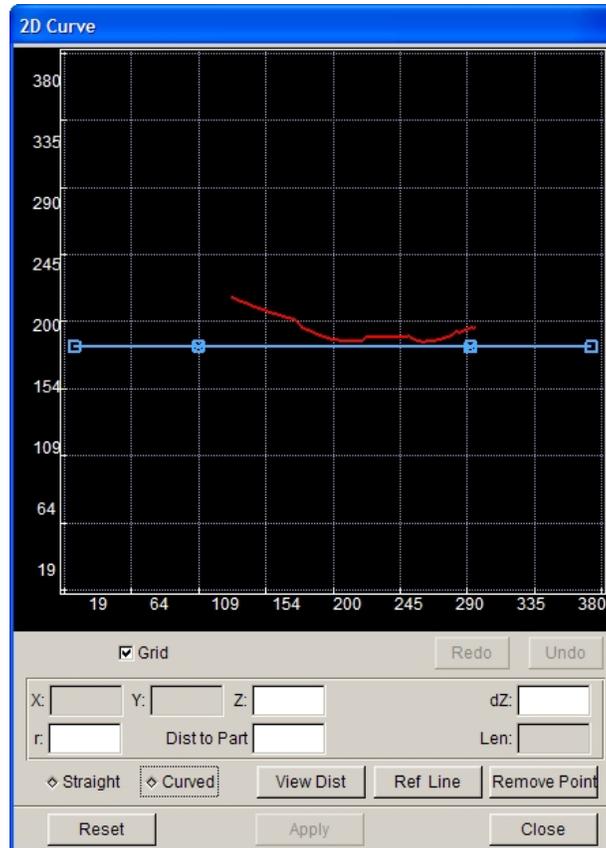


Figure 6.2.33 2D Curve window

The functions for 2D window are described below:

In the 2D window, the user can add control points on control lines by simultaneously pressing Shift key and clicking the left mouse button, as shown in Figure 6.2.34; move the locations displayed in 2D window for control lines and section lines by simultaneously pressing Ctrl key and clicking the middle mouse button, as shown in Figure 6.2.35; and scale the control lines and section lines by simultaneously pressing Ctrl key and clicking the right mouse button, as shown in Figure 6.2.36 and Figure 6.2.37.

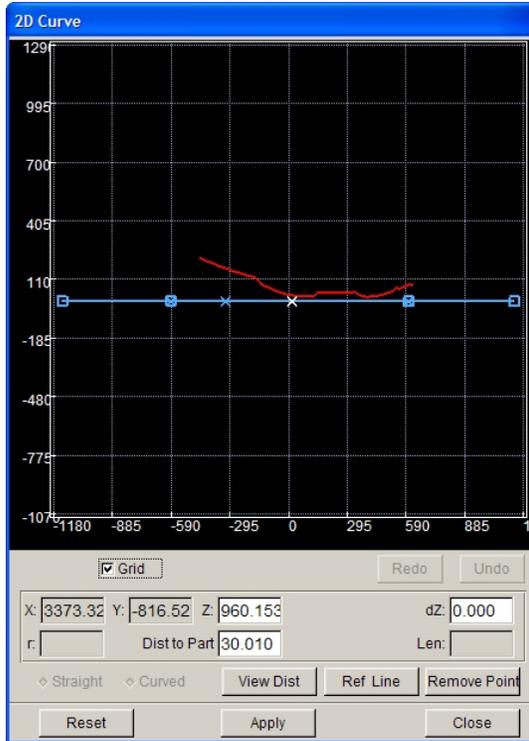


Figure 6.2.34 Add Control Points



Figure 6.2.35 Move Control Lines and Section Lines

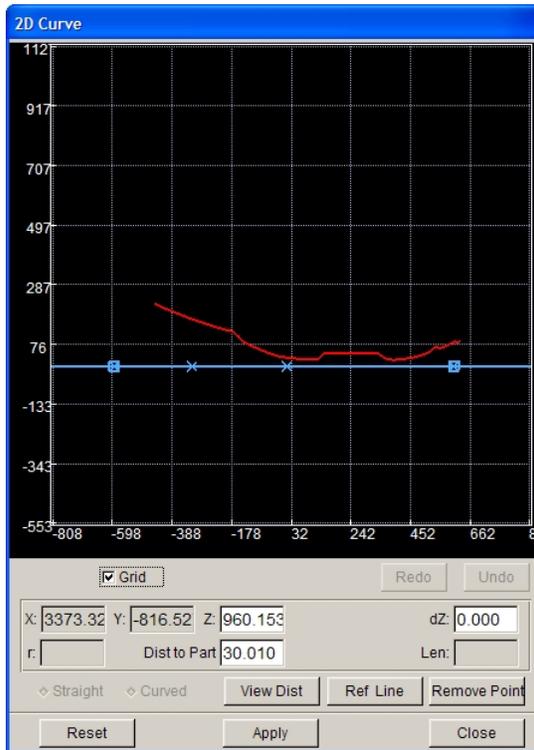


Figure 6.2.36 Zoom in Control lines and Section Lines

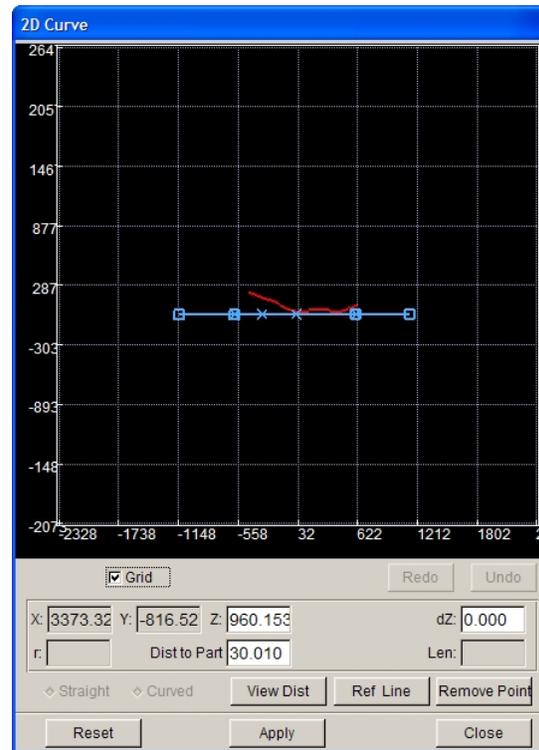


Figure 6.2.37 Zoom out Control lines and Section Lines

**GRID**

Display the grid in 2D window. If this option is toggled off, the grid will not be displayed.

**REDO**

Redo the latest operation. If there is nothing to redo, this button is disabled.

**UNDO**

Cancel the latest operation. If there is nothing to cancel, this button is disabled.

**X, Y, Z**

Display the coordinates of the selected control points. The user can enter the Z coordinate value manually and press ENTER to confirm.

**dZ**

Display the relative distance between the location after moving and the initial location of the selected control point along Z direction upon the global coordinate system.

**R**

This function is activated when two sides of the selected control points are straight lines. The user can enter the radius value manually and press ENTER to confirm. The program will generate an arc, as shown in Figure 6.2.38.

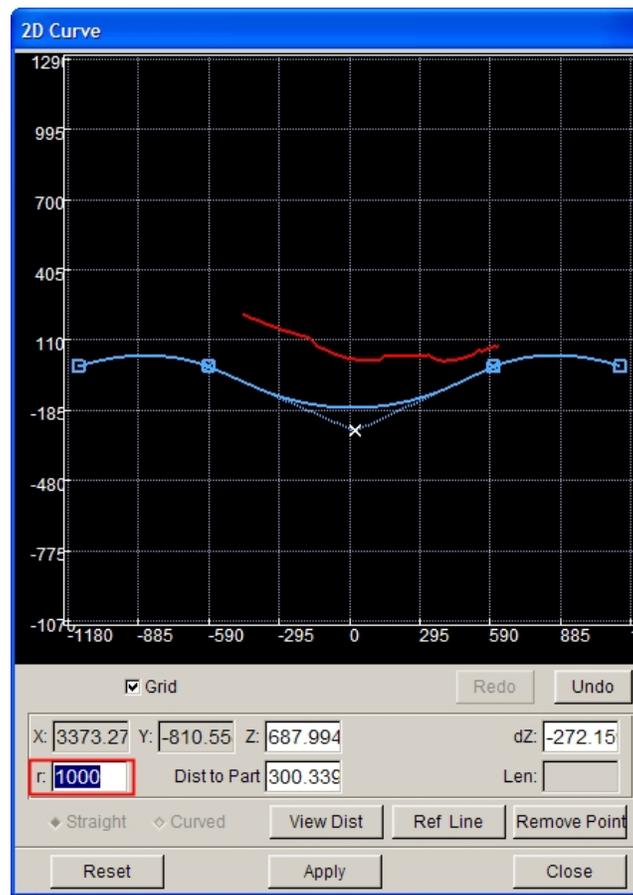


Figure 6.2.38 Generate Arc

**DIST TO PART**

Display the distance between control point and part. The user can enter the distance value manually and press ENTER to confirm. The selected control point will move to the entered distance location.

**LEN**

Display the length of the selected line segment.

**STRAIGHT**

This option is activated after a line segment is selected. If the selected line segment is curve, the user can select this option to change it to a straight line.

**CURVED**

This option is activated after a line segment is selected. If the selected line segment is straight, the user can select this option to change it to a curve.

**VIEW DIST**

Measure the view distance. Select this option to measure the distance between two points in the 2D window by mouse cursor, for example, the distance between control line and part section line at the location for control line, as shown in Figure 6.2.39. The distance value will be shown in the displayed dialog box as illustrated in Figure 6.2.40. The user can also use this function to measure the distance between two points in the 3D window.

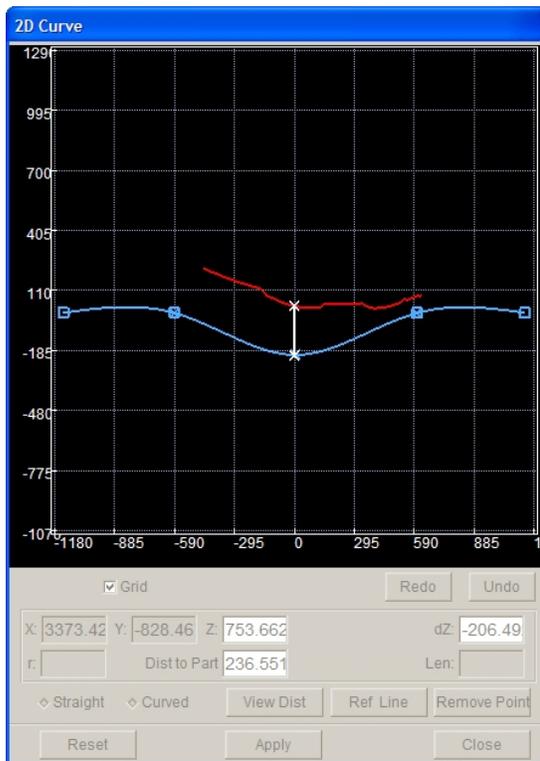


Figure 6.2.39 Measure View Distance

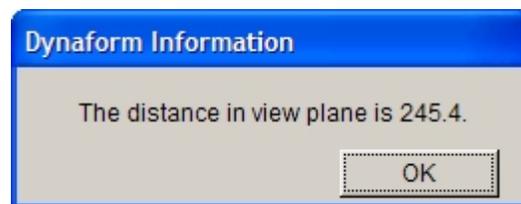


Figure 6.2.40 Display View Distance

**REF LINE**

Display the user selected line as the reference line for control line shape. The user can generate part

boundary lines by Create Boundary Line function described in Section 5.1.5. Then select the part boundary line as the reference line for control line shape, as shown in Figure 6.2.41.

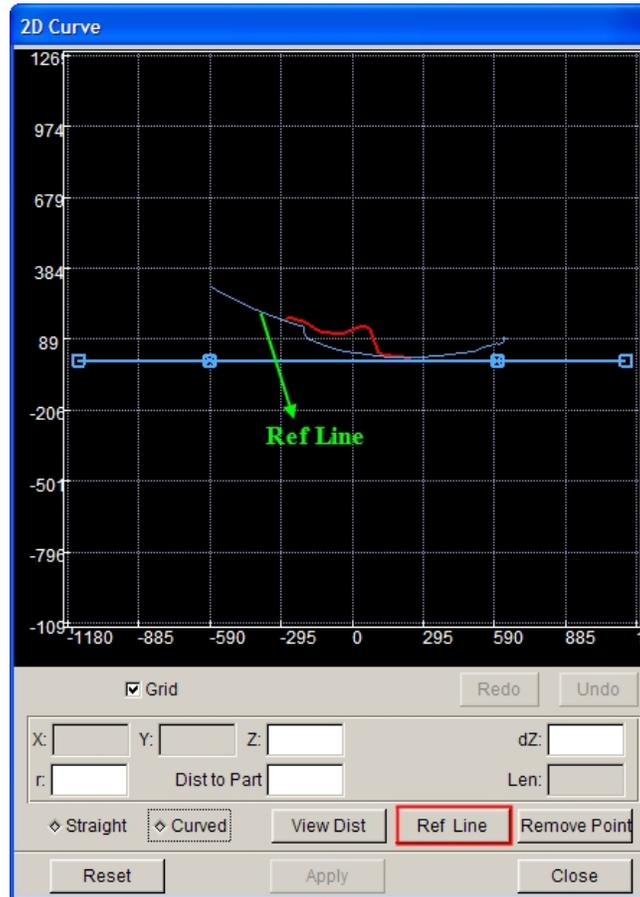


Figure 6.2.41 Display Reference Line

### REMOVE POINT

Remove the selected control points.

### RESET

Restore the latest setting to the previous one.

The user can change the shape of control lines via moving the control point by cursor, as shown in Figure 6.2.42. When moving the control point, the control lines on screen will highlight the moved control points, as shown in Figure 6.2.43.

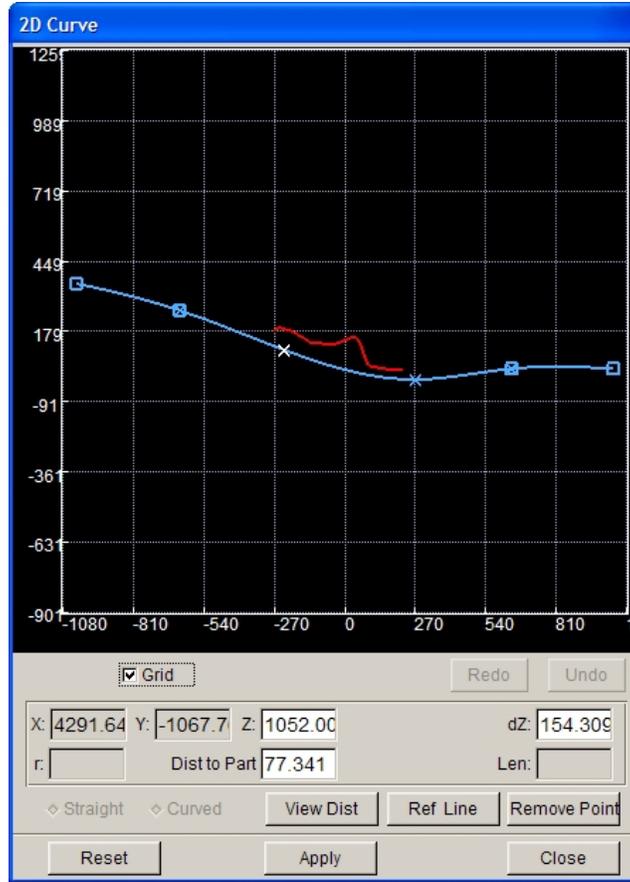


Figure 6.2.42 Move Control Point

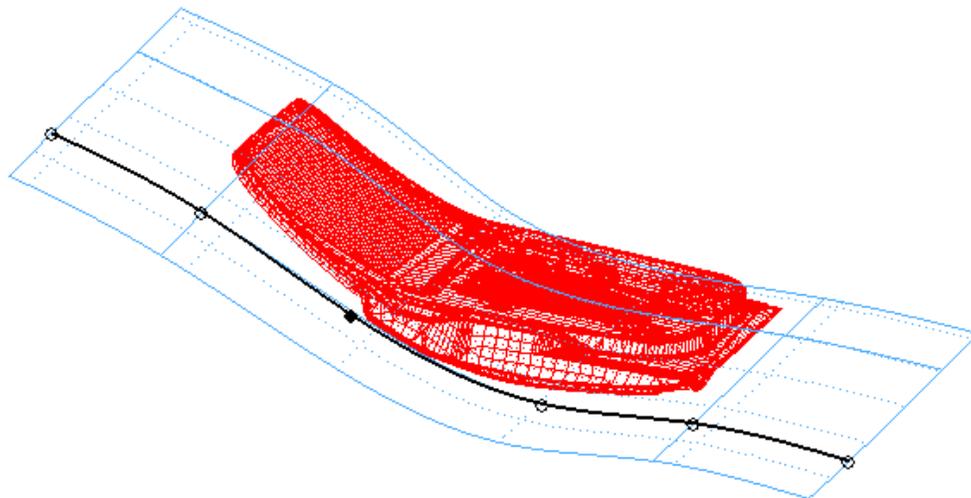


Figure 6.2.43 Display the Moved Control Points

- REMOVE

This function allows the user to delete the unnecessary control lines.

After completion of editing for all the added control lines, click on APPLY on Modify Binder interface. The binder will change in accordance with the shape of control lines, as shown in Figure 6.2.44.



Figure 6.2.44 Binder Morphing Result

### 6.2.3 FLANGE ON BINDER

This function enables the user to paste the flange on the part to the edited binder. The interface is shown in Figure 6.2.45.

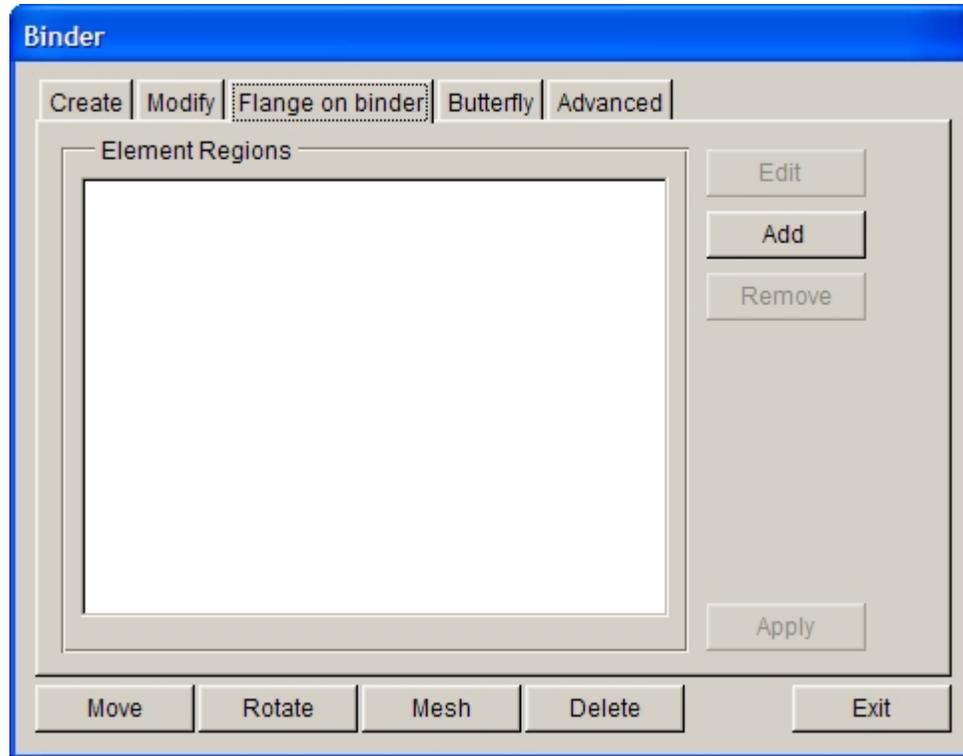


Figure 6.2.45 Flange on Binder

- CONTROL LINES

This function is used to list the added information for the selected element region and display the number of selected elements in this region.

- ADD

This function allows the user to define the region to be pasted on binder by selecting elements on part, as illustrated in Figure 6.2.46.

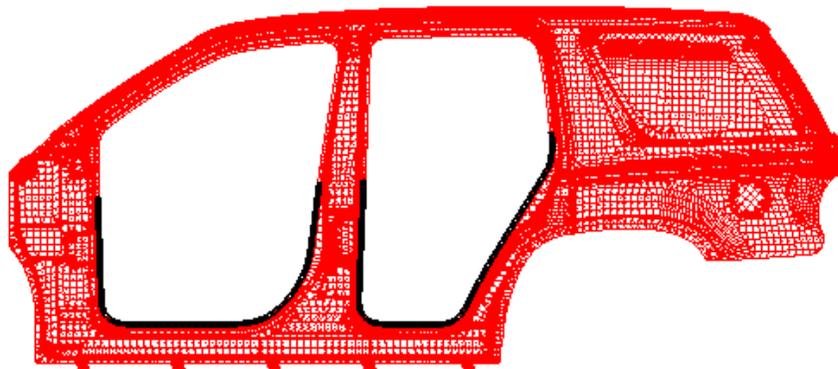


Figure 6.2.46 Select Element Region in Part

- EDIT

This function allows the user to modify element region. Select an item to be modified from the Element

Regions list and click on this button to display the Select Element dialog box. The user can add or cancel the selected elements to modify the element region.

- REMOVE

This function allows the user to delete the unnecessary element region.

Click on APPLY to paste the selected element on part to binder, as shown in Figure 6.2.47.

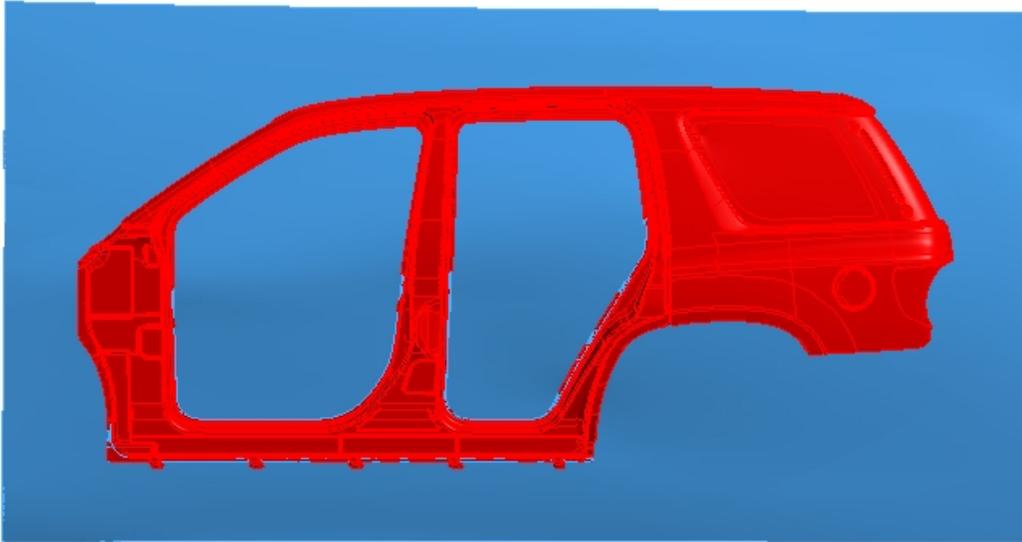


Figure 6.2.47 Flange on Binder Result

*Note: When using this function, the binder must be an un-trimmed surface.*

## 6.2.4 BUTTERFLY

This function enables the user to flip certain regions of binder to change the binder to a desired shape.

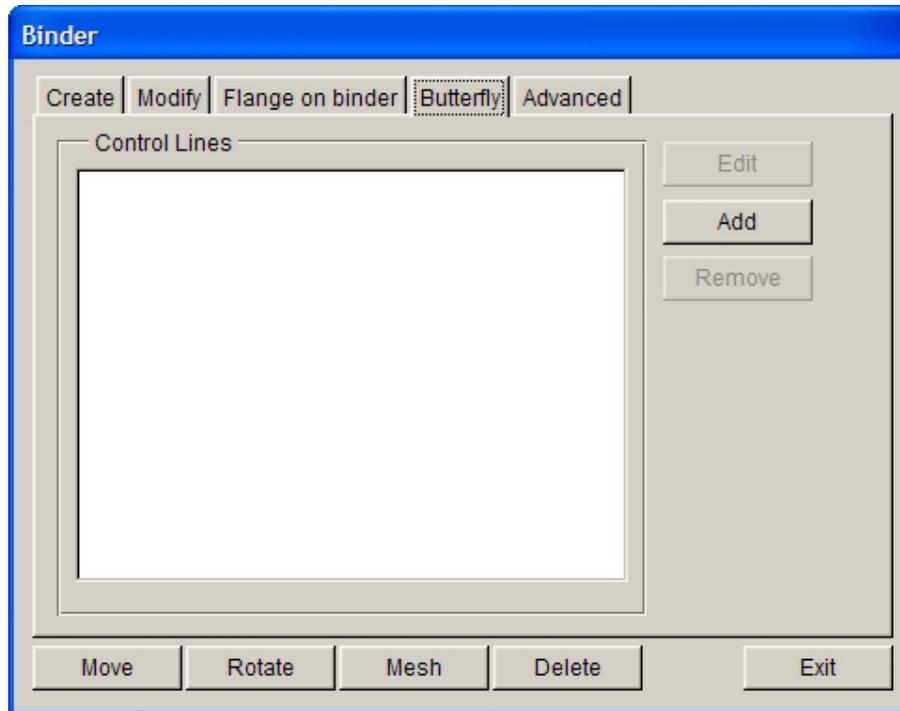


Figure 6.2.48 Butterfly interface

- CONTROL LINES

This function is used to list the added control lines information. Select an arbitrary control line in the list by cursor and double click on the left mouse button. The program will automatically display the corresponding serial numbers for all control lines for the user to identify the serial number of control lines in the list. There fore, the user can select this serial number to perform edit and delete operations.

- ADD

This function allows the user to add control lines on binder. Click on this button and select two points on binder to generate a control line. The program will simultaneously pop up a 2D window, as shown in Figure 6.2.49. In this 2D window, it will display the section line on the control line and the part section line cut by the section line. The user can change the binder by editing the section line so that the binder will be more suitable for the part.

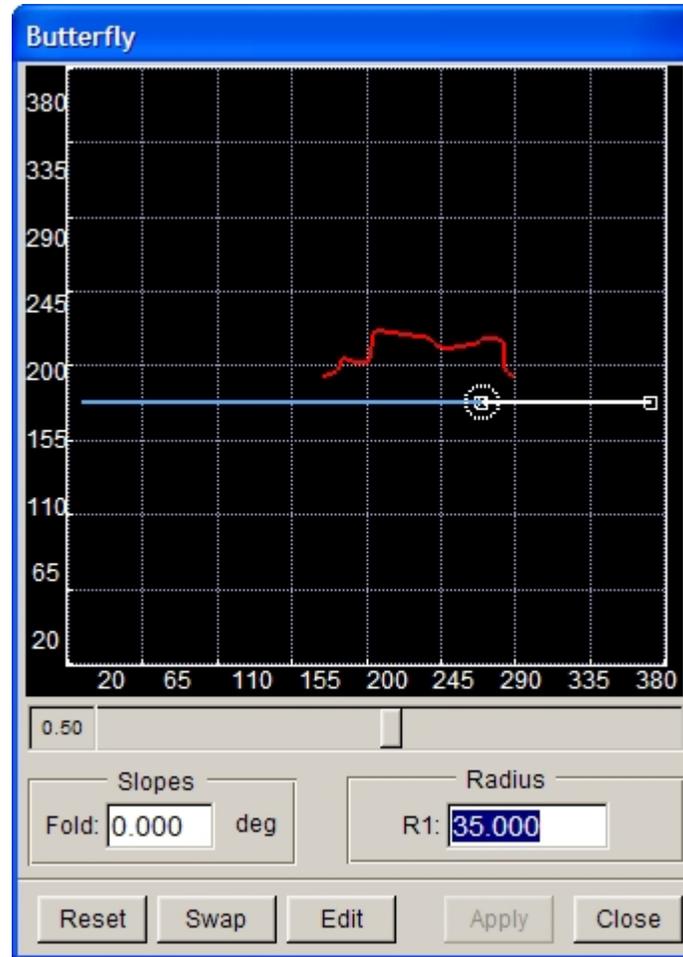


Figure 6.2.49 Butterfly 2D Window

- EDIT

This function allows the user to edit control lines. Select the control lines to be edited in Control Lines list and the 2D window appears. The user can change the parameters in this window to re-edit the flipped region of the binder.

- REMOVE

This function allows the user to remove the unnecessary control lines and restore the flipped regions on binder simultaneously.

The functions for 2D window are described below.

The user can change the shape of part section line cut by the section line on control line via moving the slider. The user can enter the FOLD value and click on ENTER key to set the angle between the section line on control line and the binder, or drag the endpoint of section line by moving the cursor up and down to control the size of the control angle. The user can enter the RADIUS value and click on ENTER key to set the radius of transition surface for the flipped region of binder, or directly drag the circle by cursor to control the radius. Click on SWAP button to change the direction of section line, as shown in Figure 6.2.50(b). The user can drag the endpoint of section line to change the cutting range of it by clicking on EDIT button.

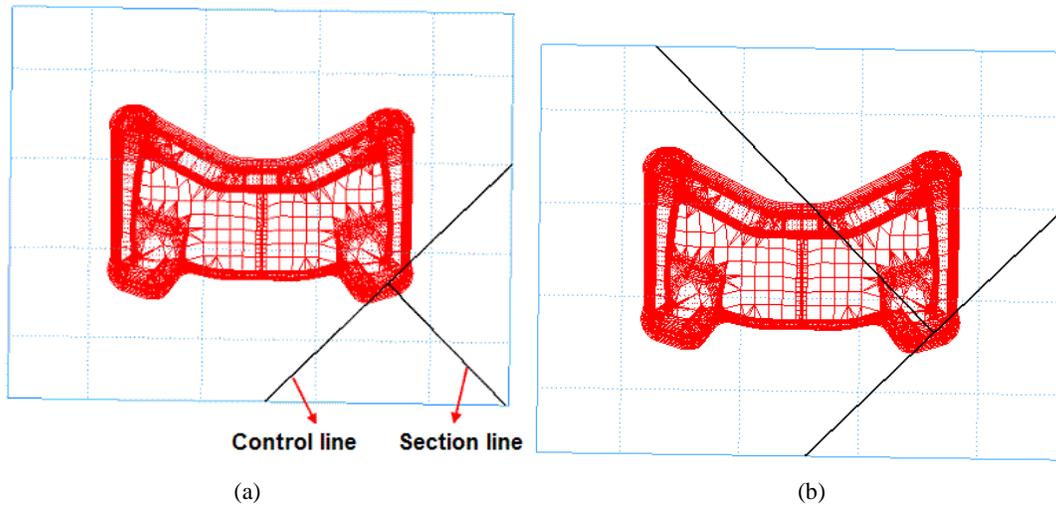


Figure 6.2.50 Edit Section Line

## 6.2.5 ADVANCED

There are some added functions of surface auxiliary operation on this interface for the user to edit the binder. These added functions contains: MIRROR, COPY, SCALE, SPLIT, TRIM, UNTRIM, RESPACE U-V LINE, EXTEND, EDIT SECTION LINE, as illustrated in Figure 6.2.51. Refer to Section 5.2, Surface, for detailed description on all these functions except EXTEND and EDIT SECTION LINE functions.

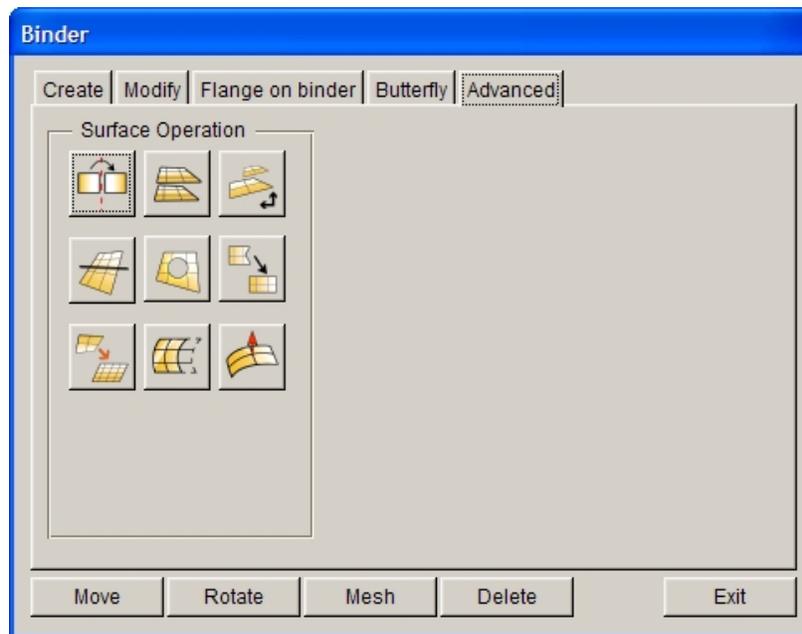


Figure 6.2.51 Advanced functions

The EXTEND and EDIT SECTION LINE functions are described below.

- **EXTEND**

This function allows the user to extend the binder surface. The operation of this function is similar to the

one described on Section 5.2.15, Extend Surface.

However, there is a difference from the function in Section 5.2.15. After completion of extend operation for binder, the user clicks on APPLY to display a Dynaform Question dialog box to determine if control lines on binder should be deleted, as shown in Figure 6.2.52. Select YES to delete control lines or select NO to keep control lines.

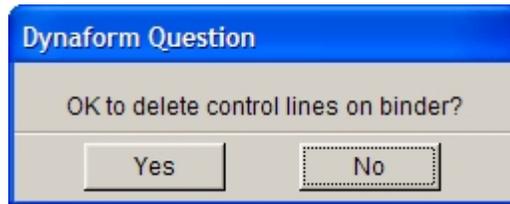


Figure 6.2.52 Dynaform Question dialog box

Note: When using the EXTEND function, the extended surface must be the binder of an un-trimmed surface.

- **EDIT SECTION LINE**

This function allows the user to modify binder by editing the section line of surface. Once this option is selected, the program will highlight the binder boundary. The user can select two points on the binder boundary by mouse cursor to create a section line. Then, drag the section line by mouse cursor to change the shape of binder.

## 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. In addition, positioning the cursor on an addendum and double clicking the left mouse button, the program will display the corresponding number of the addendum automatically. This function allows the user to identify the corresponding number of each addendum so that the user can select the number to modify or delete, etc.

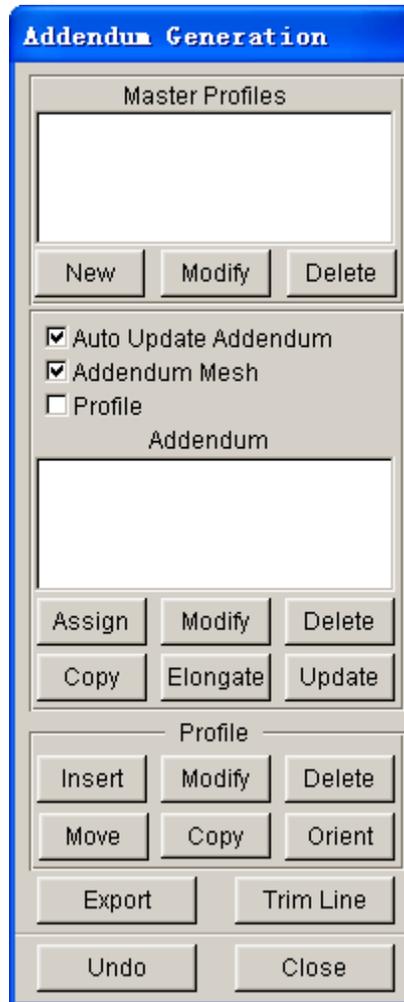


Figure 6.3.1 Addendum Generation dialog box

**NOTE:**

- 1. The binder must be defined prior to using the ADDENDUM functions.*
- 2. Once the ADDENDUM menu is initiated, three new parts is automatically created in the database: POP\_LIN, PROFILE and ADDENDUM.*

### 6.3.1 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 6.3.2 illustrates a Type 1 master profile with two radii.

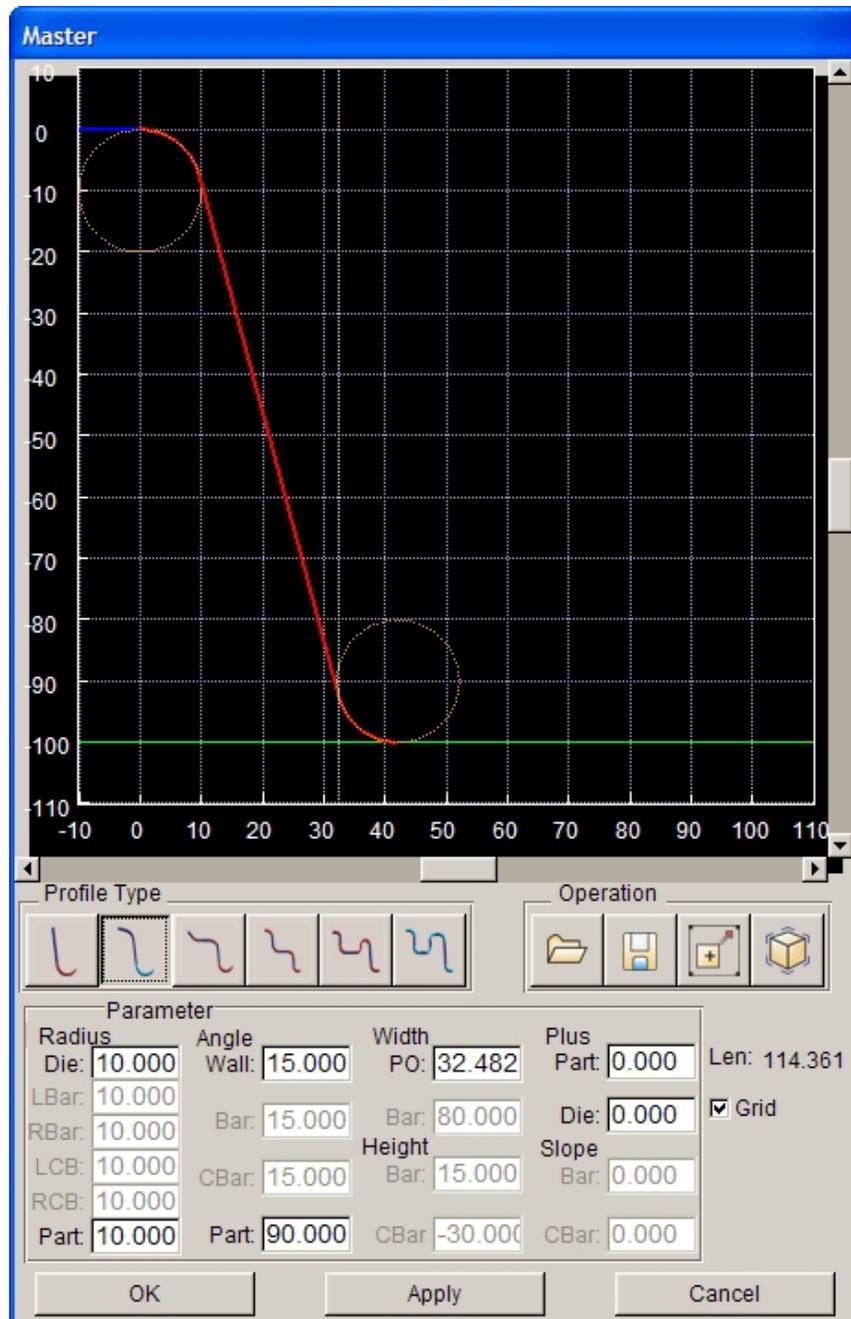


Figure 6.3.2 Master Profile window

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 6.3.3, 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 data fields.

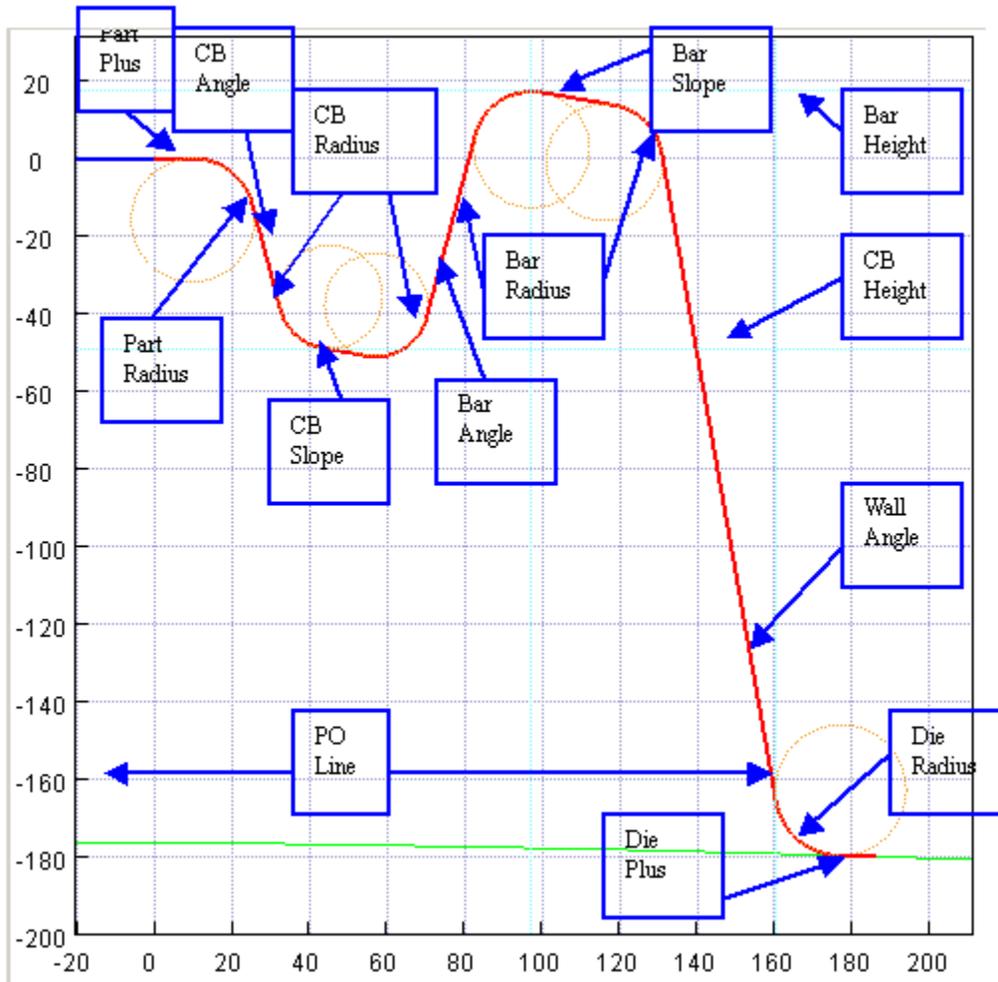


Figure 6.3.3 Various Addendum Parameters

**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:** Determine the width of the drawbar.

**PO Line:** Determine the position of the punch opening line (PO Line).

#### HEIGHTS:

**BAR:** Determine the height of the drawbar.

**CBAR:** Determine 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 6.3.4, 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.

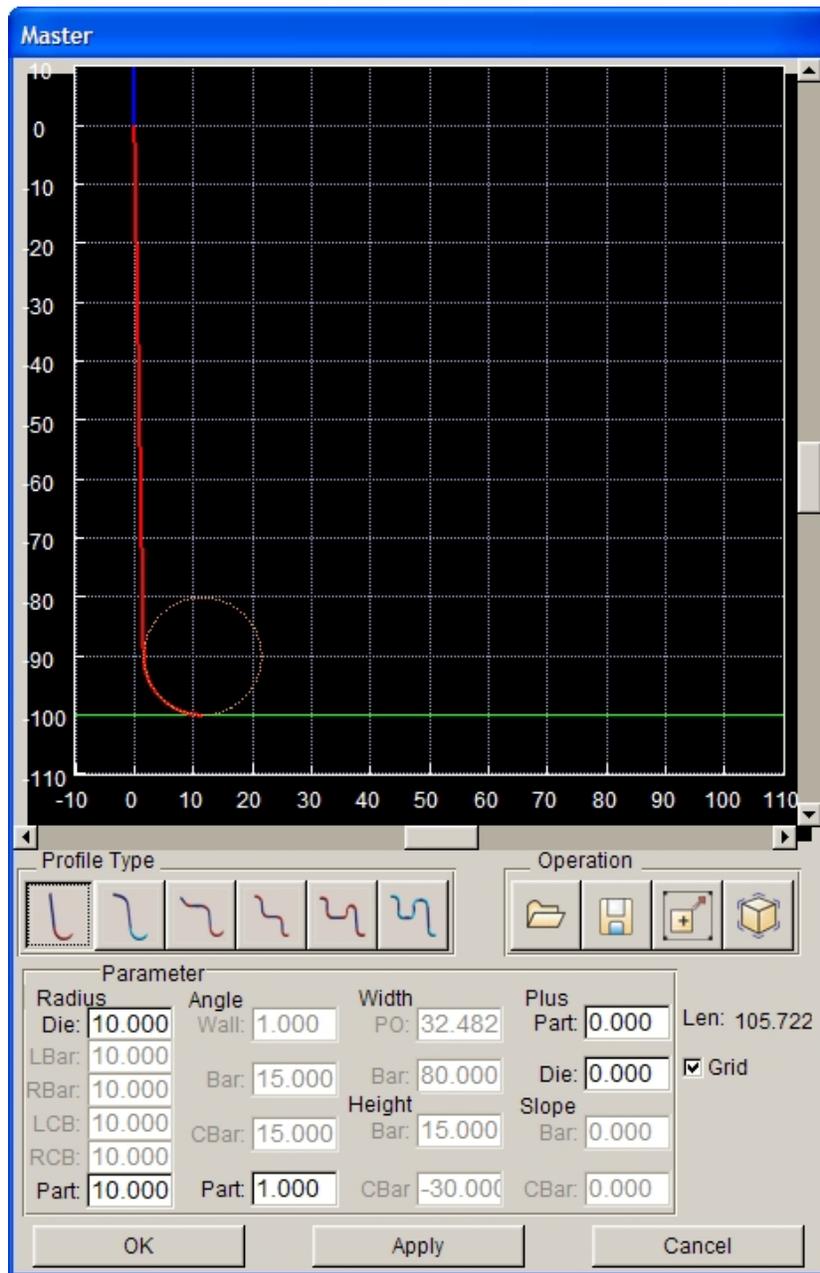


Figure 6.3.4 Profile Type 1 window

**TYPE 2**

The TYPE 2 profile produces the shortest distance between the part boundary and the punch opening line. As illustrated in Figure 6.3.5, the BAR WIDTH, BAR HEIGHT and CB HEIGHT are disabled in TYPE 2 profile.

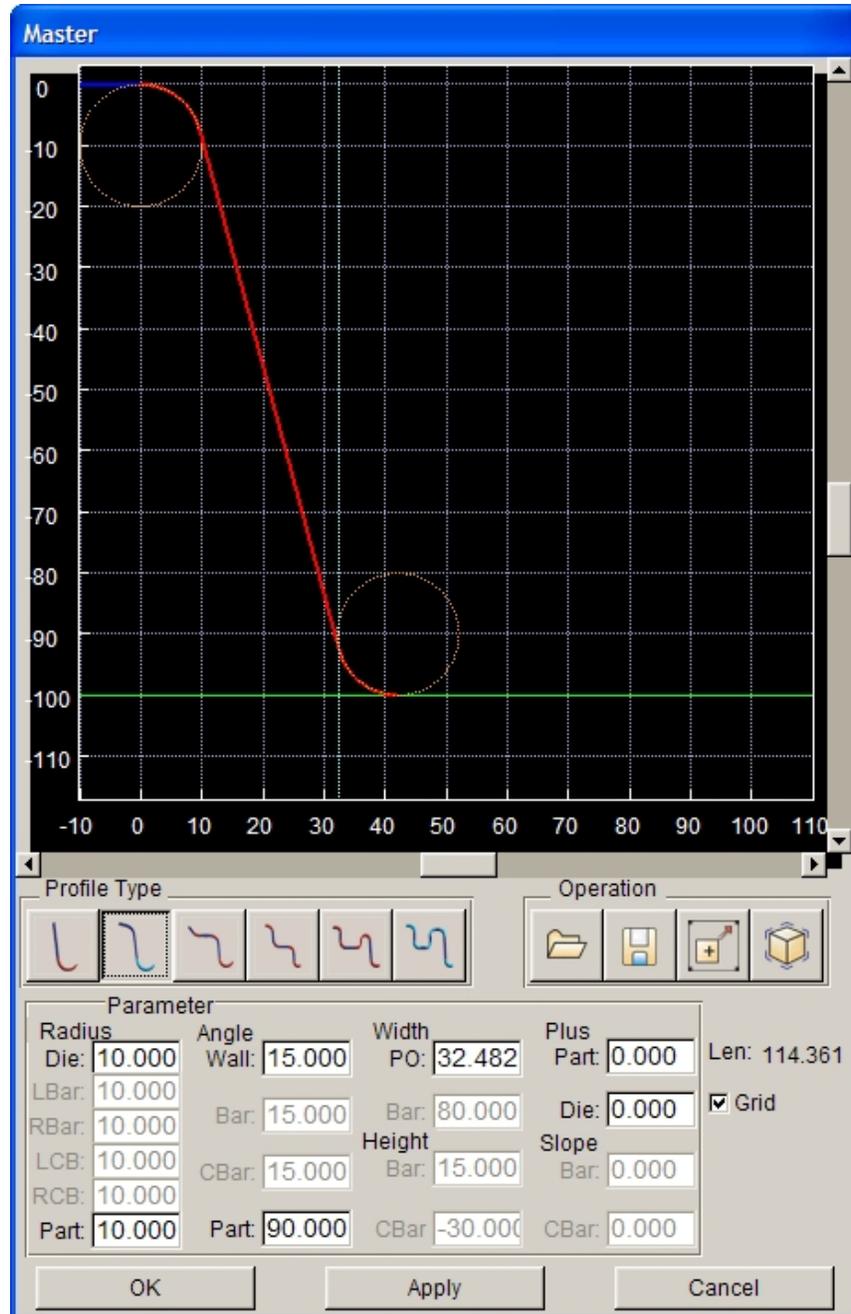


Figure 6.3.5 Profile Type 2 window

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 6.3.6. 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.

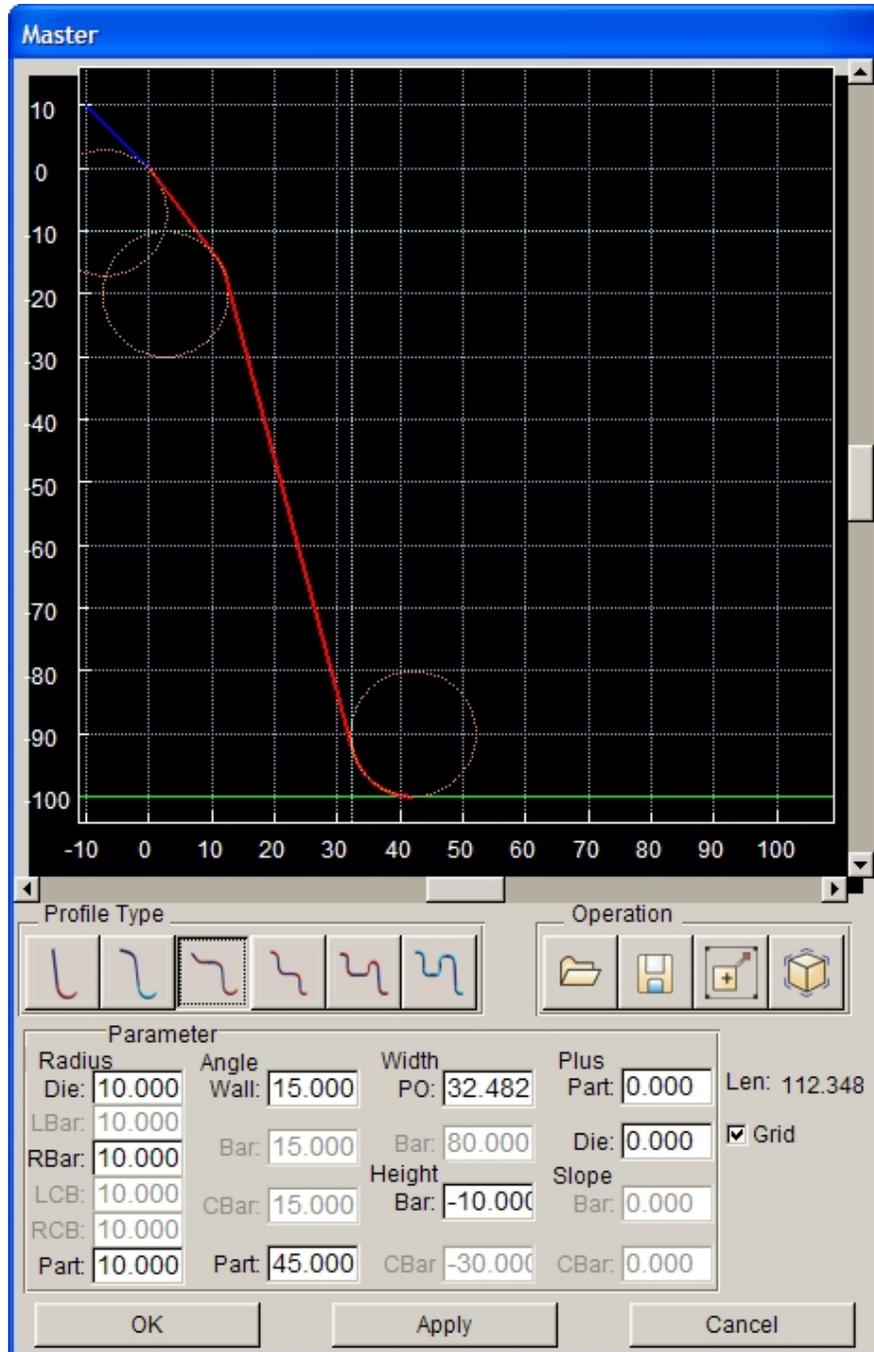


Figure 6.3.6 Profile Type 3 window

**TYPE 4**

The TYPE 4 profile is illustrated in Figure 6.3.7. CB HEIGHT is measured from the part boundary. Both BAR WIDTH and BAR HEIGHT are disabled.

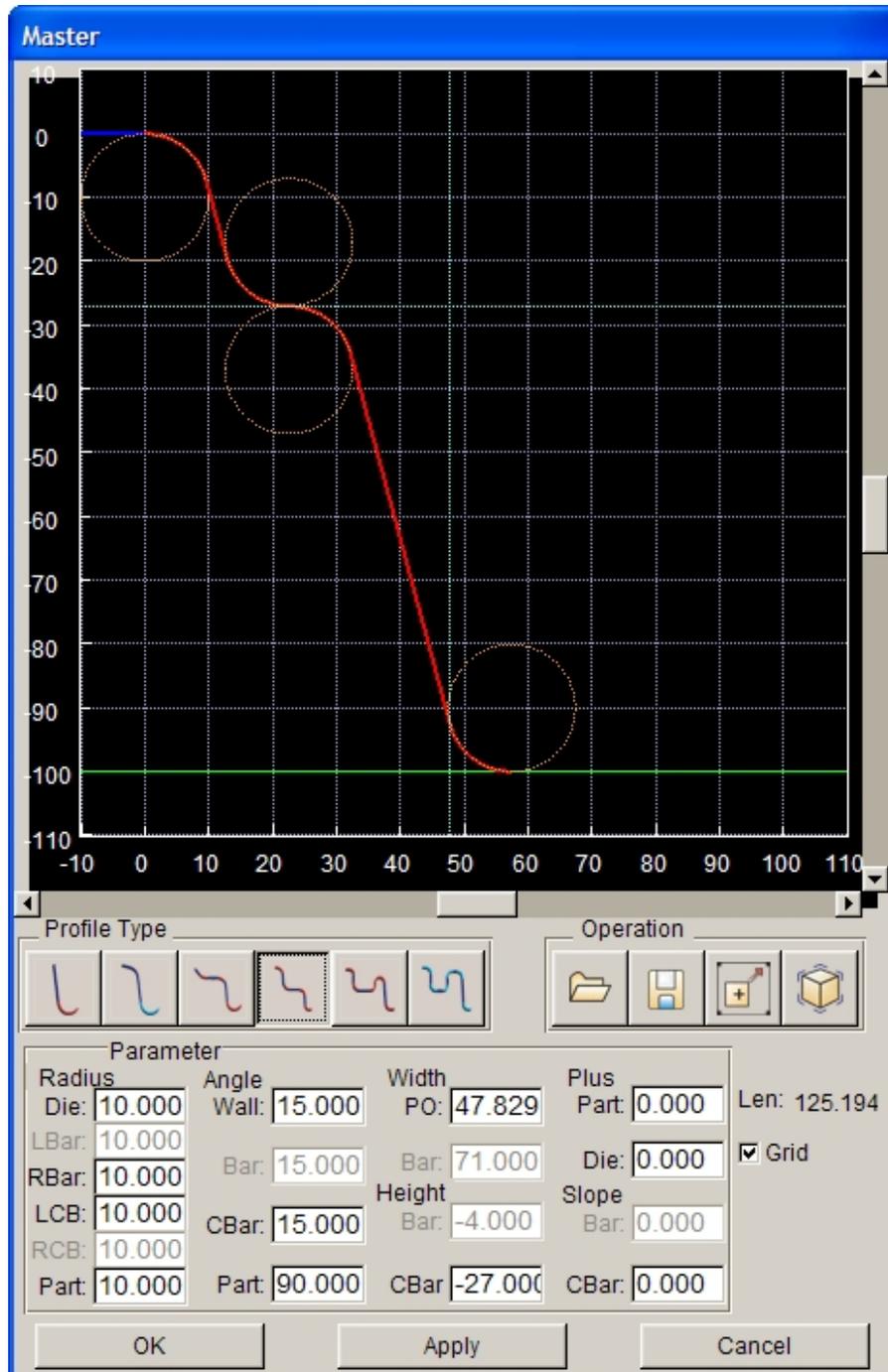


Figure 6.3.7 Profile Type 4 window

**TYPE 5**

The TYPE 5 profile is the combination of TYPE 4 profile with a simple drawbar. As illustrated in Figure 6.3.8, both BAR WIDTH and CB HEIGHT are disabled.

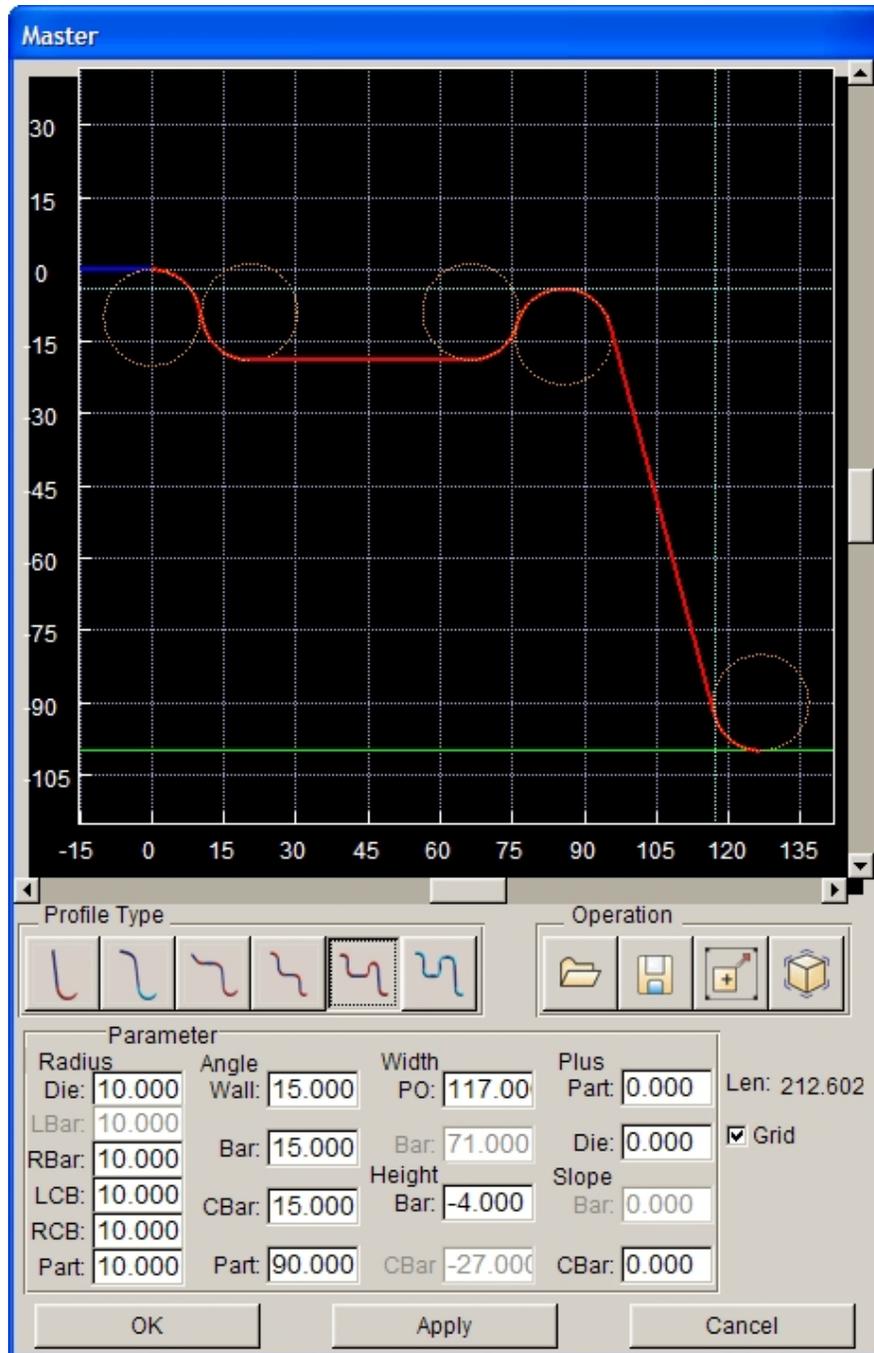


Figure 6.3.8 Profile Type 5 window

**TYPE 6**

The TYPE 6 profile is a combination of TYPE 5 profile with a full-featured drawbar. As illustrated in Figure 6.3.9, all parameters are enabled in this type of profile.

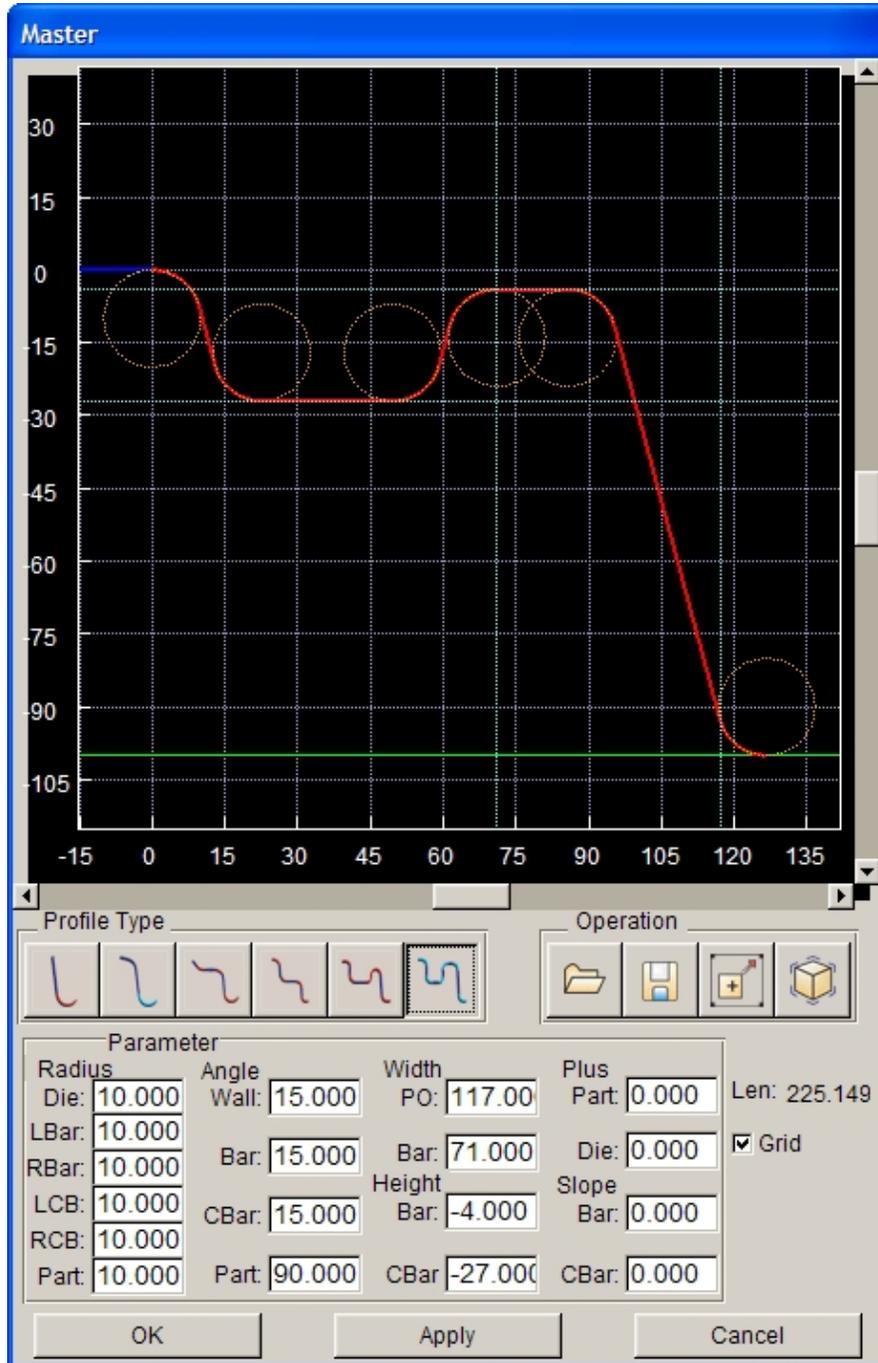
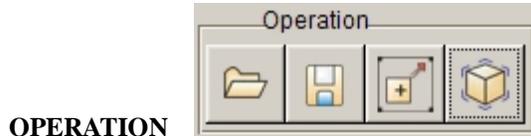


Figure 6.3.9 Profile Type 6 window

**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 6.3.10.

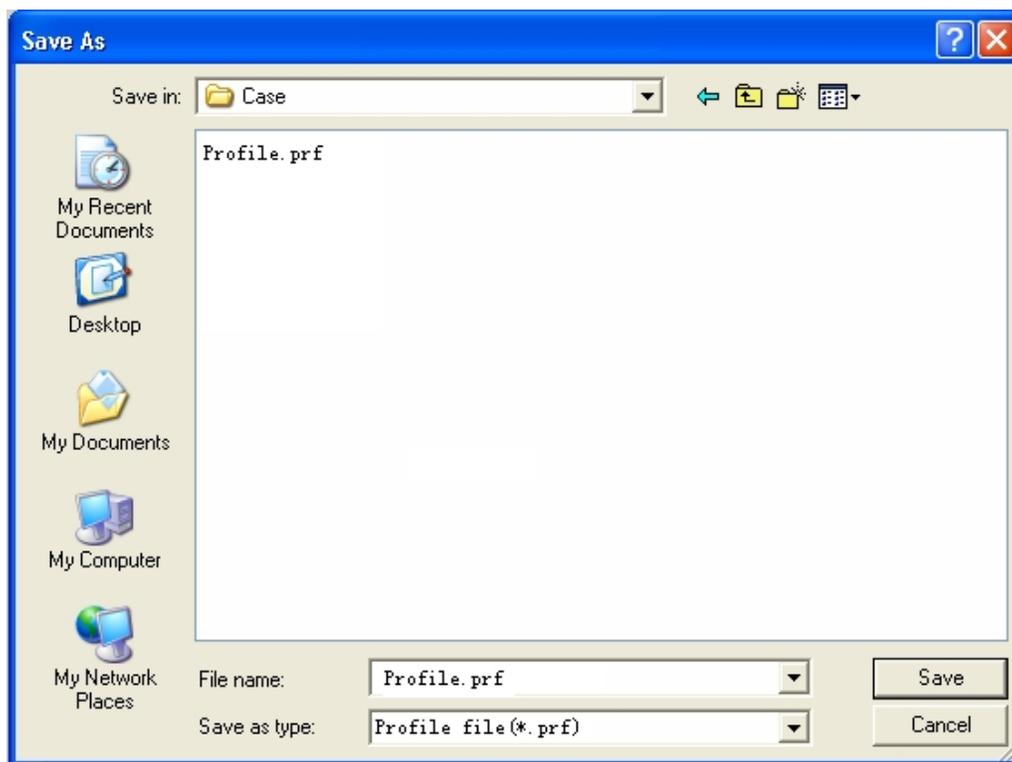


Figure 6.3.10 save Profile Parameters in the File

**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 6.3.11.



Figure 6.3.11 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.*

## 6.3.2 ADDENDUM

This menu provides functions to create, edit and modify the addendum, etc. The functions provided in the ADDENDUM menu are illustrated in Figure 6.3.12.

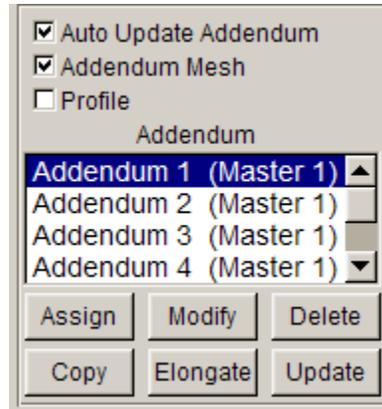


Figure 6.3.12 Create and Edit Addendum

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.

### **6.3.2.1 ASSIGN**

eta/DYNAFORM provides two methods to create the addendum. If checkbox of the BY SEGMENTS option is toggled on, the user can create the addendum by multiple segments using different master profiles. If the checkbox is 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 6.3.13.

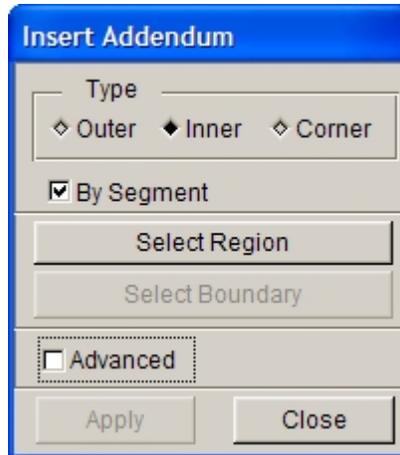


Figure 6.3.13 Insert Addendum dialog box

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 6.3.14 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 6.3.15 illustrates a typical addendum around the entire out boundary of a part.

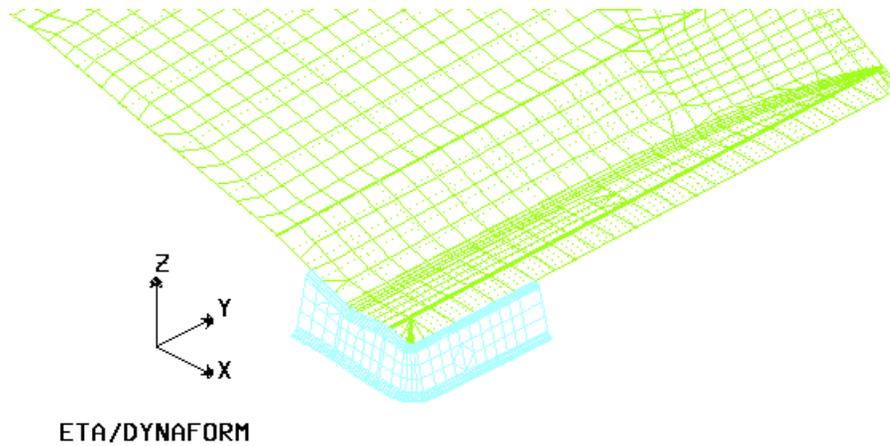


Figure 6.3.14 Create Addendum by Segment

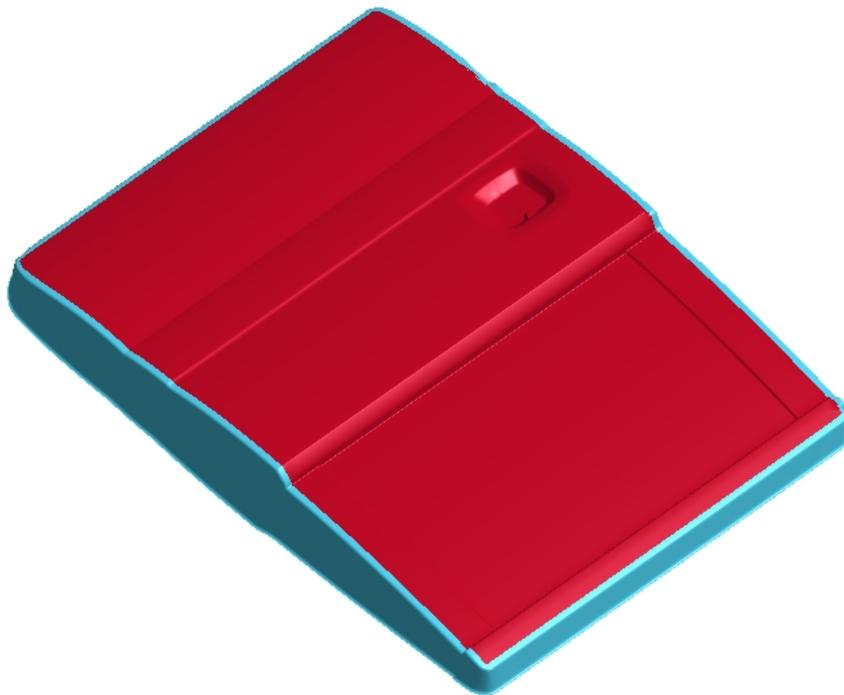


Figure 6.3.15 Complete Addendum by Master Profile

**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. Select CORNER and ADVANCED, as shown in Figure 6.3.16. ADVANCED is used to create the PO line for corner addendum. The user can use the Create Spline Line function in ADVANCED options to create a PO line, as shown in Figure 6.3.19. About detailed descriptions for all the functions in ADVANCED, please refer to Section 5.2.11, Split Surface.
2. Click the SELECT PROFILE button to select a profile on the addendum boundary. The selected profile will be highlighted as illustrated in Figure 6.3.20.
3. Select two nodes on the part boundary as illustrated in Figure 6.3.21. Click on the YES button to accept the defined corner region.
4. As illustrated in Figure 6.3.16, APPLY button is enabled. Clicking on APPLY, the program will pop up a question dialog box for the user to determine whether boundary line should be selected as the PO line for corner addendum, as shown in Figure 6.3.17. Click YES to select the previously created PO line. The program will auto generate the corner addendum, as shown in Figure 6.3.22.

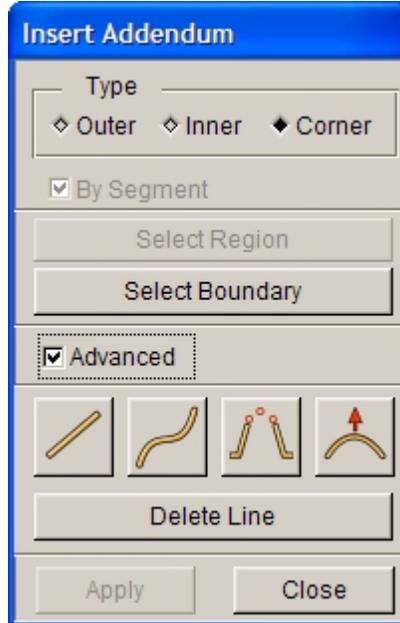


Figure 6.3.16 Corner Addendum

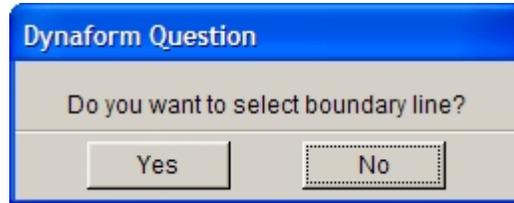


Figure 6.3.17 Dynaform Question dialog box

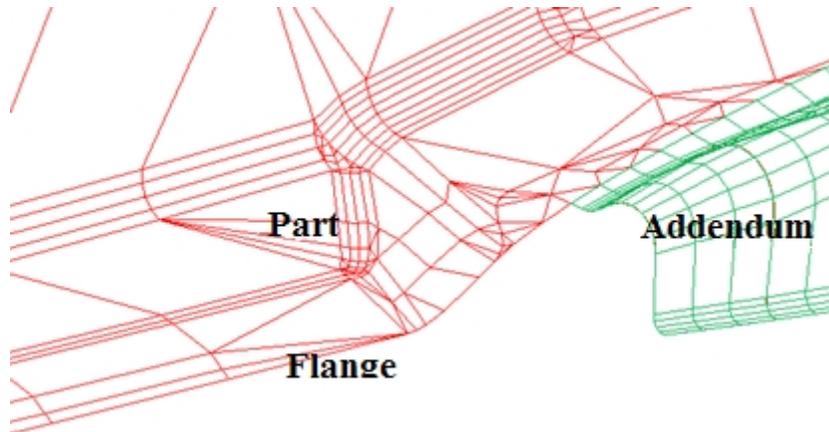


Figure 6.3.18 Flange on Binder Surface

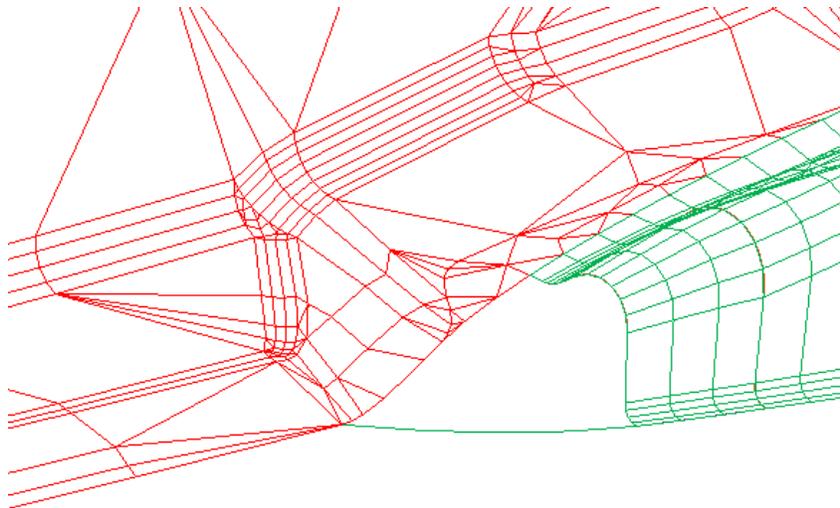


Figure 6.3.19 Created PO Line

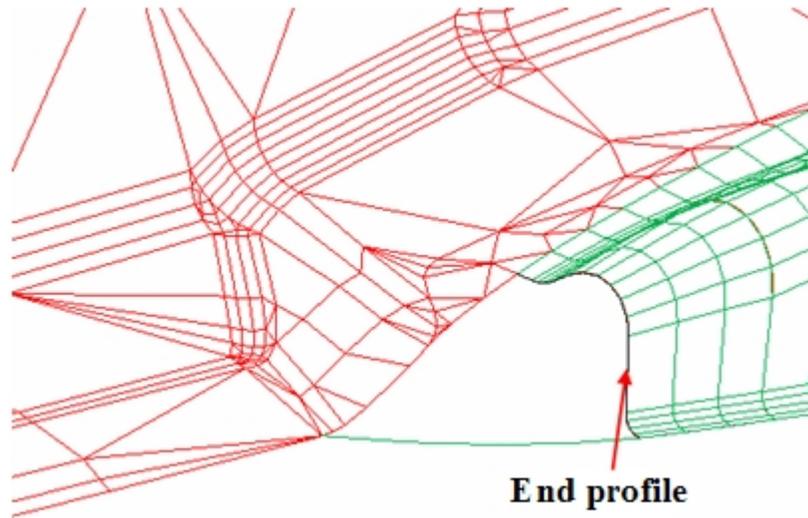


Figure 6.3.20 Select End Profile on Addendum

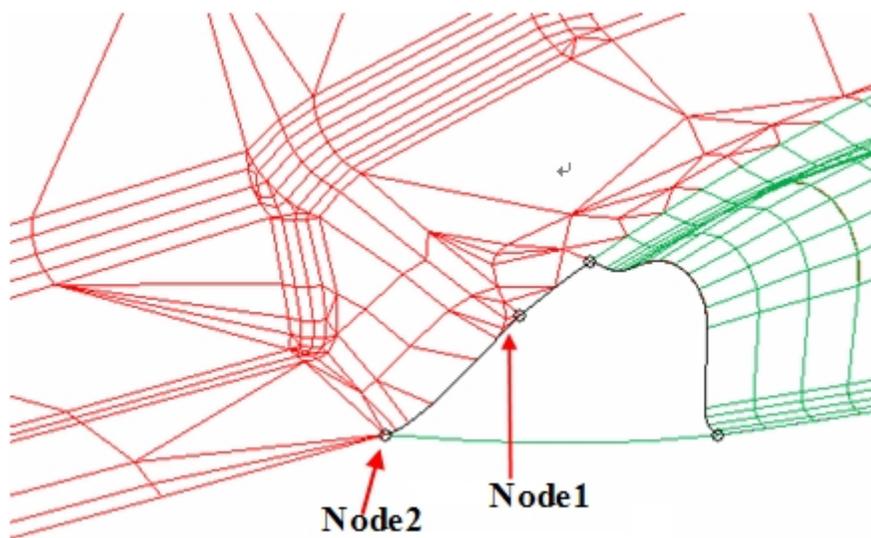


Figure 6.3.21 Select Two Nodes on Part Boundary

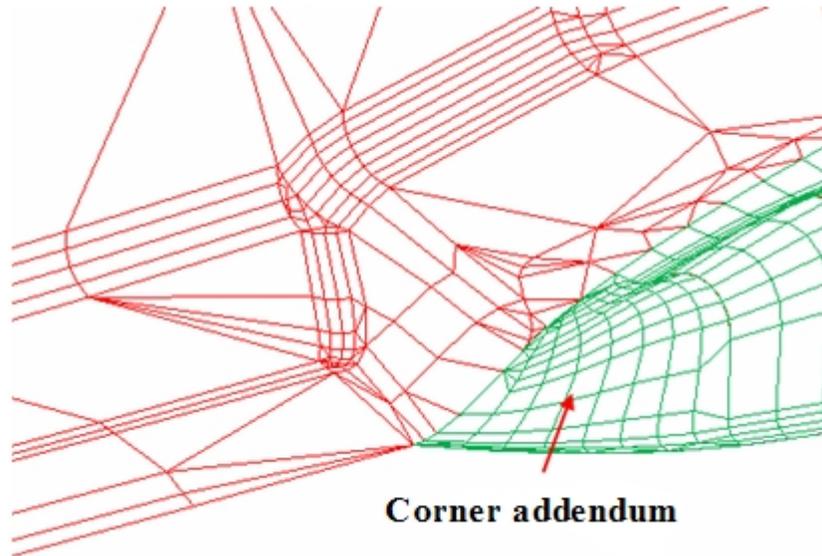


Figure 6.3.22 Generated Corner Addendum

### 6.3.2.2 MODIFY

Once MODIFY button is selected, the program will pop up the MODIFY ADDENDUM dialog box, as shown in Figure 6.3.23.

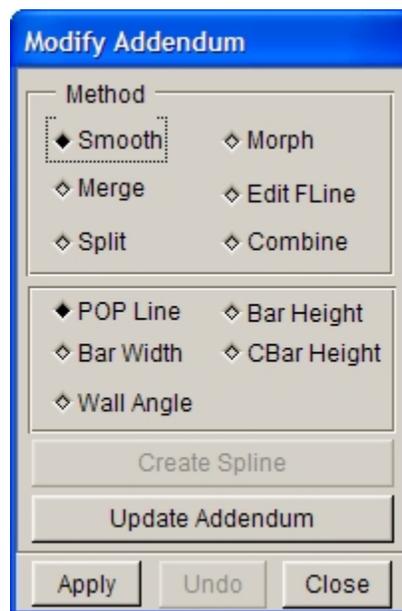


Figure 6.3.23 Modify Addendum

The user can modify addendum by the following 6 methods.

#### 1. SMOOTH

This option allows the user to modify addendum by selecting POP Line, Bar Height, Bar Height, Bar Width, CBar Height or Wall Angle.

Once the SMOOTH option is selected, all the section lines and POP lines on addendum are displayed in graphic window. As illustrated in Figure 6.3.24, the purple line is the section line and the black line is the POP line. Then select a smooth type and select a PROFILE. The entire addendum where located the selected profile will be the object for this smooth operation.

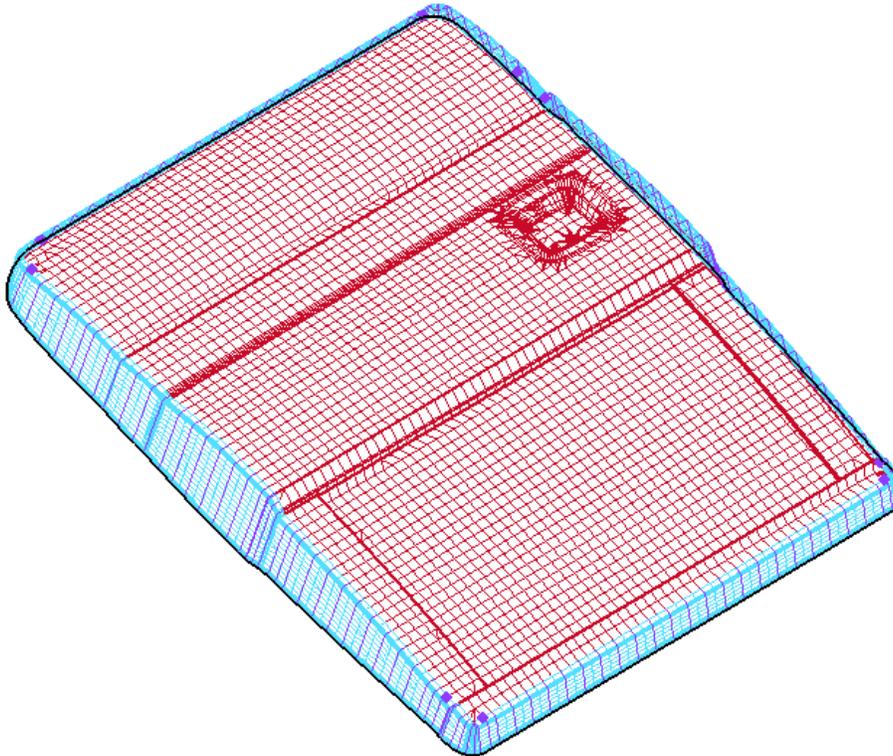


Figure 6.3.24 Profile and POP Line

#### POP Line

This function allows the user to change the addendum by smoothing the shape of part POP line. after a profile is selected, the program will highlight the POP line in the selected region as shown in Figure 6.3.24 and pop up a dialog box as shown in Figure 6.3.25.

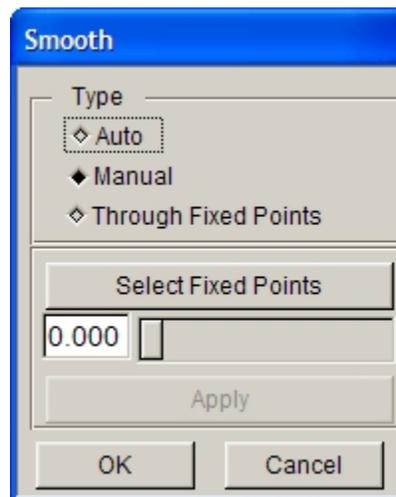


Figure 6.3.25 Smooth POP line

There are three methods can be used to smooth the POP line.

#### **AUTO SMOOTH**

Automatically smooth the POP LINE in the selected region.

#### **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.

#### **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.

Figure 6.3.27 illustrates the smoothed POP line and the relevant profiles. After the POP line is smoothed, click CLOSE to exit the dialog box. The program will update the profiles and addendum according to the smoothed POP line. The updated addendum is illustrated in Figure 6.3.28.

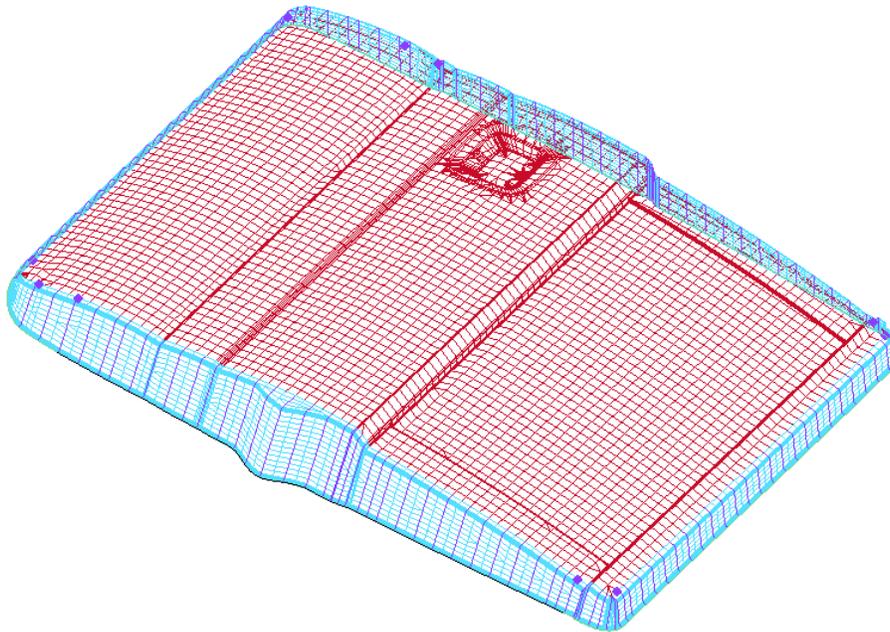


Figure 6.3.26 Selected POP Line for Smoothing

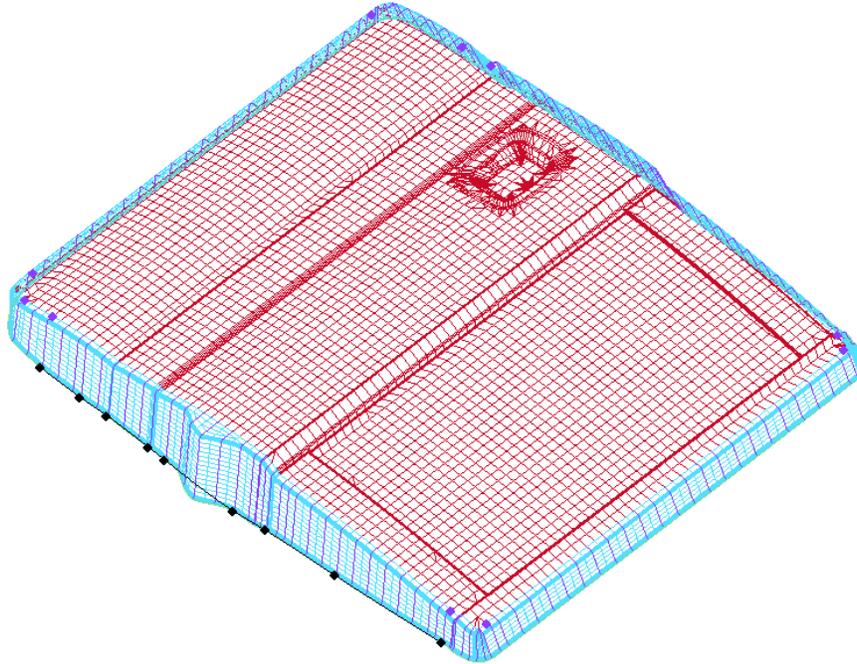


Figure 6.3.27 Smoothed POP Line and Profiles Through Fixed Points

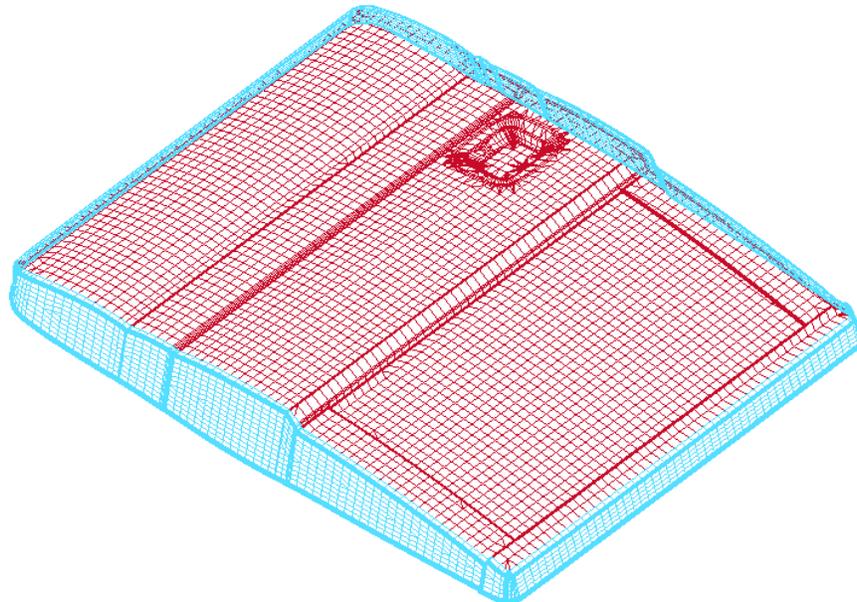


Figure 6.3.28 The Updated addendum

### **BAR HEIGHT**

This function allows the user to smooth drawbar height of the profiles in the selected region. The program prompts the user to select a segment of profile to define the region. The operation is similar to the POP LINE smoothing.

**BAR WIDTH**

This function allows the user to smooth drawbar width of the profiles in the selected region. The program prompts the user to select a segment of profile to define the region. The operation is similar to the BAR HEIGHT.

**CBAR HEIGHT**

This function allows the user to smooth CBar height of the profiles in the selected region. The program prompts the user to select a segment of profile to define the region. The operation is similar to the POP LINE smoothing.

**WALL ANGLE**

This function allows the user to smooth wall angle of the profiles in the selected region. The program prompts the user to select a segment of profile to define the region. The operation is similar to the POP LINE smoothing.

**2. MORPH**

This function enables the user to modify addendum by morphing the POP LINE, BAR HEIGHT, BAR WIDTH, CBAR HEIGHT and WALL ANGLE. Once this option is selected, the user needs to select a morph type and click on the APPLY button. The program prompts the user to select the starting and ending profile to define the region to be modified.

**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 that is subjected to morphing. If more than one region is defined in the addendum, the starting and ending profiles in each region are 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 6.3.29. The morphing operation of POP LINE is similar to the LINE MORPHING function. Figure 6.3.30 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 6.3.31 illustrated the updated addendum after the POP LINE is morphed.

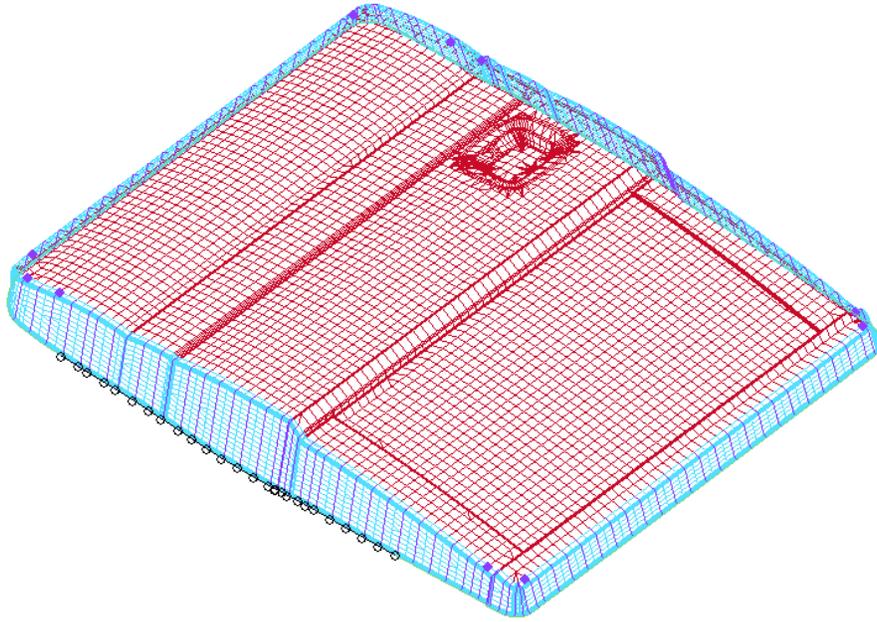


Figure 6.3.29 Selected POP Line for Morphing

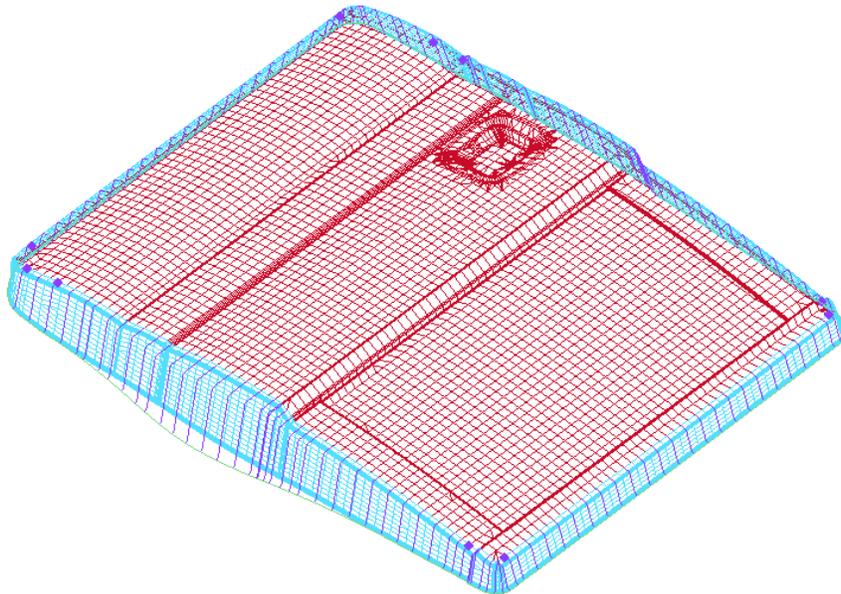


Figure 6.3.30 Morphed POP Line and Profiles

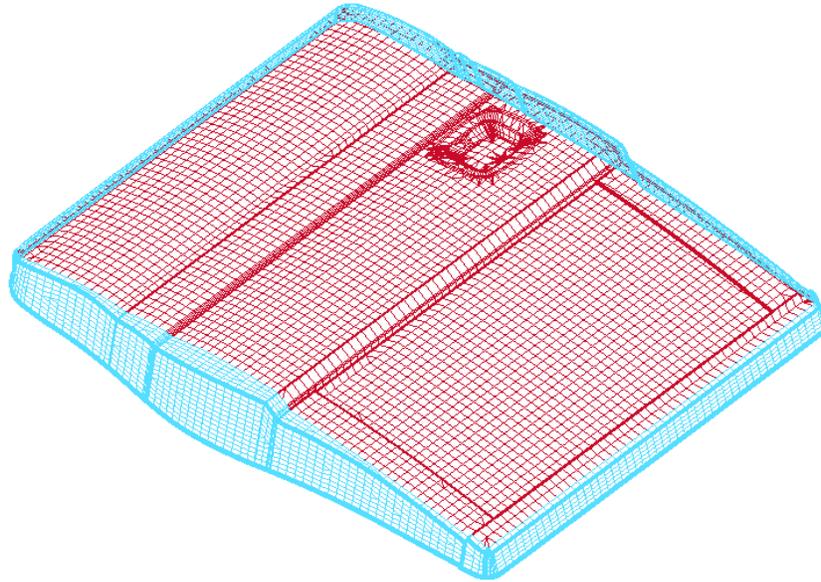


Figure 6.3.31 Updated Addendum According to the Morphed POP Line

#### **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.

### 3. MERGE

This function enables the user to merge 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. Once this option is selected, the Create Spline button will be activated, as shown in Figure 6.3.32. Clicking on this button, the program will pop up the Create Spline dialog box as shown in Figure 6.3.33. Refer to Section 6.1.1.11 INNER FILL for detailed description of this dialog box.



Figure 6.3.32 Merge Addendum

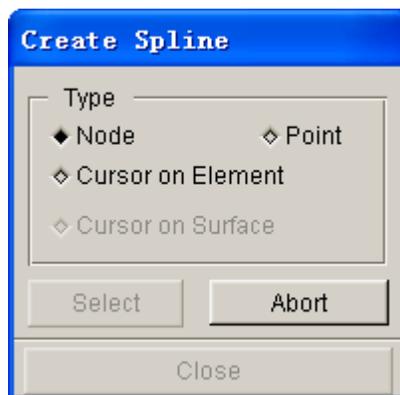


Figure 6.3.33 Create Spline

### POP LINE

Modify profiles according to the merged POP LINE.

### BAR HEIGHT

Modify profiles according to the merged BAR HEIGHT control line.

**BAR WIDTH**

Modify profiles according to the merged BAR WIDTH control line.

**CBAR HEIGHT**

Modify profiles according to the merged CBAR HEIGHT control line.

**WALL ANGLE**

Modify profiles according to the merged WALL ANGLE control line.

Figure 6.3.34a illustrates the definition of a straight line for MERGE POP LINE operation. Figure 6.3.34b 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.

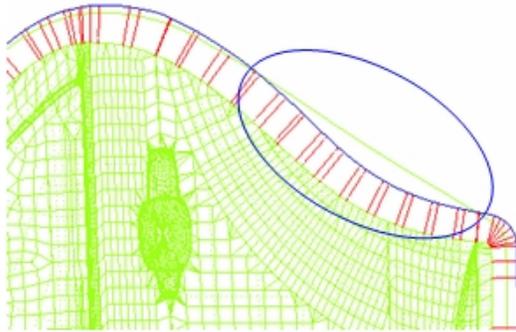


Figure 6.3.34a Merge POP Line

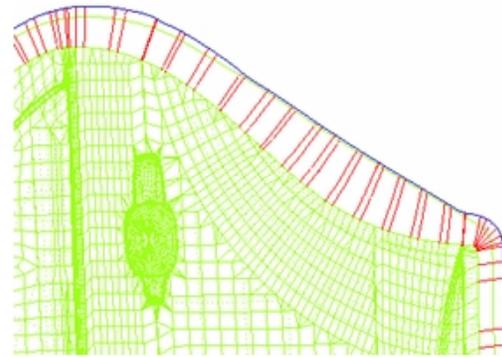


Figure 6.3.34b Merge POP Line Result

**4. EDIT FLINE**

This function enables the user to modify addendum by editing the entire POP LINE, BAR HEIGHT, BAR WIDTH, CBAR HEIGHT and WALL ANGLE. Once this option is selected, the user needs to select an edit type and click on the APPLY button. The program pops up a dialog box as illustrated in Figure 6.3.35. In this dialog box, click ADD POINT button to add control points on the boundary line and then click on the middle mouse button to confirm; click REMOVE POINT button to remove the control points on the boundary line; click MOVE POINT button to move the control points on the boundary line and then click on the middle mouse button to confirm. After editing the boundary line, click EXIT button to update the addendum automatically.



Figure 6.3.35 Edit FLine

### 5. 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. Once this option is selected, the options POP LINE, BAR HEIGHT, BAR WIDTH, CBAR HEIGHT and WALL ANGLE will be disabled. The user should select a profile in the region to be split. If the addendum region where the profile locates is closed, the program will prompt the user to select another profile to split this closed region. If the selected region is open, the program will automatically split this region along the 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. COMBINE

This option allows the user to combine two adjacent independent regions into one region. Once this option is selected, the options POP LINE, BAR HEIGHT, BAR WIDTH, CBAR HEIGHT and WALL ANGLE will be disabled, the program will prompt the user to select two adjacent independent addendum regions. After the selection, the program will automatically combine the two regions into one region.

### 6.3.2.3 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 6.3.12. eta/DYNAFORM highlights the selected addendum and displays a dialog box.

Click on the YES button to confirm removal of the selected addendum. If NO is selected, the operation is aborted.

### 6.3.2.4 COPY

Once the user clicks on the COPY button, the program will pop up a dialog box as shown in Figure 6.3.36. With this dialog box, the user can COPY, MIRROR, MOVE, or PATCH the addendum.



Figure 6.3.36 Copy Addendum

- COPY

This function enables the user to copy a region of addendum to another position. If there is more than one addendum region in the database, the program prompts the user to select a profile region:

***PICK A PROFILE REGION***

The user can select a profile in the addendum region to be copied. The entire addendum corresponding to this PROFILE will be highlighted, as shown in Figure 6.3.37(a). The program will prompt the user to select a target node on part boundary as starting point for the addendum to be copied:

***PICK NODE ON DIE BOUNDARY AS DESTINATION POSITION***

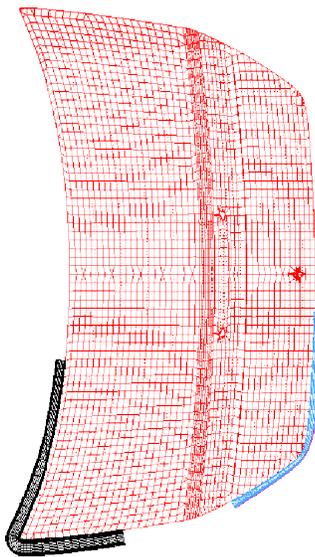


Figure 6.3.37(a) Copy Addendum

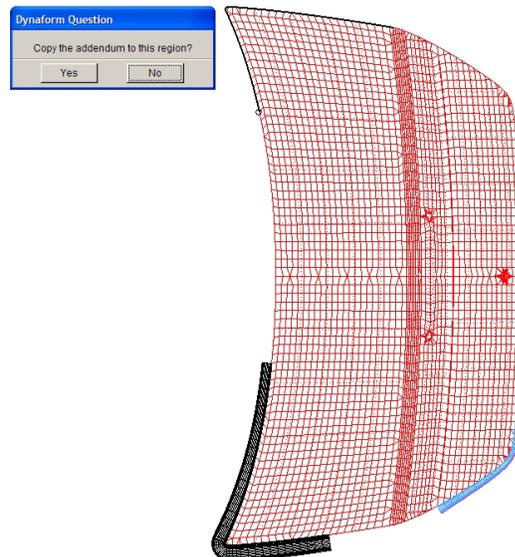


Figure 6.3.37(b) Copy Addendum

After a destination position is selected on the part boundary, the program pops up a Dynaform Question dialog box for the user to confirm the copied position and highlight the new part boundary, as shown in Figure 6.3.37(b). If the position of new boundary is not correct, select NO to put the new boundary on the other side of the reference point. Click YES to accept the boundary location. The selected addendum will be copied as shown in Figure 6.3.37(c).

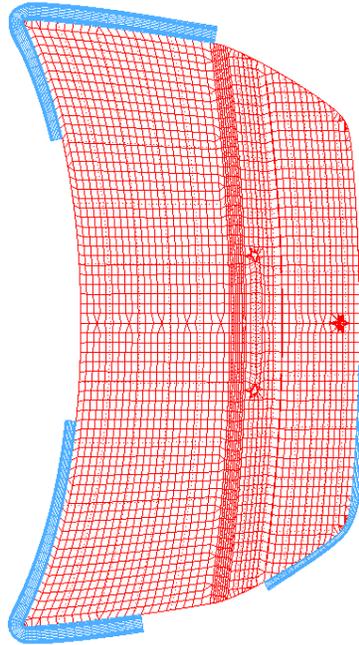


Figure 6.3.37(c) Copy Addendum

- **MIRROR**

This function generates the entire addendum by mirroring it about a 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 6.1.2 for description 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 6.3.38a, the user selects YES to mirror the addendum. The mirrored result is illustrated in Figure 6.3.38b.

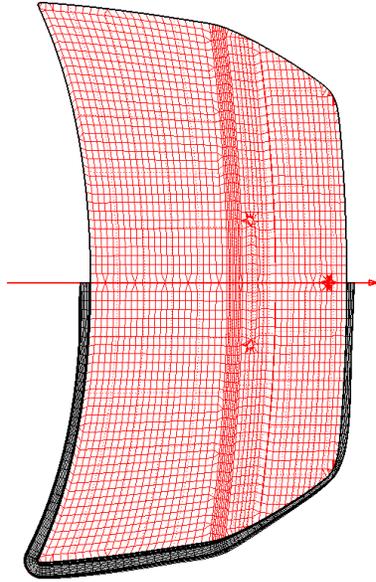


Figure 6.3.38a Mirror Addendum

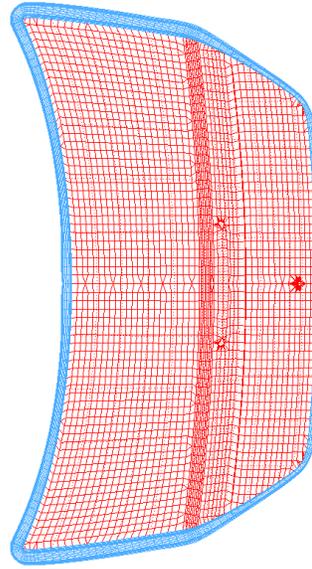


Figure 6.3.38b Mirror Addendum

**NOTE:** *If the generated addendum by the MOVE, COPY or MIRROR function interferes with existing addendum, the operation will be aborted.*

- **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 given 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 as illustrated in Figure 6.3.33a. The following message is then given in the message prompt window.

**PICK NODE ON DIE BOUNDARY AS DESTINATION POSITION**

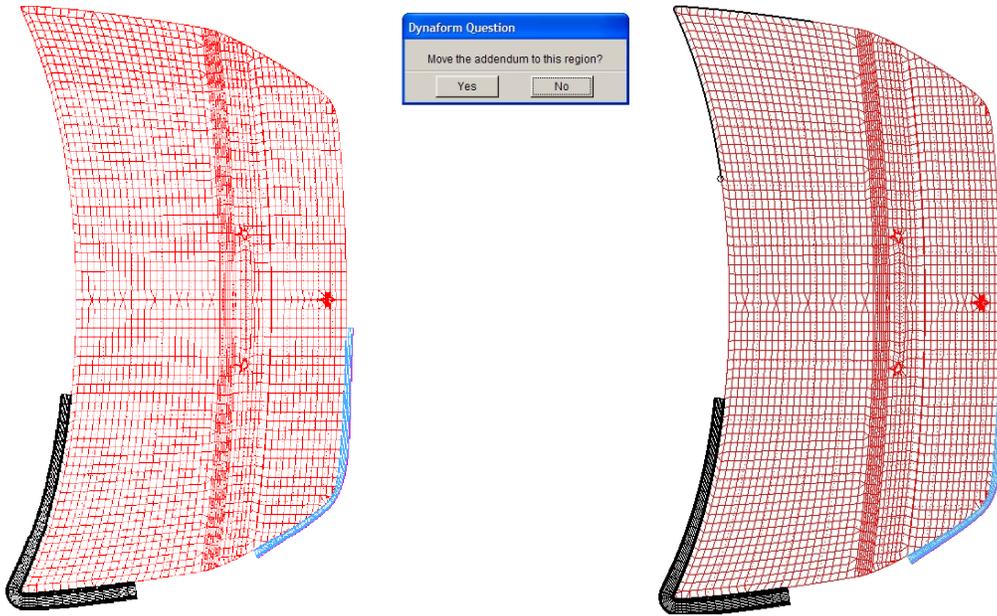


Figure 6.3.39a Move Addendum

Figure 6.3.39b 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 6.3.39b. 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. Click on the YES button to accept the boundary location. The selected addendum will be moved to the location as illustrated in Figure 6.3.39c.

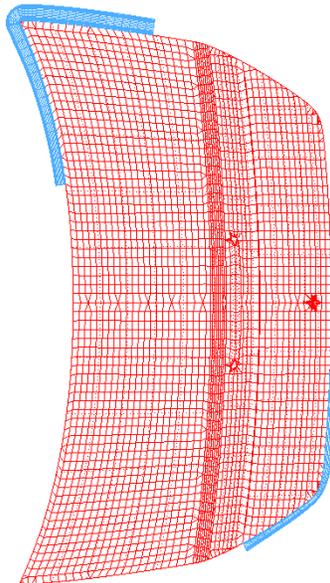


Figure 6.3.39c Move Addendum

- 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 6.3.40 illustrates a typical addendum region of different profile types.

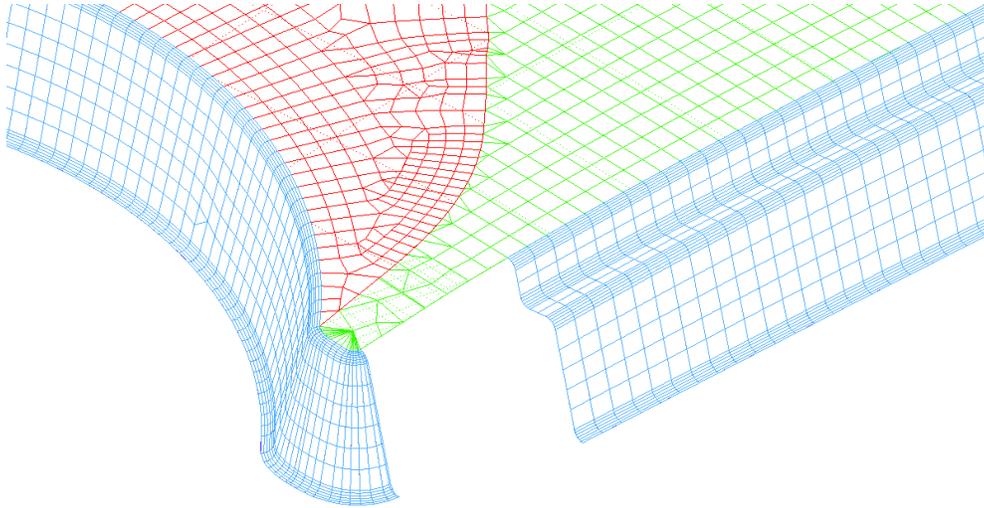


Figure 6.3.40 Transition Area between Adjacent Regions

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 6.3.41 illustrates a typical transition area generated by the patch function.

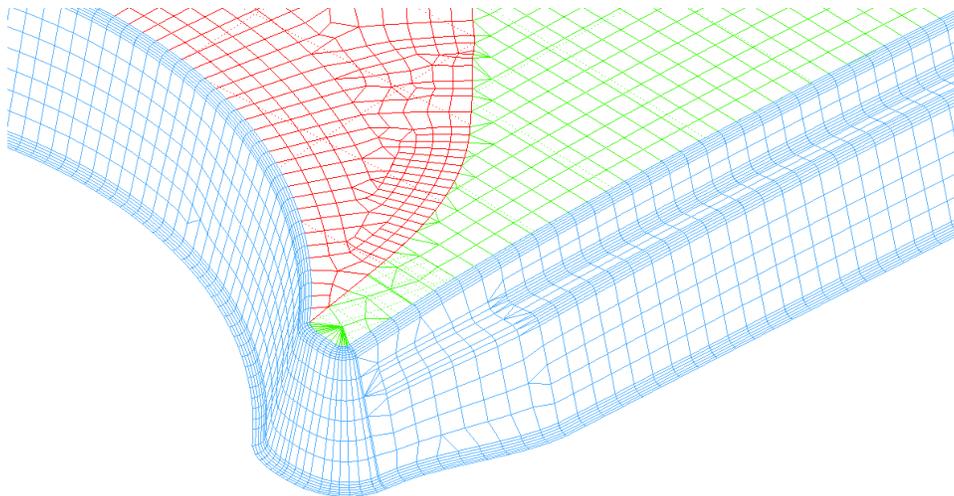


Figure 6.3.41 Typical Result of Patch Function

### 6.3.2.5 ELONGATION

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 6.3.42. These section lines are created from the plane of the profile intersecting with the binder and the die.

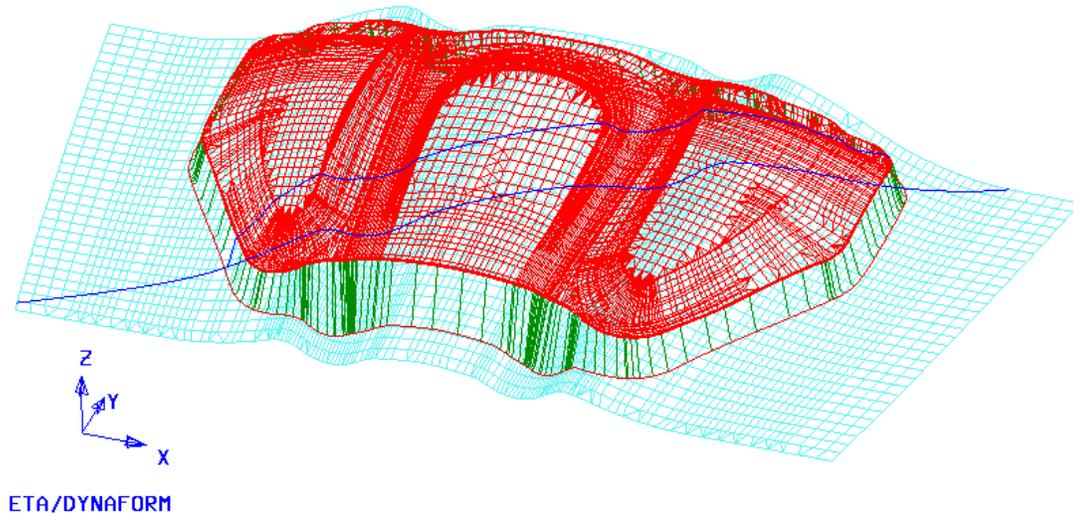


Figure 6.3.42 Select profile

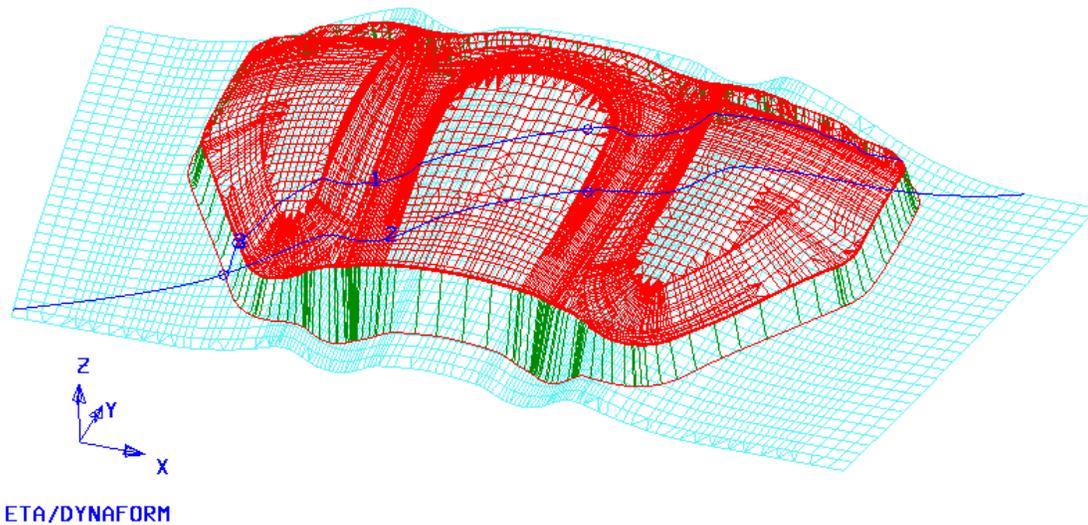


Figure 6.3.43 Point Selection on Line

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 6.3.43, 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:

$$ELONGATION = (LENGTH\ 1 + LENGTH\ 3 - LENGTH\ 2) / LENGTH2 * 100\%$$

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.

### 6.3.2.6 UPDATE

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 necessary.

## 6.3.3 PROFILE

The profile functions enable the user to INSERT, DELETE, MODIFY or ORIENT one or more profiles. Figure 6.3.44 illustrates the functions provided in the PROFILE section.

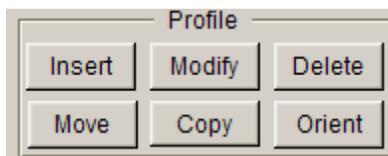


Figure 6.3.44 Profile Options

### 6.3.3.1 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.

### 6.3.3.2 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 as shown in Figure 6.3.45. The user can modify profile types and/or profile parameters. During modification of profiles, the grid will be displayed in profile window if the user selects GRID. If UPDATE is selected, the profile is highlighted. During modification of parameters, the change of profile can be viewed in real time in the graphic window. If CTRLPRF is selected, the shape transition of profile between adjacent or similar control profiles can be controlled. As profiles are modified, eta/DYNAFORM will update the profile in the graphic display window.

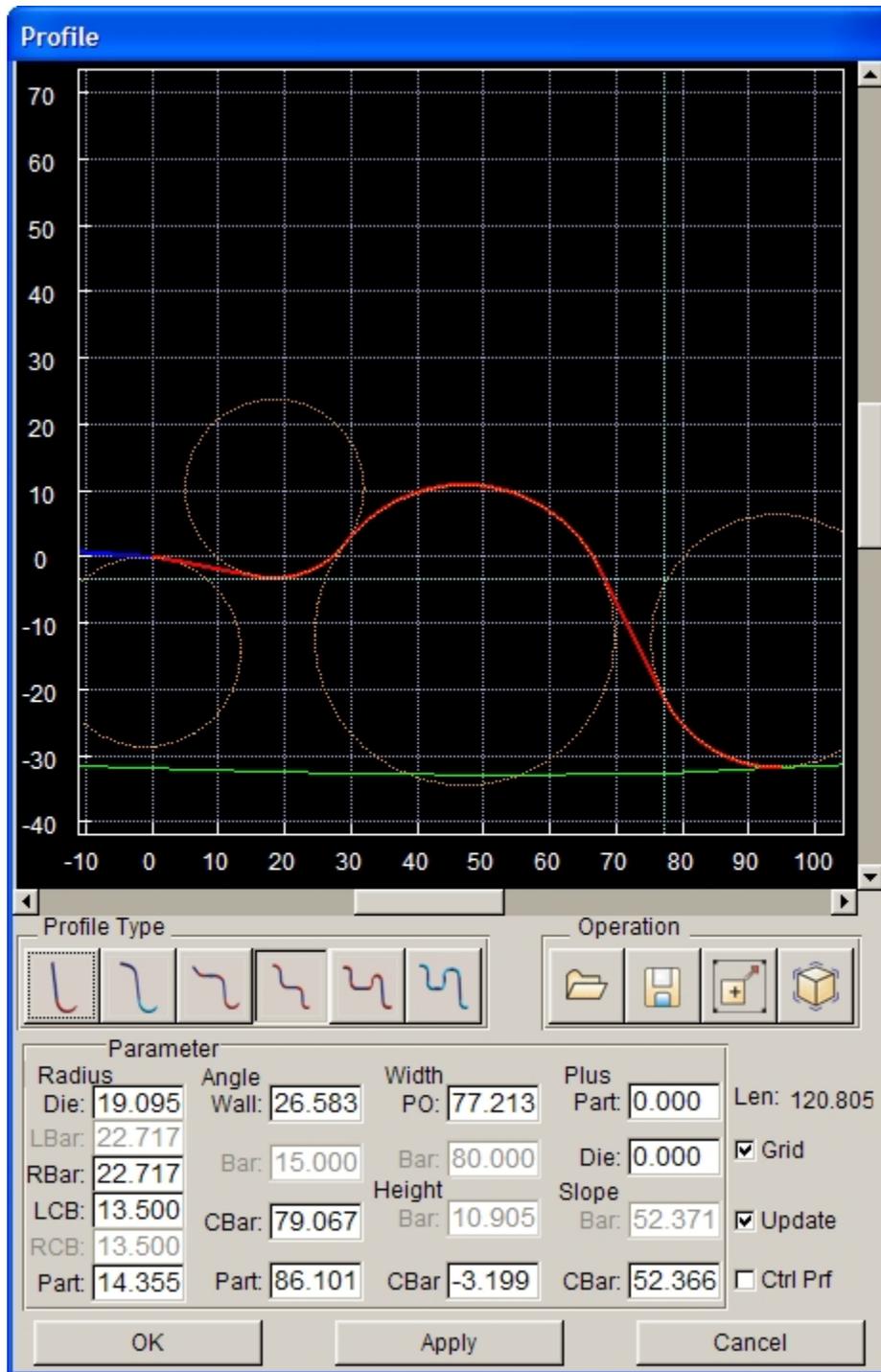


Figure 6.3.45 Modify Profiles

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.

### 6.3.3.3 DELETE

This function enables the user to remove one or more profiles. The dialog box is illustrated in Figure 6.3.46.

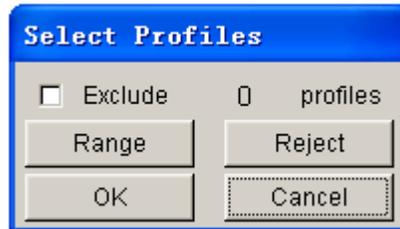


Figure 6.3.46 Select Profiles dialog box

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.

### 6.3.3.4 MOVE

This function enables the user to move a selected profile to another location. Clicking on this button, the program will hide the elements of addendum automatically and prompt the user to select a profile to be moved by using the left mouse button. The user can select any profile except the boundary profile. The selected profile will be highlighted on the screen.

The program will prompt the user to select a node on the die boundary as the new location of the profile

and then select another node on the POP LINE as the location of the other end of profile. After these two nodes are selected, the program will move the profile to the new position automatically.

The user can click the middle mouse button to confirm and complete the move operation. If other similar operations are required, the above operation can be repeated.

**NOTE:** *The two end profiles defining an addendum region cannot be moved.*

### 6.3.3.5 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.

### 6.3.3.6 ORIENTATION

Once ORIENTATION is selected, a dialog box as illustrated in Figure 6.3.47 appears. With this dialog box, the user can change the orientation of a profile relatively to the POP LINE by using four methods.

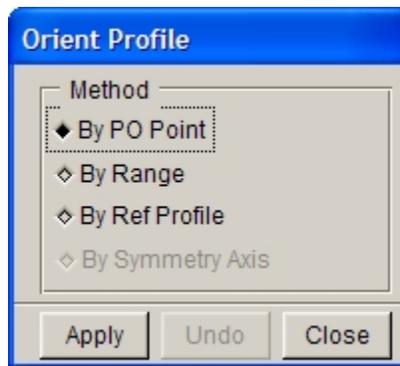


Figure 6.3.47 Orient Profile dialog box

- **BY PO POINT**

This function allows the user to select a profile to change its orientation relatively to the POP LINE. If the function is selected, the following message is printed in the message prompt window:

***SELECT PROFILE TO CHANGE 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 by clicking the middle mouse button, the user clicks on CLOSE button to exit the function if AUTO UPDATE ADDENDUM is selected. The addendum mesh is automatically updated.

- **BY RANGE**

This function allows the user to change the orientation of a selected profile relatively to the POP LINE by selecting a profile segment. The user can select this option and click the APPLY button to highlight some profiles in order to display the profile region. The user can also customize the profile region. Double click

on the left mouse button to cancel the specified highlighted profiles and click on the left mouse button to select and highlight the specified profile. Then, select a profile in the selected region and drag the profile left and right. All the profiles in this region will move accordingly. Therefore, the orientation of this profile segment relatively to the POP line will be changed. After completing the profile orientation operation by clicking the middle mouse button, the user clicks on CLOSE button to exit the function if AUTO UPDATE ADDENDUM is selected. The addendum mesh is automatically updated.

- BY REF PROFILE

This function allows the user to select a reference profile and then select other profiles. The direction of the reference profile will be an orientation standard for the selected profile. After completing the profile orientation operation by clicking the middle mouse button, the user clicks on CLOSE button to exit the function if AUTO UPDATE ADDENDUM is selected. The addendum mesh is automatically updated.

- BY SYMMETRY AXIS

This function allows the user to adjust the orientation of profile so that it can parallel to the orientation of the symmetry plane. For symmetric parts, after the completion of half addendum, the user usually mirrors the generated addendum onto the other half. Therefore, before the mirror operation, the user needs to adjust the profile orientation into the symmetry plane.

Once this dialog box is selected, the program will prompt the user to select a profile to adjust its orientation. After a profile is selected, if it locates within the symmetry plane, this profile will be adjusted into the symmetry plane automatically. Figure 6.3.48 shows a typical result of symmetry operation.

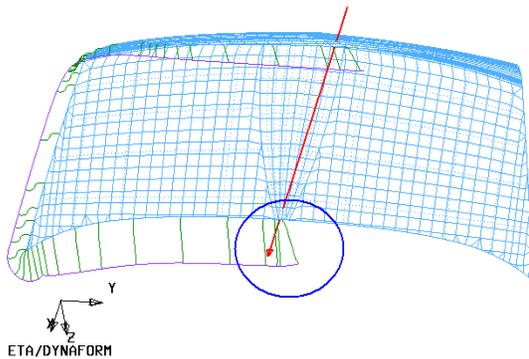


Figure 6.3.48(a) Before Profile Adjustment

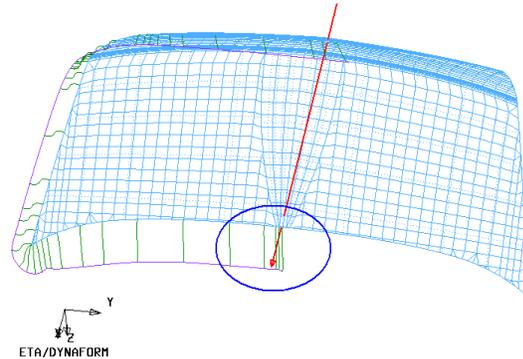


Figure 6.3.48(b) After Profile Adjustment

If the selected profile is not in the symmetry plane, a dialog box as illustrated in Figure 6.3.49 appears.

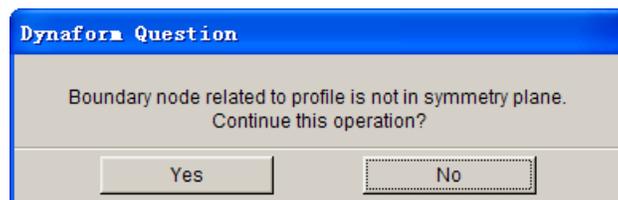


Figure 6.3.49 Dynaform Question dialog box

Click YES to continue operation. Click NO to cancel this operation.

After the completion of operation, the program prompts the user to select other profiles to continue adjustment, or click on the right mouse button to exit this function.

Note: this function is only available for the symmetric parts and cannot be applied on non-symmetrical parts.

### 6.3.4 EXPORT

After the user clicks on the EXPORT button, a dialog box as illustrated in Figure 6.3.50 appears. This function enables the user to create POP LINE, PROFILE LINE and ADDENDUM SURFACE. The user can import these created lines and surfaces to other CAD software for further operation

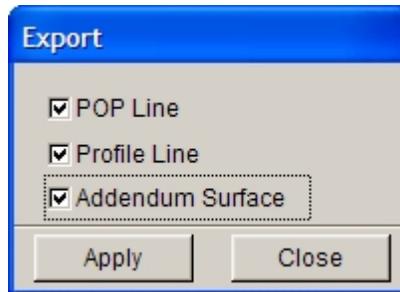


Figure 6.3.50 Export

- **POP 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.
- **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.
- **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. After completion of addendum design in eta/DYNAFORM, the user can click on this button to generate CAD surfaces in NUBUS format.

### 6.3.5 TRIM LINE

This function enables the user to determine the trim line of the flange on 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 6.3.51.

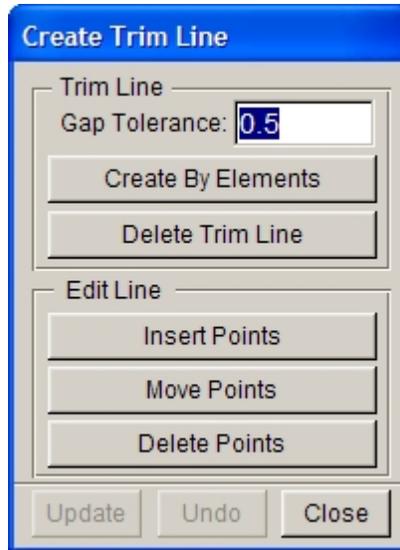


Figure 6.3.51 Create Trim Line dialog box

### **GAPTOLERANCE**

This parameter sets the tolerance between the selected flange and the addendum elements. The default is 0.5. 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 that are 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. Refer to Figure 6.3.52 and Figure 6.3.53 for illustration of CREATE TRIM LINE operation.

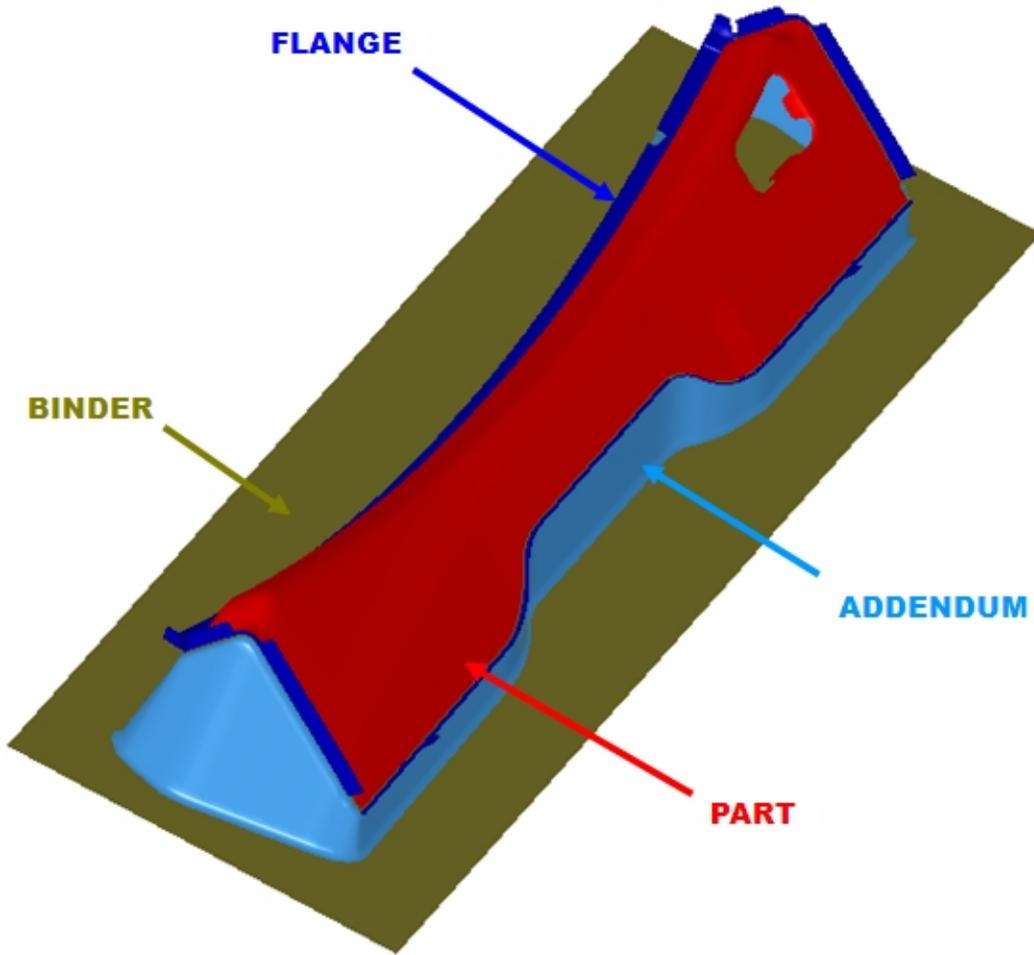


Figure 6.3.52 Before Create the Trim Line operation.

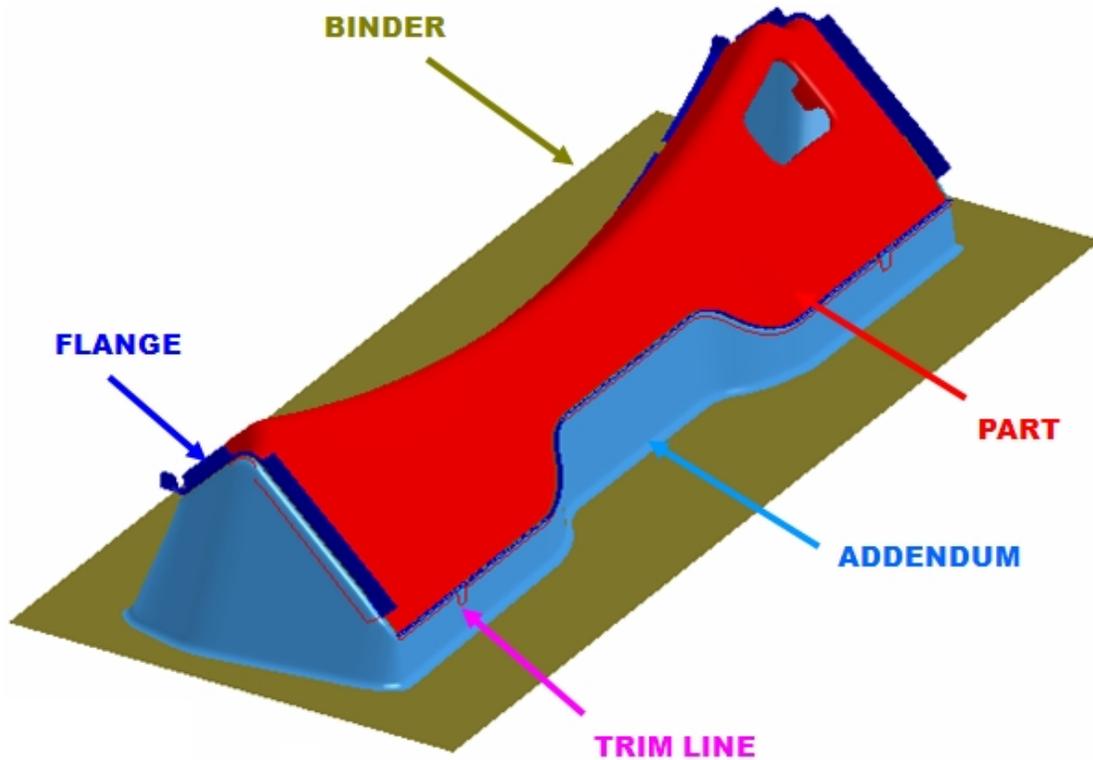


Figure 6.3.53 After Created the Trim Line operation.

### 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.

## 6.4 RE-ENGINEERING

In RE-ENGINEERING module, there are three main functions: CONVERT, NEW ADDEN and EVALUATE.

The dialog box of RE-Engineering is illustrated in Figure 6.4.1.

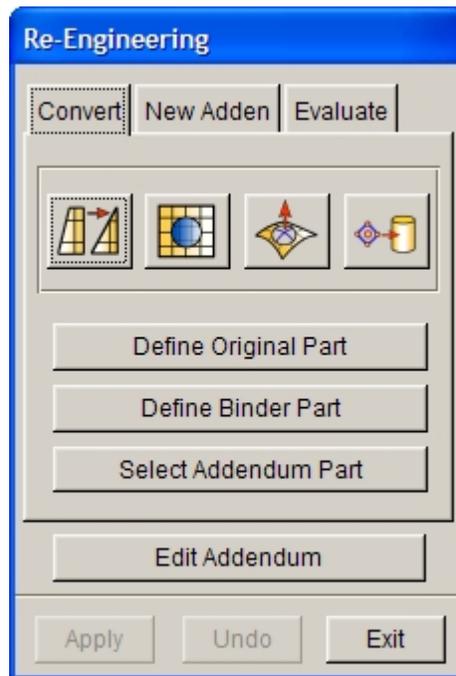


Figure 6.4.1 Re-Engineering dialog box

### 6.4.1 CONVERT DF ADDENDUM

This function allows the user to convert the addendum which is generated by other CAD software to the addendum that eta/DYNAFORM recognizes. The user can directly modify the addendum by using the tools in eta/DYNAFORM environment. The included auxiliary functions for convenient conversion are:

MODEL CHECK / REPAIR, INNER FILL, MORPH BY LINE and ADD... TO PART. The dialog box of Convert DF Addendum is illustrated in Figure 6.4.1.

- **MODEL CHECK / REPAIR**

This function allows the user to check and repair the addendum mesh before or after conversion. Refer to Section 5.5 for detailed description about the check/repair functions.

- **INNER FILL**

This function is used to fill the inner hole of the trimmed binder, as shown in Figure 6.4.2. Refer to Section 6.1.1.11 for detailed description about this function.

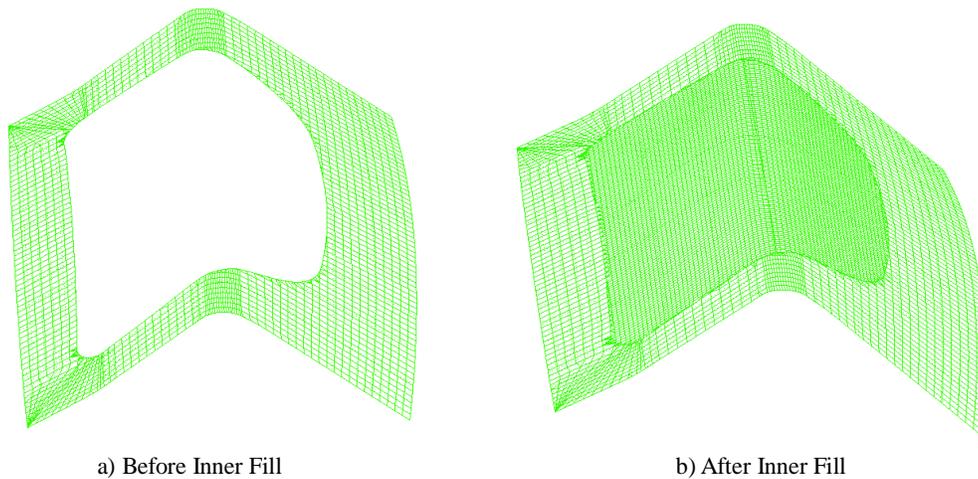


Figure 6.4.2 Inner Fill

- **MORPHING HOLE FILL**

This function is used to edit the filled inner hole of binder by adding control lines. After clicking on this button, the program displays a dialog box as illustrated in Figure 6.4.3. The user can select TANGENT to make sure that the generated surface boundary after editing is tangent to the binder. Click ADD to add control lines, EDIT to edit control lines, REMOVE to remove control lines, PREVIEW to preview the generated inner hole surface of binder after editing. The operation of this dialog box is similar to the one in Section 6.2.2 Modify Binder.

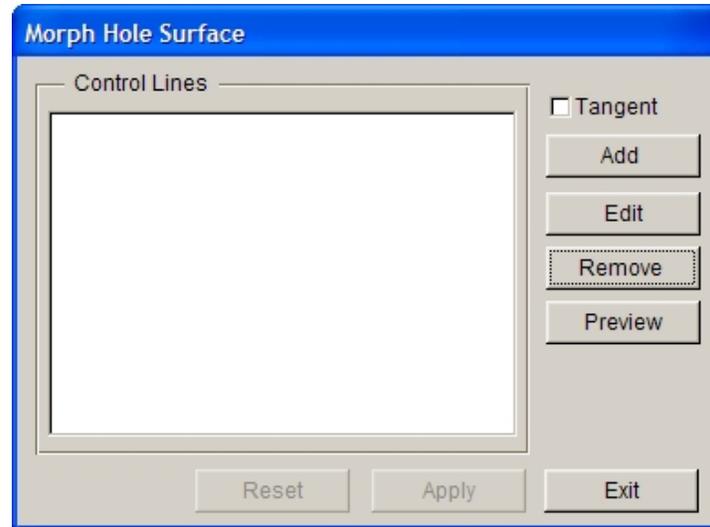


Figure 6.4.3 Morph Hole Surface

- **ADD... TO PART.**

This function allows the user to move line, element or surface from one part to another. The user can enter part name manually to create a new part. Refer to Section 4.4 for detailed description about this function.

- **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 6.4.4, 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 6.4.4, 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 6.4.4, the user selects all elements in the blue part as the addendum part.

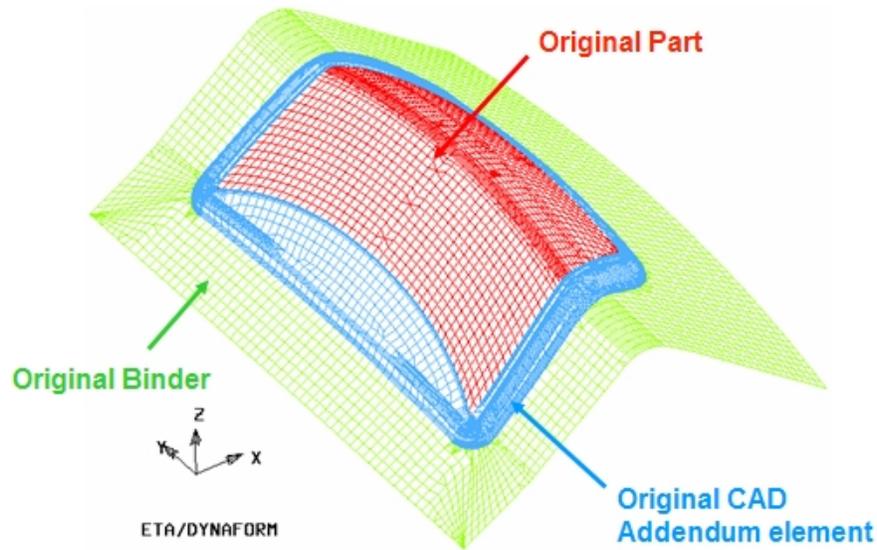


Figure 6.4.4 Classification of parts in Re-Engineering

#### 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 6.4.5.

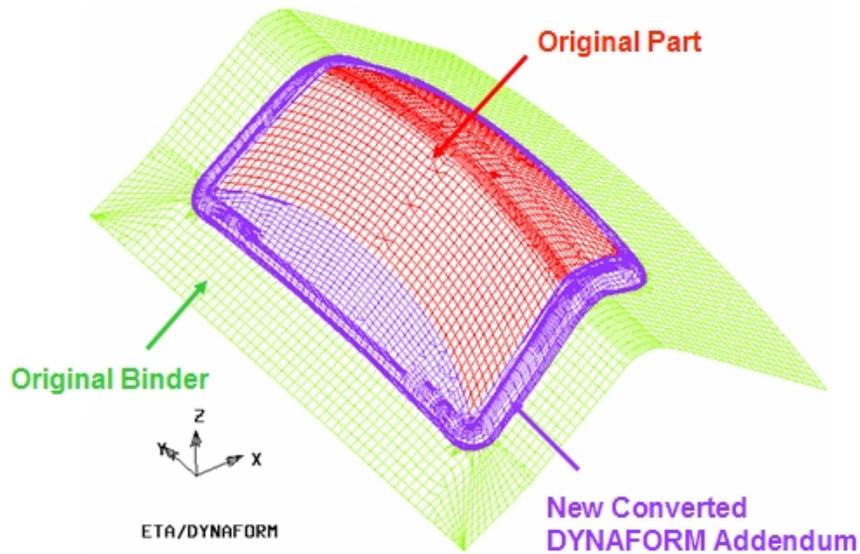


Figure 6.4.5 The new converted addendum in DYNAFORM

#### 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 from original part design to another part that shares similar shape. The dialog box of Create New Addendum is illustrated in Figure 6.4.6.

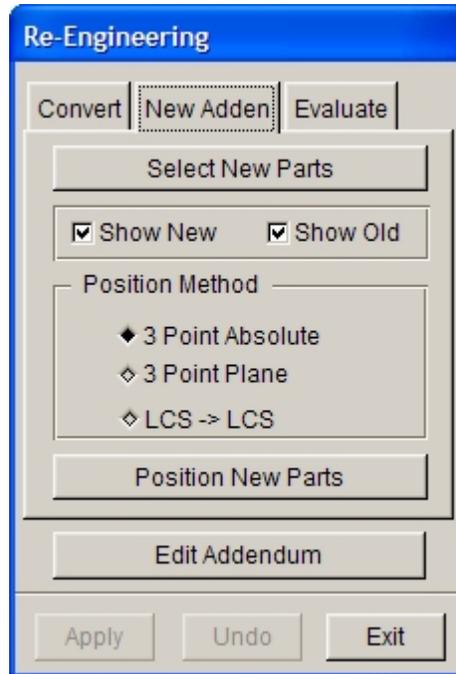


Figure 6.4.6 Create New Addendum dialog box

### 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 6.4.7.

**NOTE:** The distance between the selected three points should be large enough to ensure good positioning result.

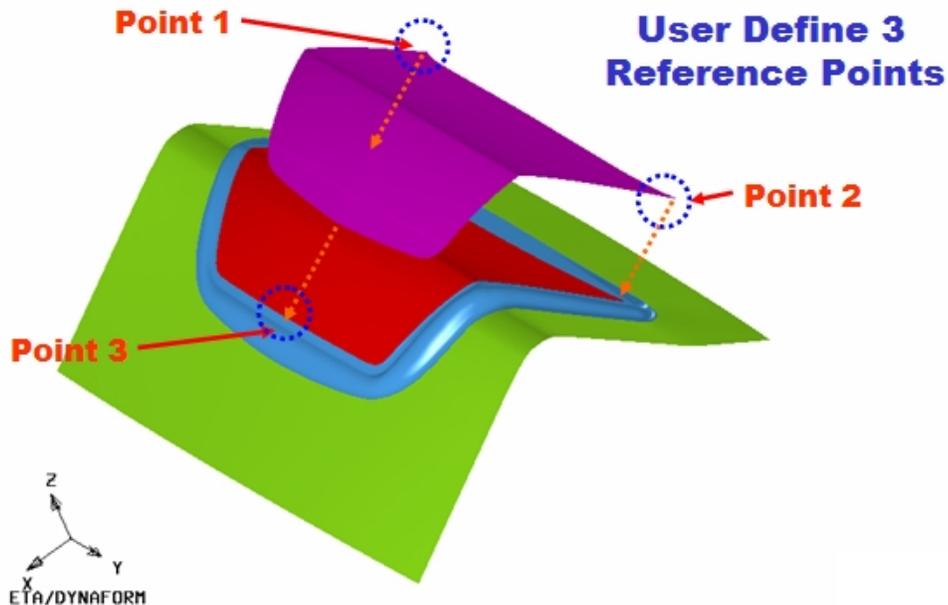


Figure 6.4.7 Three Points Absolute

- **3 Point Plane**

The average of the three selected points (the center point of three points) is used as 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 LCS on original die parts to position the new and original die parts.

### **UNDO**

Undo the last position operation.

### **APPLY**

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 6.4.8 illustrates the position of corresponding

points.

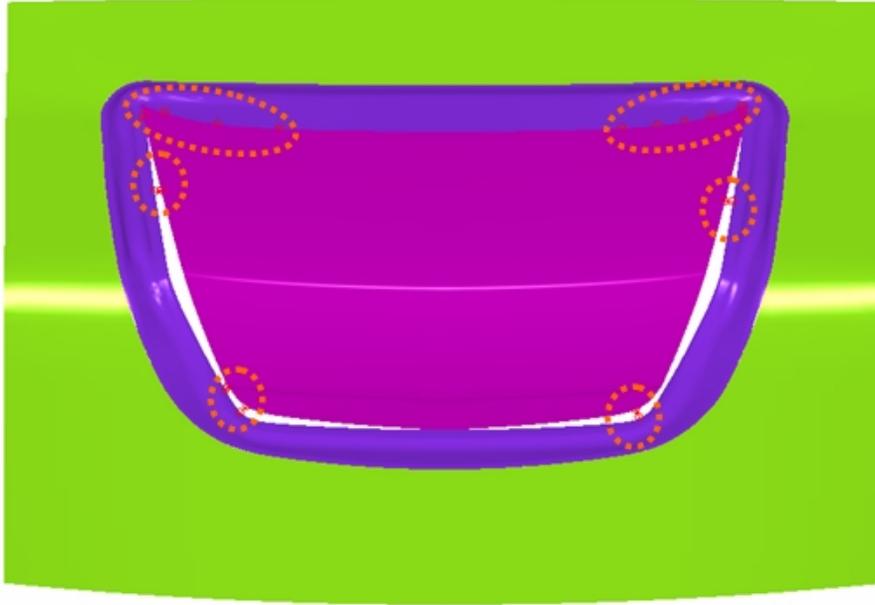


Figure 6.4.8 Moving the corresponding points

After the adjustment is completed, the user can click on middle mouse button to update the addendum on new die parts.

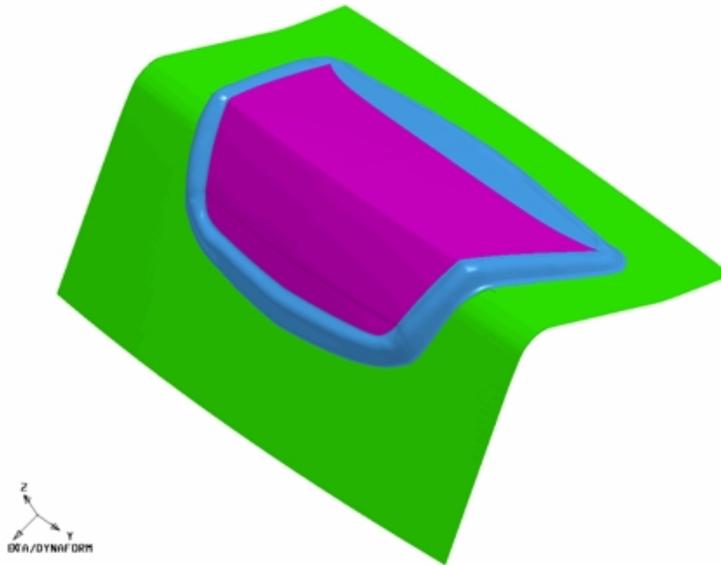


Figure 6.4.9 Updated addendum

### 6.4.3 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.

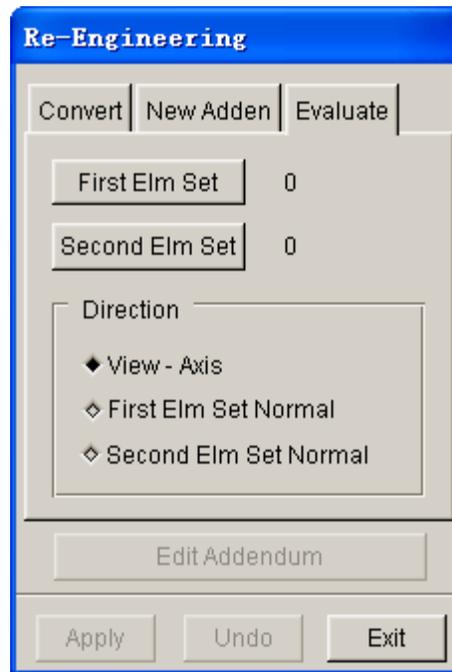


Figure 6.4.10 Evaluate dialog box

**DIRECTION**

This function enables user to define the direction that 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 sets 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.

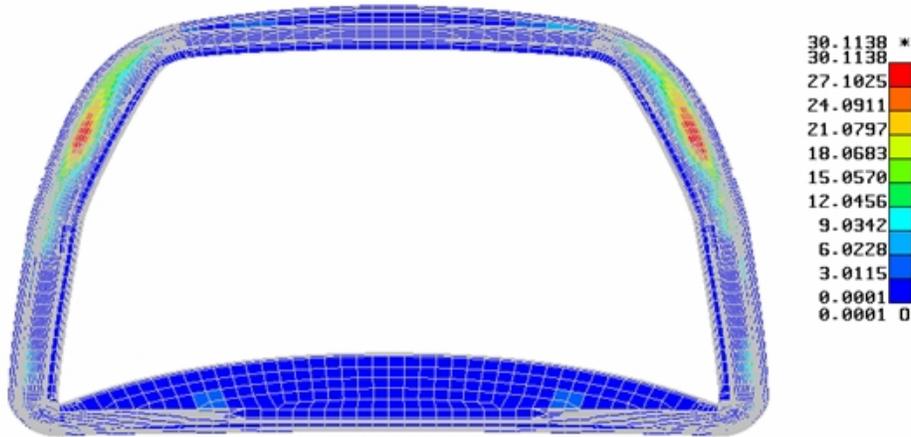


Figure 6.4.11 Contour of the Evaluate

## 6.4.4 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. Refer to section 6.3 for information about editing addendum.

## 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.

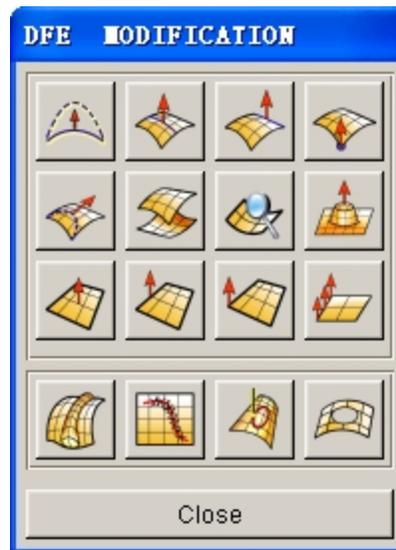


Figure 6.5.1 DFE Modification dialog box

## 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.

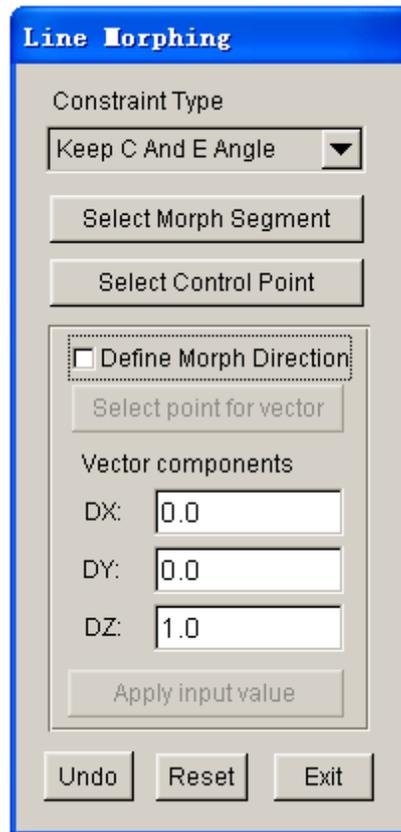


Figure 6.5.2 Line Morphing dialog box

- **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 5.1.15. 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 defines 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 MORPHING BY SURFACE INTERIOR



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.

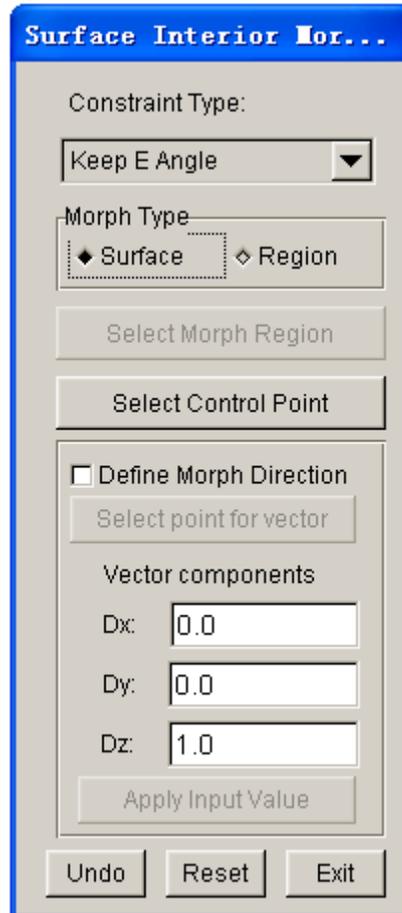


Figure 6.5.3 Surface Morphing – Interior

- 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 5.1.15.
- 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 5.1.15.
- UNDO aborts the last morphing operation. RESET sets the surface back to its original shape.

### 6.5.3 MORPHING BY SURFACE EDGE



This function uses a point on the surface boundary as the control point. It operates similarly to the INTERIOR MORPHING function.

### 6.5.4 MORPHING BY SURFACE CORNER



This function uses a corner point on the surface as the control point. It operates similarly to the INTERIOR MORPHING function.

### 6.5.5 MORPHING BY LINE ON SURFACE



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 6.5.4 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.

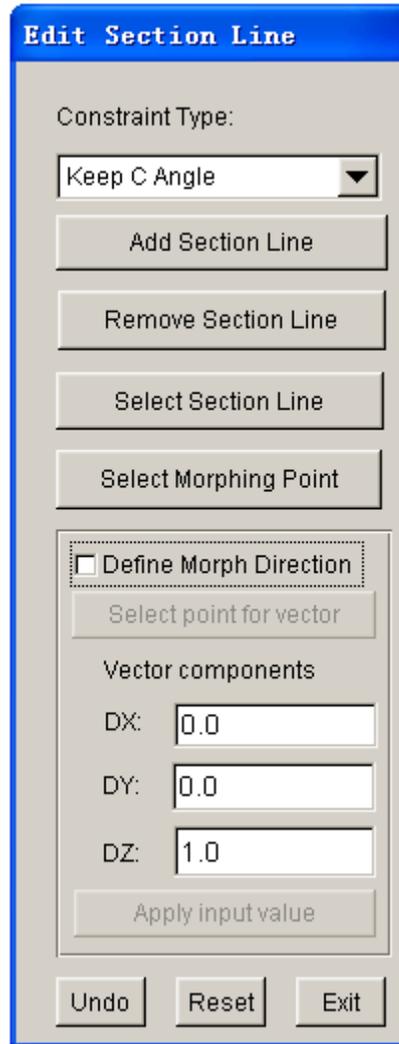


Figure 6.5.4 Section Line Morphing dialog box

- 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.

## 6.5.6 SURFACE EDIT



This function allows the user to edit surface by control lines. Refer to Section 6.1.1.5 for detailed description about this function.

## 6.5.7 SURFACE CHECK



This function allows the user to check TINY SURFACE, SURFACE BOUNDARY, CONTROL POINTS DISTRIBUTION, SPIKE / CUT SURFACE and SURFACE INTERSECTION, etc. Refer to Section 5.2.20 for detailed description about this function.

## 6.5.8 MORPHING BY ELEMENT



This function allows the user to morph a region of selected elements, as illustrated in Figure 6.5.5. All elements within the region must be selected without excluding any elements from the region. With the morphing of elements, the surfaces in the morphing region will also morph.

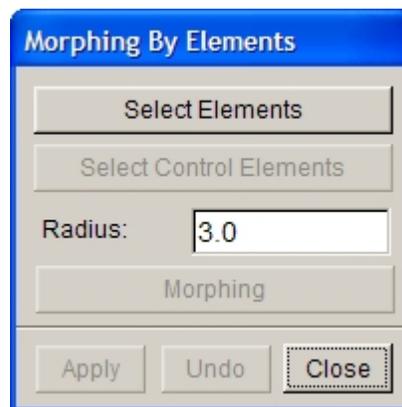


Figure 6.5.5 Morphing By Elements

Note: this function can only be used for the morphing of boss in parts.

- Click on the SELECT ELEMENTS button and select elements in the morphing region. Usually all the elements of the boss are selected.
- Then click on the SELECT CONTROL ELEMENTS button to select control elements for morphing. Usually the elements on the top of the boss are selected.
- Enter RADIUS value. RADIUS value is the transition radius of the morphing boss.
- Click on the MORPHING button and then move the cursor to the top of the boss. Hold and drag the left mouse button up and down to form the morphed boss outline, as shown in Figure 6.5.6.

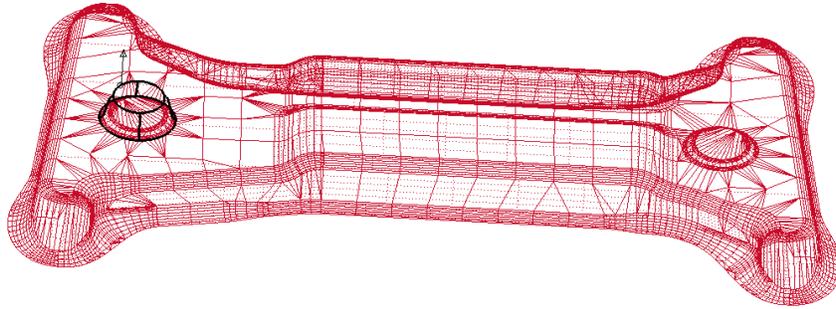


Figure 6.5.6 Boss Outline after Morphing

- Click on the APPLY button to generate the morphed boss, as shown in Figure 6.5.7.

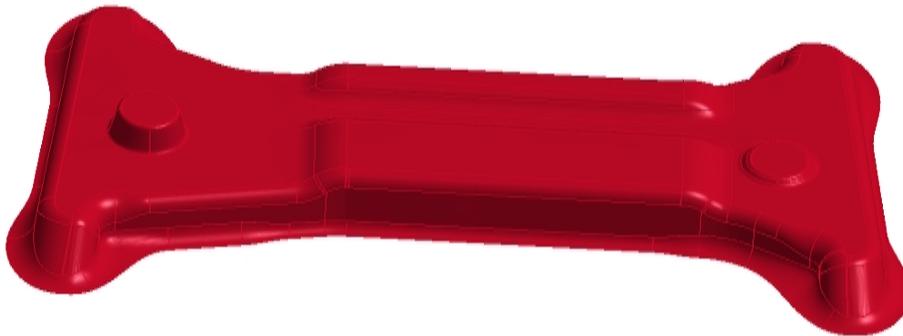


Figure 6.5.7 Morphed Boss

### 6.5.9 MORPHING BY INTERIOR ELEMENT



This function allows the user to morph the selected mesh region by using a node within the selected element domain as the control point. 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.

- 1) Select a group of elements to morph.
- 2) The following message is printed in the message prompt window:

***SELECT CONTROL NODE FOR MORPHING***

Select any interior node from the selected element group.

- 3) After the node is selected, the dialog box as illustrated in Figure 6.5.8 is displayed.



Figure 6.5.8 Morphing Options

The drop down button allows the user to select type of constraint for morphing. The description of each constraint type is discussed in Section 5.1.15.

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 6.5.9 illustrates a typical result (red) generated by element interior morphing.

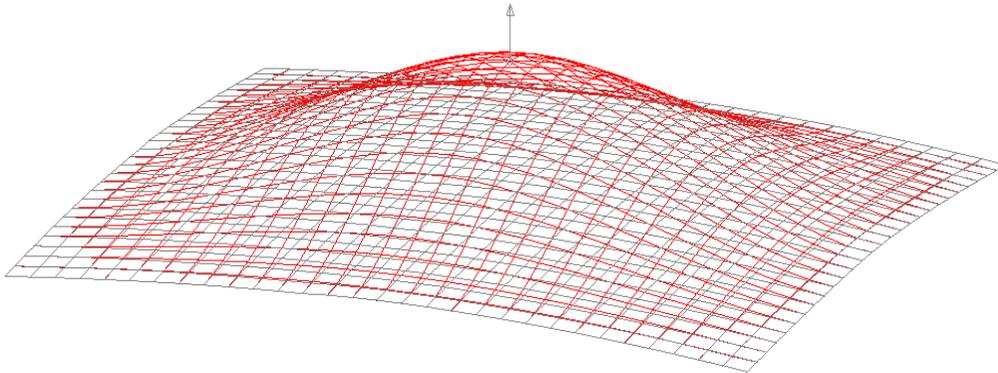


Figure 6.5.9 An illustration showing a typical result generated by Element Interior Morphing

- 4) Click on EXIT button to update the morphed elements and display the SELECT ELEMENTS dialog box.
- 5) Select other element to repeat the operation or click on CANCEL button to exit the function.

### 6.5.10 MORPHING BY ELEMENT EDGE



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.

### 6.5.11 MORPHING BY ELEMENT CORNER



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.

### 6.5.12 MORPHING BY 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.

### 6.5.13 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 6.5.10 shows the control parameters in the dialog box.

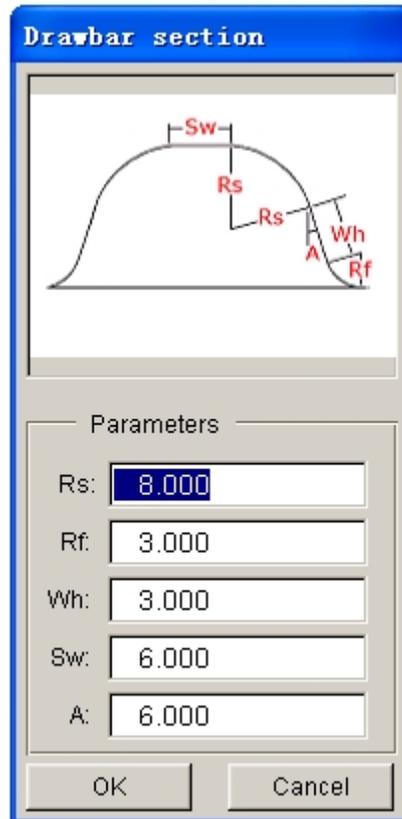


Figure 6.5.10 Parameters for Tear Drop Head

2. DRAW BAR ENDS WITH A CONICAL SHAPE (TEAR DROP TAIL). Figure 6.5.11 shows the parameters in the dialog box.

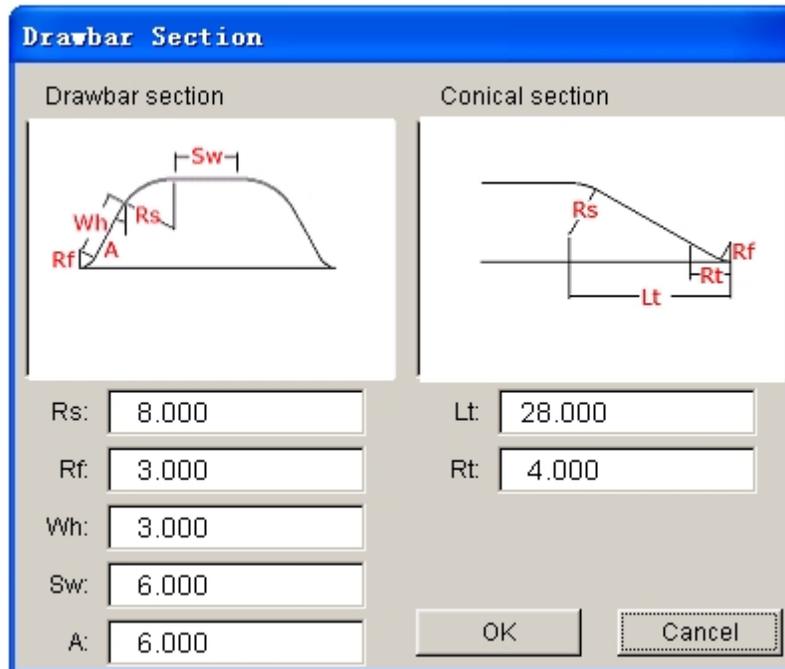


Figure 6.5.11 Parameters for Tear Drop

3. TEAR DROP (DRAW BAR WITHOUT MAIN BODY). Figure 6.5.12 shows the parameters in the dialog box.

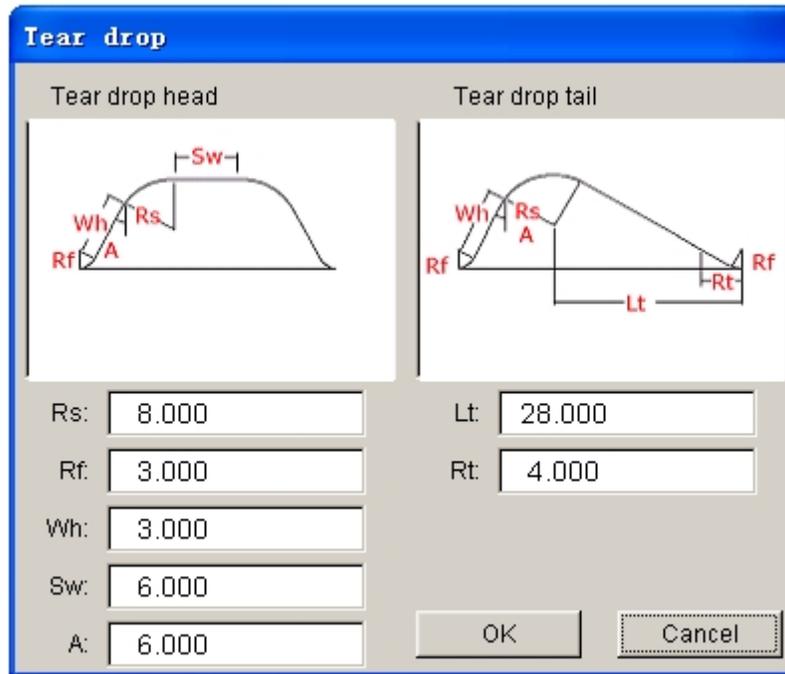


Figure 6.5.12 Parameters for Tear Drop

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 6.5.13a. 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 6.5.13b, 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 6.5.10-6.5.12.

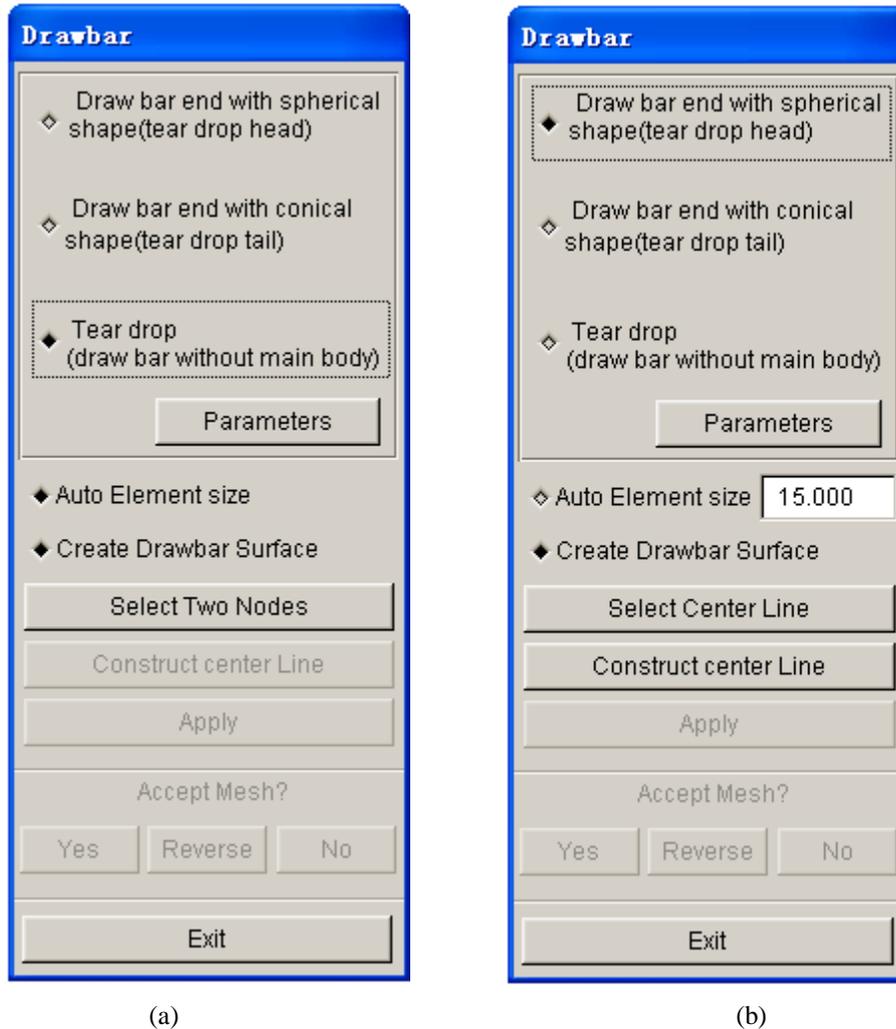


Figure 6.5.13 Drawbar dialog box (a) With Auto Element Size (b) With User Defined Element Size

- 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.

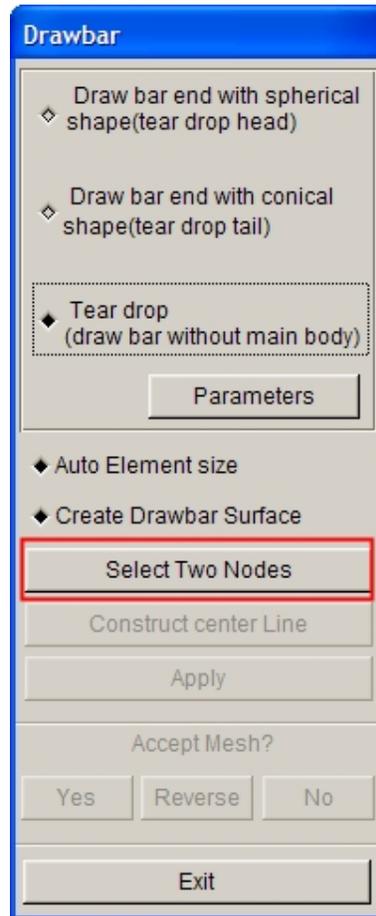


Figure 6.5.14 Tear Drop

Figure 6.5.15 shows the details of three types of drawbars.

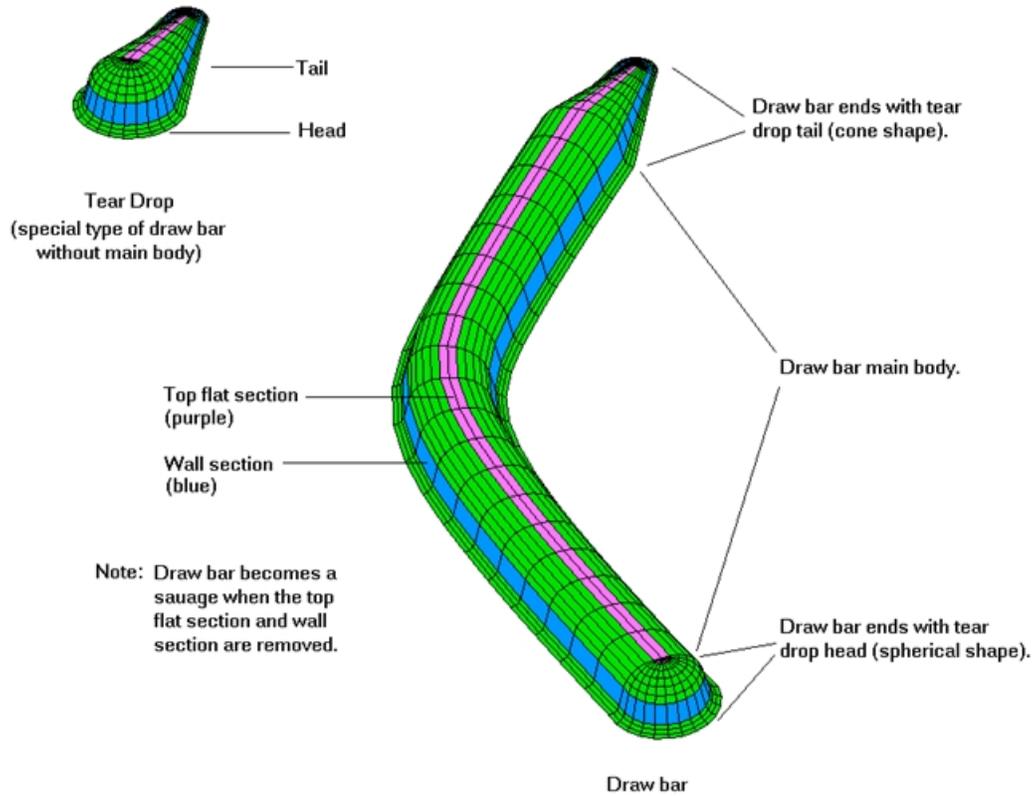


Figure 6.5.15 Three Types of Drawbars

### 6.5.14 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 6.5.16.
- Select OK to accept the selected line. The trimmed result is displayed along with a DYNAFORM QUESTION box as illustrated in Figure 6.5.18.
- Click YES to accept the mesh.

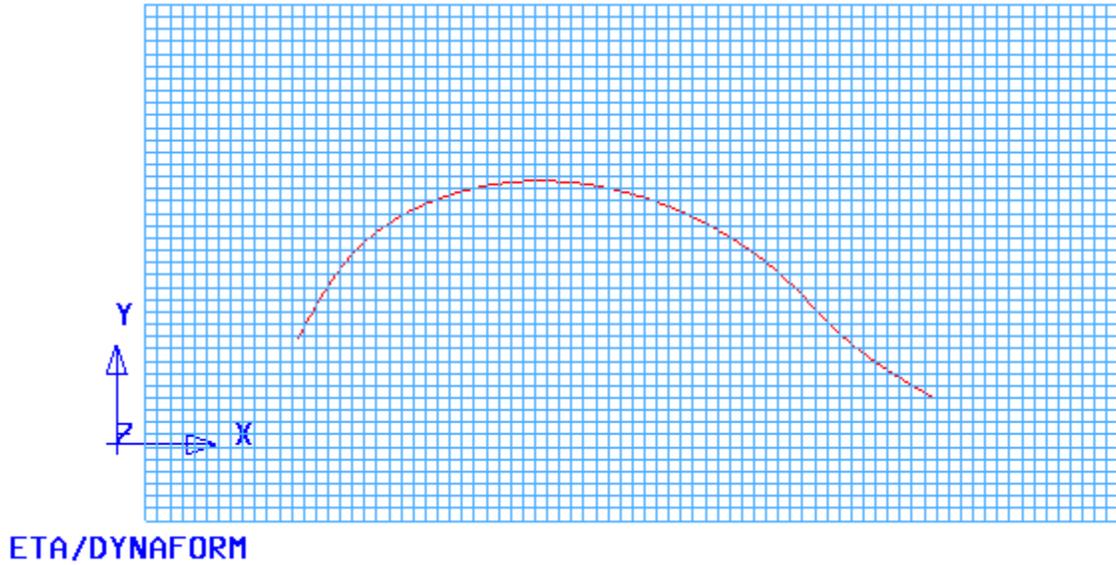


Figure 6.5.16 Select Drawhead Outline

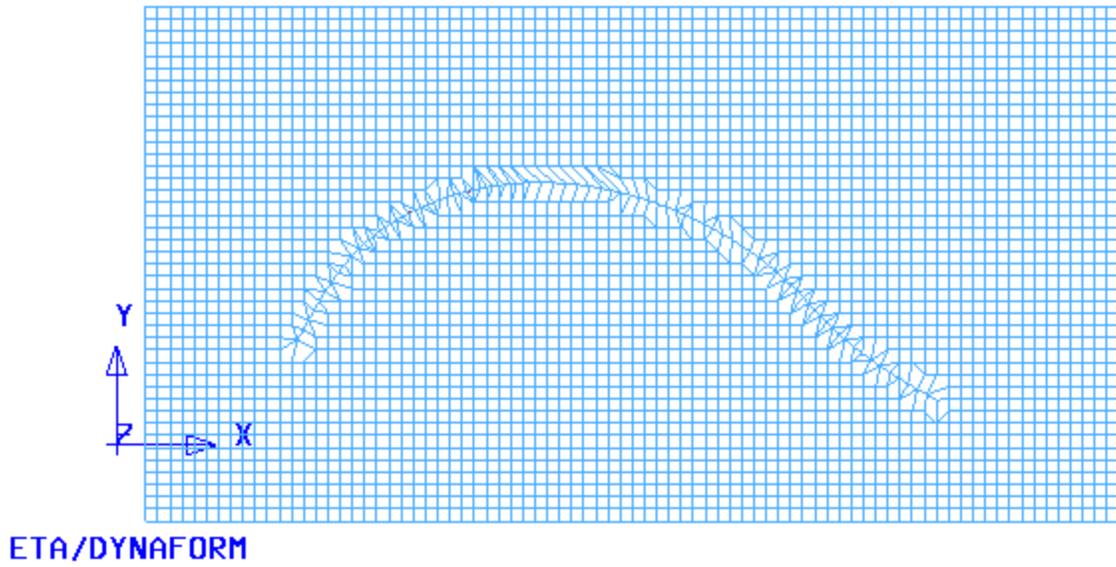


Figure 6.5.17 Trim Result

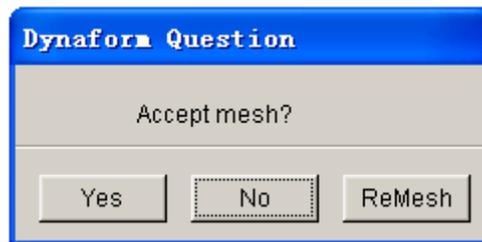


Figure 6.5.18 DYNAFORM QUESTION dialog box

### 6.5.15 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.

### 6.5.16 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 6.5.19.

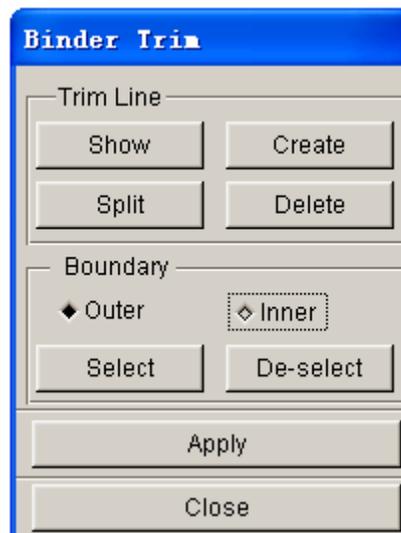


Figure 6.5.19 Binder Trim dialog box

- **EDIT LINE**

These functions enable the user to edit the trim line. Refer to Section 5.1 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.

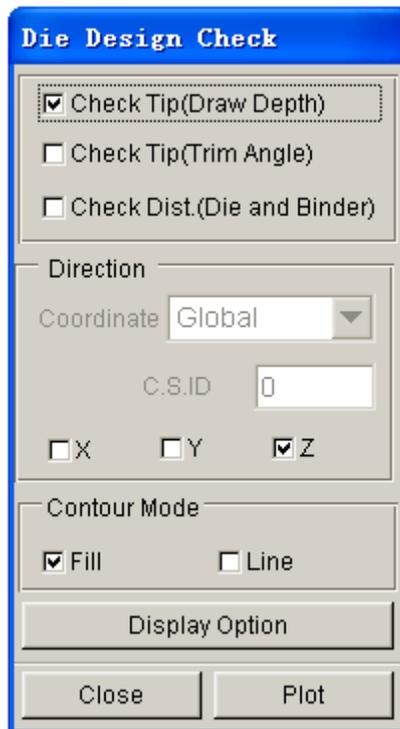


Figure 6.6.1 Die Design Check dialog box

### 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 add on module which is equipped with an advanced one step code MSTEP for rapid 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



### IMPORT

Enable the user to read in part geometry.



### CHECK DUPLICATE SURFACE

Enable the user to check and highlight duplicated surfaces. Duplicated surfaces are included in a new part. Refer to Section 5.2.21 for detailed description.



### MIDDLE SURFACE

Allow the user to create middle surfaces from a solid part. For detailed information, refers to Section 5.2.7.



### SEPARATE SURFACE

Allow the user to separate the TOP and BOTTOM surfaces from a solid part. Detailed information about this function is described in Section 5.2.8.



### UNFOLD FLANGE

Enable the user to unfold the flanges by incremental angles on a product. Refer to Section 6.1.1.4 for detailed information about unfold flange operation.



### **PART MESH**

Enable the user to mesh the part geometry using PART MESHER. Detailed information on PART MESHER is provided in Section 5.3.4.



### **MODEL CHECK/REPAIR**

Launch the MODEL CHECK/REPAIR dialog box. Detailed information about MODEL CHECK/REPAIR is provided in Section 5.5.



### **INNER FILL**

Automatically fill in any holes in the part mesh. Refer to Section 6.1.1.11 for more details.



### **TIPPING**

Position the part in stamping direction. Refer to Section 6.1.3 for more information about TIPPING.



### **BLANK SIZE ESTIMATE**

Unfold the part to obtain blank outline using MSTEP. Refer to Section 7.1.1.

## **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 OUTLINE. An example of BLANK SIZE ESTIMATION is shown in Figure 7.1.3.

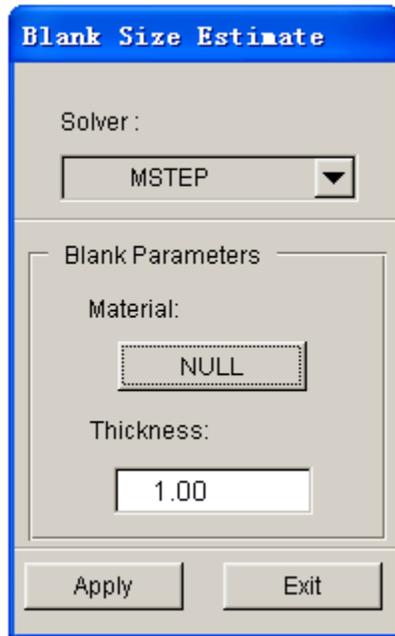


Figure 7.1.2 The Blank Size Estimate dialog box

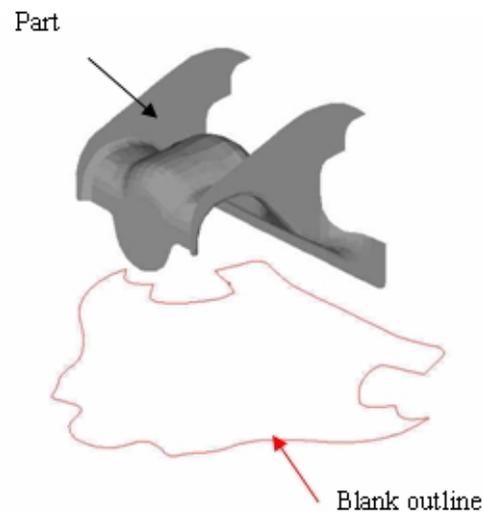


Figure 7.1.3 An example of Blank Size Estimation

## 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 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 Modified-One-Step solver developed by ETA. It provides an option for iterative one-step solution to achieve more accurate result with slightly longer computational time. It is primarily used to quickly assess formability and estimate the blank outline in the early stage of automobile body design and development cycle. It also supports evaluation of trimline.

### 7.2.1 DRAW

Figure 7.2.1 shows the interface of MSTEP. The method of defining tools is similar to method described in

Quick-Setup. The Sheet part is the final part geometry and is required as input for MSTEP calculation. The tools marked with blue color are optional input tools.

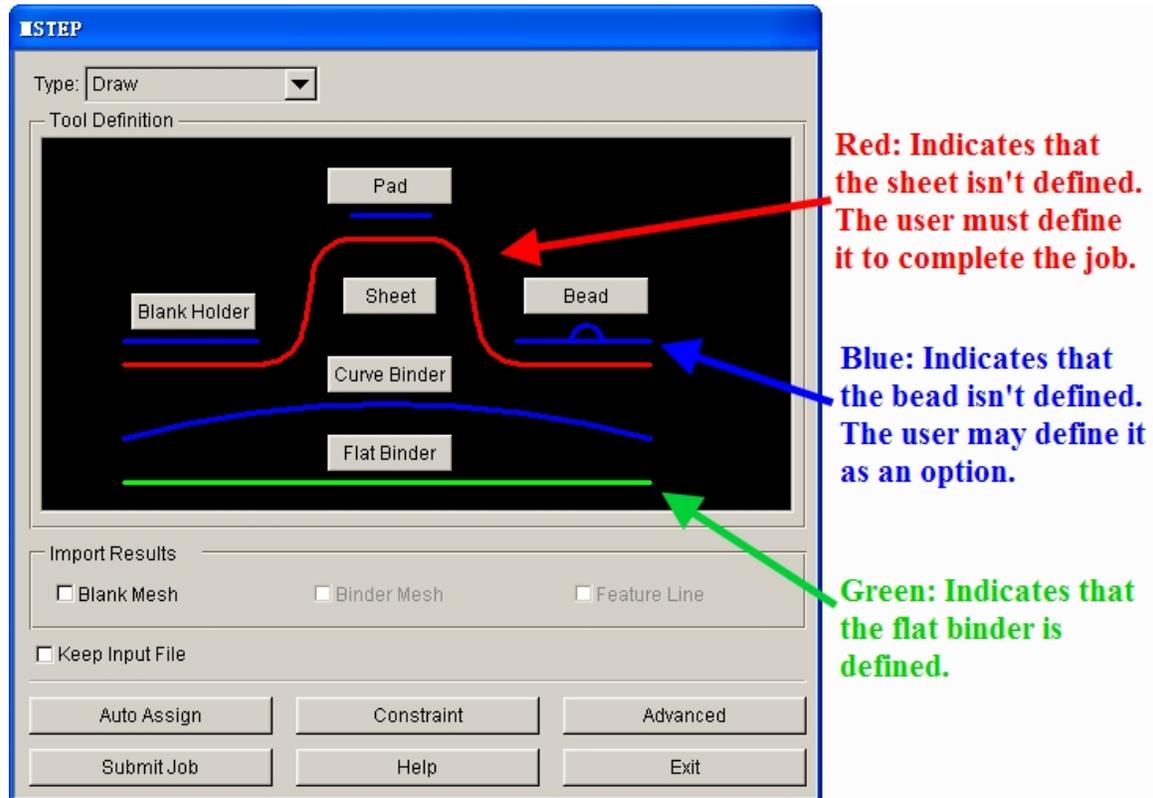


Figure 7.2.1 The Analysis GUI for MSTEP

**AUTO ASSIGN:** Automatically assigns tools and sheet with common names such as blank holder, pad, etc.

**CONSTRAIN:** Define constraints for the blank. Please refer to Preprocess / Boundary Condition / CONSTRAINTS for definition of constraint set.

**ADVANCED:** Allow the user to edit default parameters related to MESEP setting illustrated in Figure 7.2.2.

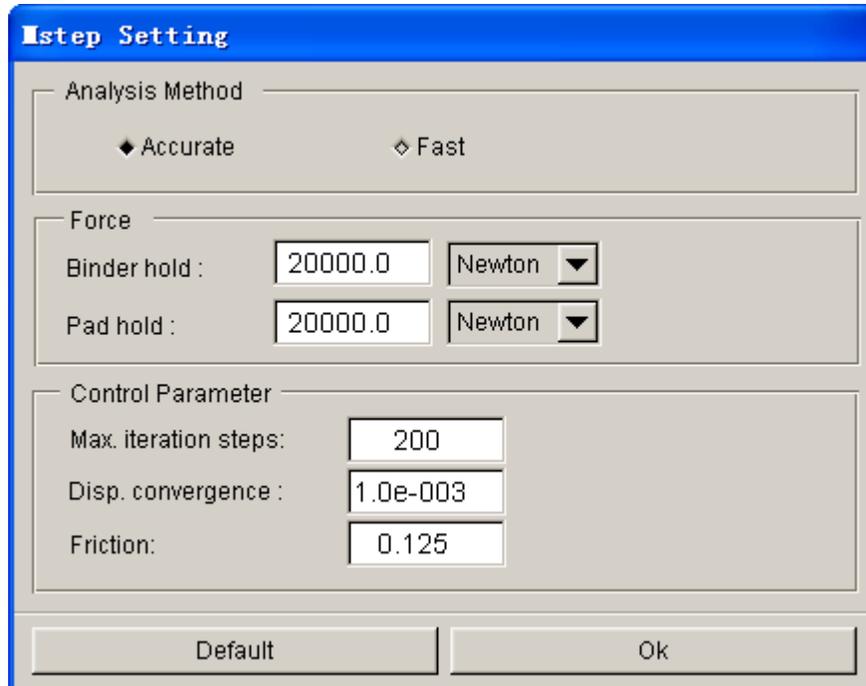


Figure 7.2.2 MSTEP Setting

In the MSTEP Setting dialog box, the user can choose one of the following Analysis Method: Accurate and Fast. In accurate analysis, both plasticity and elastic deformation of blank are considered, also in addition to all other boundary conditions such as friction and drawbead. In fast analysis, the elastic deformation is considered. The result obtained using Accurate method is much better than Fast method for formability cases where large blank deformation takes place. Of course, the solving time of Accurate method is longer than the Fast method.

#### FORCE

**Binder Hold:** Define amount of blankholder force. The default blankholder force is 20000 Newton. User can switch the unit of force from Newton to Ton.

**Pad Hold:** Define amount of pad force. The default pad force is 20000 Newton.

#### CONTROL PARAMETER

**Max. Iteration Steps:** Define the Max. iteration steps during calculation. If calculation does not convergence after reaching the max. iteration steps, an error message is printed in the \*.out file to inform the user. The default max. iteration steps is 200.

**Disp. Convergence:** Define the displacement convergence for calculation. The default displacement convergence is 0.001.

**Friction:** Define the friction between the blank and tools. The default friction is 0.125. User can change the value according to the real scenario.

**HELP:** Provide useful tips for stamping simulation using MSTEP.

**SUBMIT JOB:** Allow the user to submit the current job to the MSTEP solver.

**EXIT:** Allow the user to exit the MSTEP interface with submitting the simulation.

## 7.2.2 TRIM LINE

This is a new one-step solver developed by ETA for estimating trimline. The user-friendly Trimline GUI is illustrated in Figure 7.2.3. The user must define parts highlighted in red for Trimline calculation, while blue part is option. The tool definition and parameter setting are similar to those described in the previous section.

*Note: The flange of part is defined as SHEET in Trimline evaluation.*

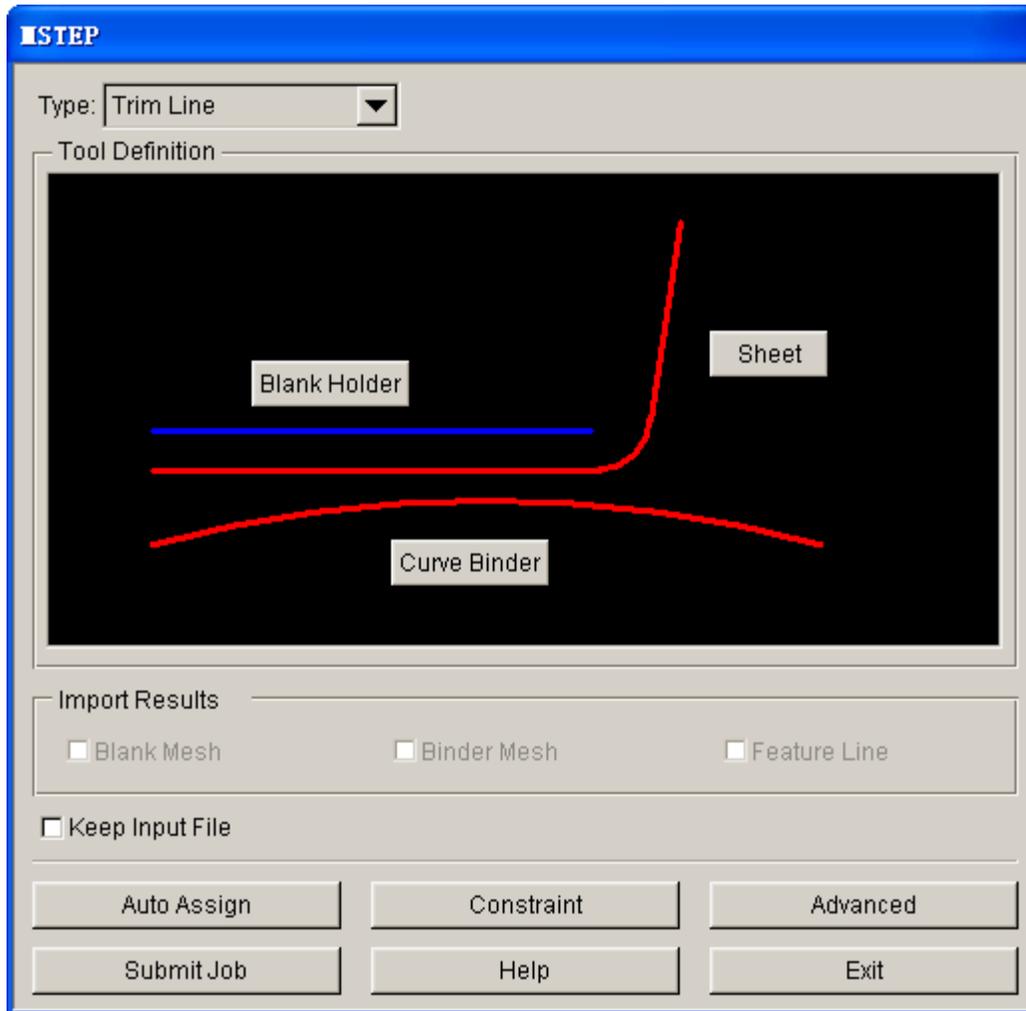


Figure 7.2.3 Trim Line dialog box

## 7.3 UNFOLD

This is a new one-step solver developed by ETA for unfolding complicated parts. MULTI UNFOLD allows the user to unfold the selected flange element. The program will automatically add the constraint information and set binder as an option. The MULTI UNFOLD dialog box is illustrated in Figure 7.3.1.

**MATERIAL:** Define blank material.

**THICKNESS:** Define blank thickness.

**CURVE BINDER:** Define surface of curve binder. This function is optional. If curve binder is defined, the color of button tag is changed from blue to light green.

**UNFOLD REGION:** Define region to be unfolded. If unfold region is defined, the calculation is automatically activated.

**ADVANCED:** This option allows the user to change the default setting of advanced MSTEP parameters such as binder force, pad force, friction, max. iteration step and displacement convergence. Refer to Section 7.2.1 for description about defining advanced parameters.

**LINE:** Automatically import the unfolded line and display it on the screen upon completion of calculation.

**MESH:** Automatically import the unfolded mesh and display it on the screen upon completion of calculation.

**KEEP INPUT FILE:** Keep input deck in the directory.

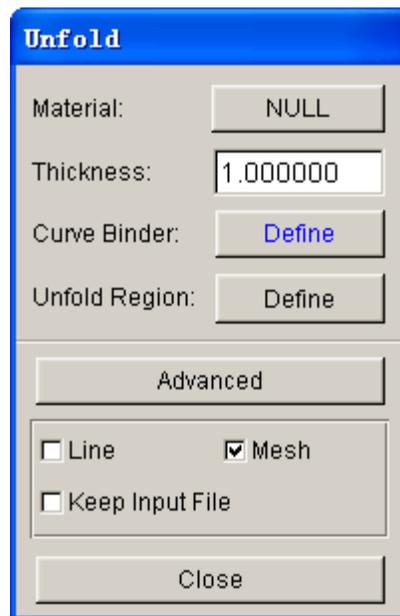


Figure 7.3.1 Multi Unfold dialog box

## 7.4 DEVELOPMENT

After obtaining the estimated blank outline, BLANK DEVELOPMENT enables the user to fine-tune the blank outline for subsequent application such as 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 illustrated in Figure 7.4.1.



Figure 7.4.1 The Blank Development dialog box

### 7.4.1 BLANK GENERATION



This function enables the user to generate flat blank mesh using the blank outline. The elements generated using BLANK GENERATION is QUAD (quadrilateral) dominant (as shown in Figure 7.4.2) which is required for an incremental stamping simulation using LS-DYNA.

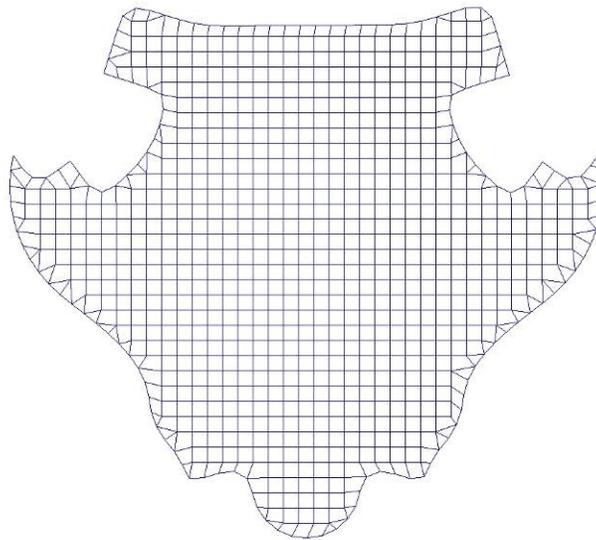


Figure 7.4.2 Blank mesh generated by Blank Generator

### 7.4.2 OUTER SMOOTH



The OUTER SMOOTH functions enable the user to smooth blank outer boundary. Refer to Section 6.1.1.10 for more information about the OUTER SMOOTH functions. An example of smoothing blank boundary is illustrated in Figure 7.4.3.

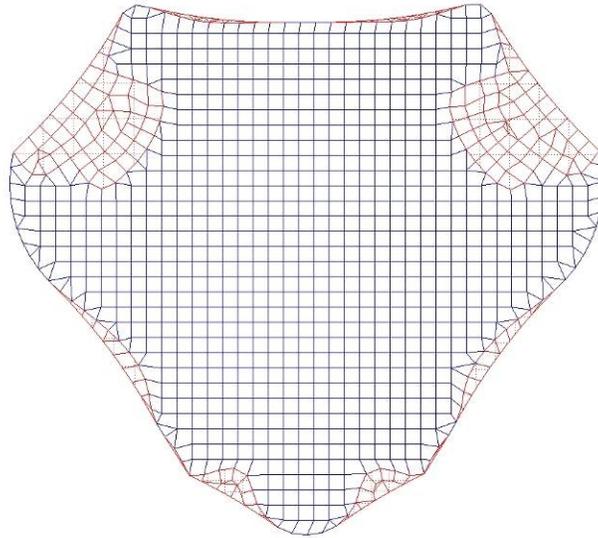


Figure 7.4.3 Blank mesh after smoothing

### 7.4.3 BLANK FITTING



This function enables the user to fit the estimated blank outline by using a rectangle or trapezoid. Dimension of rectangle or trapezoid is displayed upon completion of blank fitting operation. The dialog box is illustrated in Figure 7.4.4.

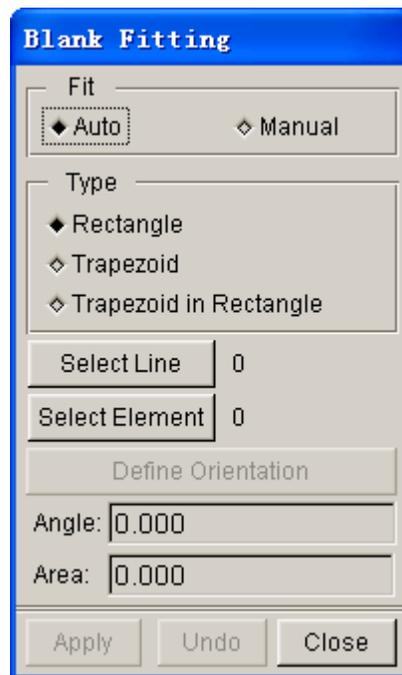


Figure 7.4.4 The Rectangular Fitting dialog box

There are two types of BLANK FITTING: AUTO and MANUAL. If AUTO FIT is selected, the Angle

(rotation) and Area (rectangular blank) are automatically determined by the program. An example of rectangular blank outline created using AUTO and MANUAL FIT is illustrated in Figure 7.4.5.

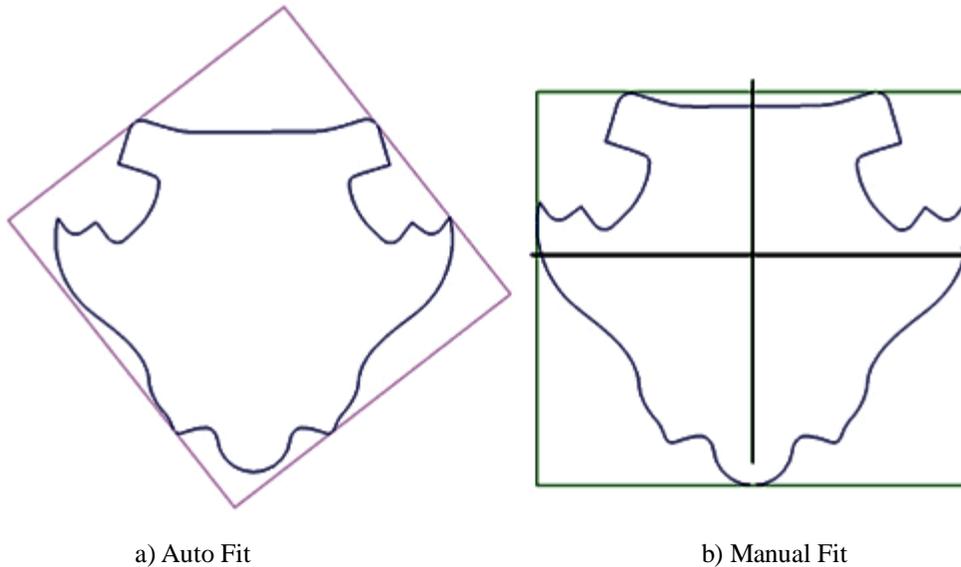


Figure 7.4.5 Rectangular blank outline created by Rectangular Fit

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.
- When generating the rectangular outline, the length and width of the rectangle are marked.

#### **BLANK FIT TYPE**

##### **RECTANGLE**

If this type of blank fitting is chosen, the program fits the blank using a rectangle illustrated in Figure 7.4.6

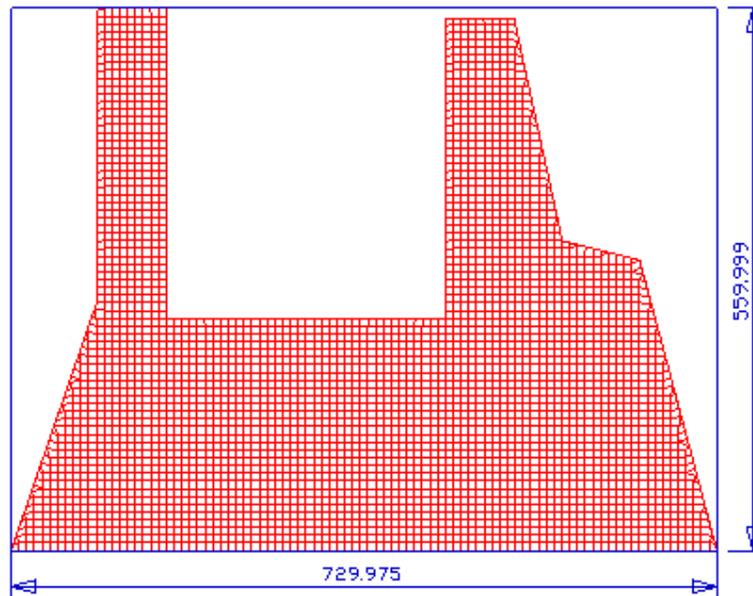


Figure 7.4.6 Rectangle Fit

**TRAPEZOID FIT**

If this type of blank fitting is chosen, the program fits the blank using a trapezoid illustrated in Figure 7.4.7.

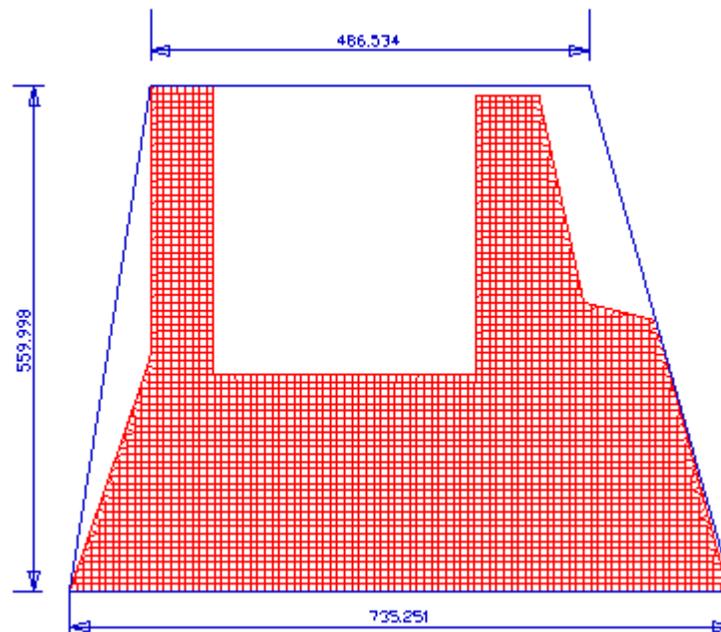


Figure 7.4.7 Trapezoid Fit

**TRAPEZOID IN RECTANGLE**

If this type of blank fitting is chosen, the program fits the blank using a Trapezoid in a smallest rectangle illustrated in Figure 7.4.8. The difference between TRAPEZOID IN RECTANGLE and TRAPEZOID is insignificant. However, the TRAPEZOID IN RECTANGLE option helps to avoid the shape angle for some specific examples.

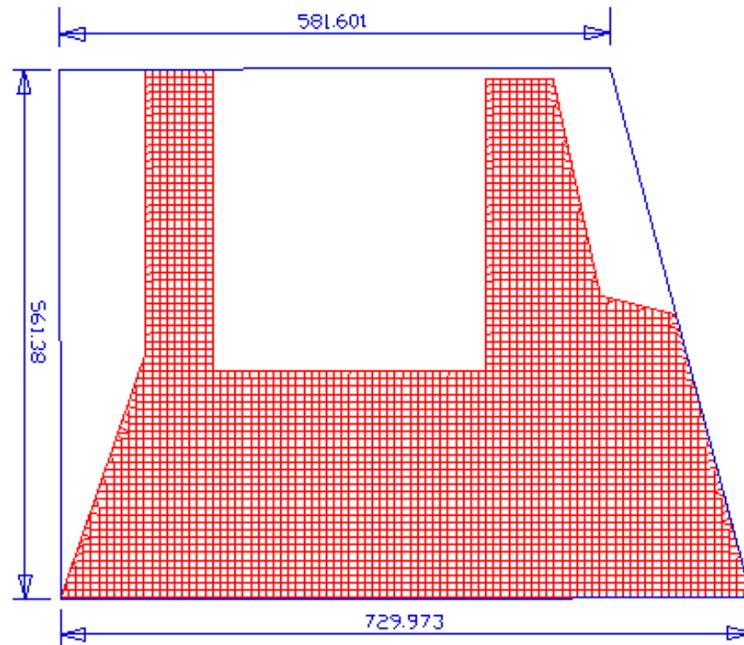
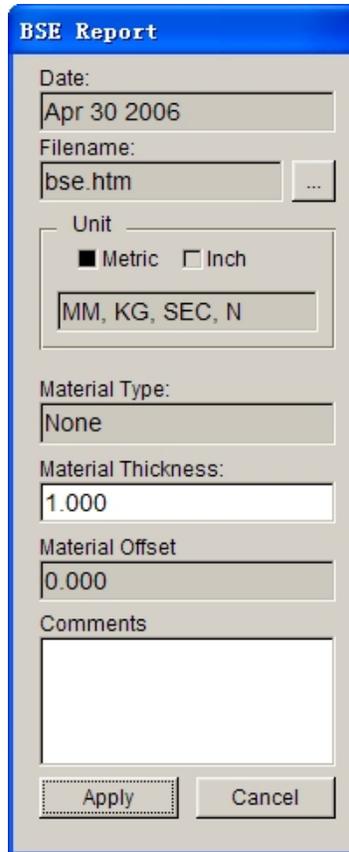


Figure 7.4.8 Trapezoid in Rectangle Fit

#### 7.4.4 BSE REPORT



This function enables the user to generate a summary report of the Blank Size Estimation, which is saved as in \*.htm format. The interface of the Blank Size Estimate Report is illustrated in Figure 7.4.9



**BSE Report**

Date: Apr 30 2006

Filename: bse.htm ...

Unit  
 Metric  Inch  
MM, KG, SEC, N

Material Type: None

Material Thickness: 1.000

Material Offset: 0.000

Comments

Apply Cancel

Figure 7.4.9 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 is captured as JPG image and included to the BSE report in .htm format.

**DATE**

Display the date of the blank evaluation report.

**FILE NAME**

Enable the user to assign a file name and path to save the current blank evaluation report.

**UNIT**

Display current unit system in the blank evaluation report.

**MATERIAL TYPE**

Display name of the selected blank material.

**MATERIAL THICKNESS**

Display selected material thickness. The user may edit this field.

**MATERIAL OFFSET**

Display the user defined material offset used for calculations.

**COMMENTS**

Enable the user to write comments about blank evaluation.

Click on the Apply button to save the current blank evaluation report in the .htm format. The .htm file is automatically displayed in the default web browser (PC with Windows operating system). A sample of BSE report is illustrated in Figure 7.4.10.

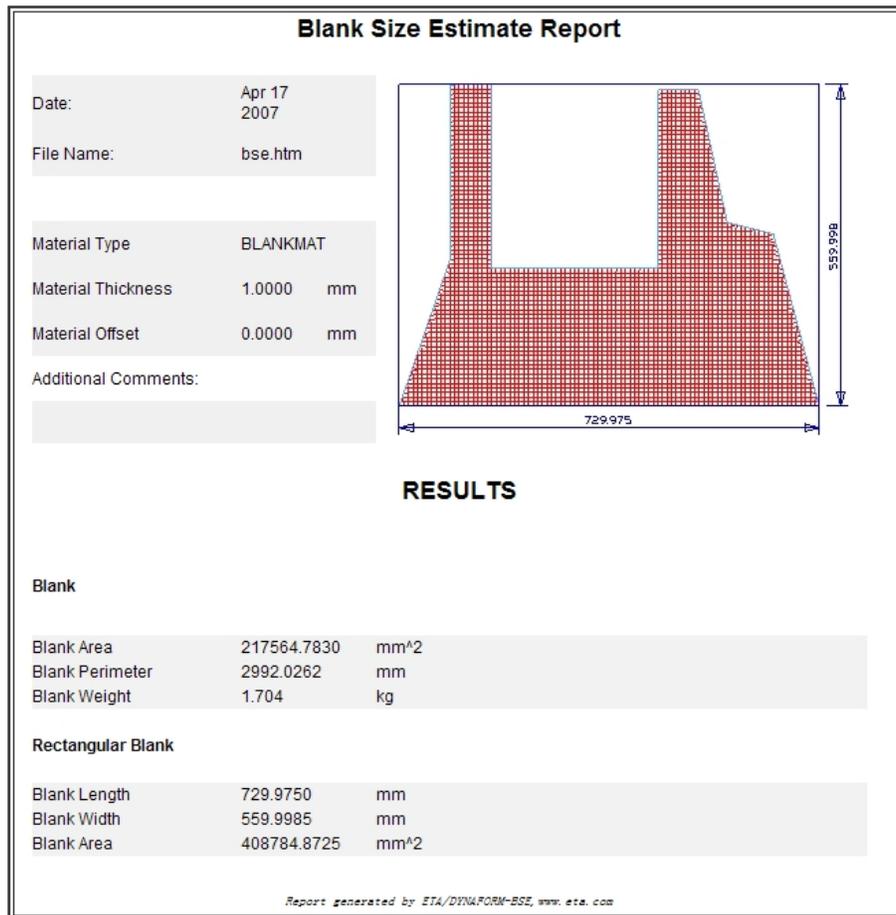


Figure 7.4.10 A sample blank evaluation report

## 7.4.5 EXPORT



This function enables the user to export the blank outlines in various CAD data format.

## 7.4.6 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 7.4.11 shows the types of nesting options available in the BLANK NESTING menu.



Figure 7.4.11 Blank Nesting Types

### ONE-UP NESTING

Enable the user to generate the nesting result in one row.

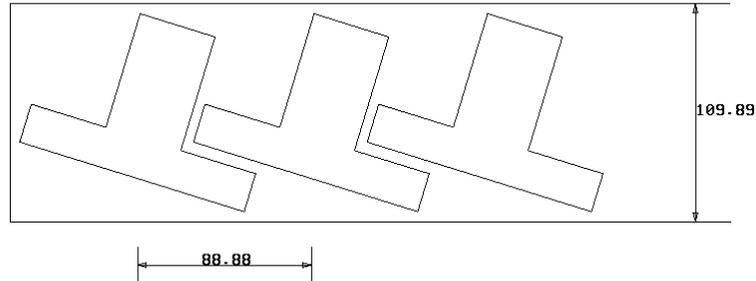


Figure 7.4.12 One-up Nesting

### TWO-UP NESTING

Enable the user to generate the nesting result with two identical blanks using a common orientation.

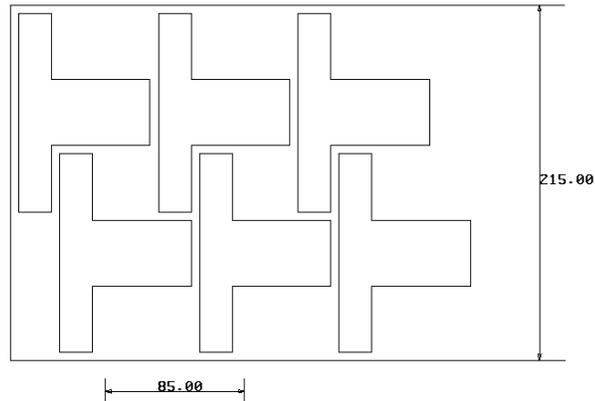


Figure 7.4.13 Two-up Nesting

### TWO-PAIR NESTING

Enable the user to generate the nesting result with a pair of identical blanks in opposite orientation.

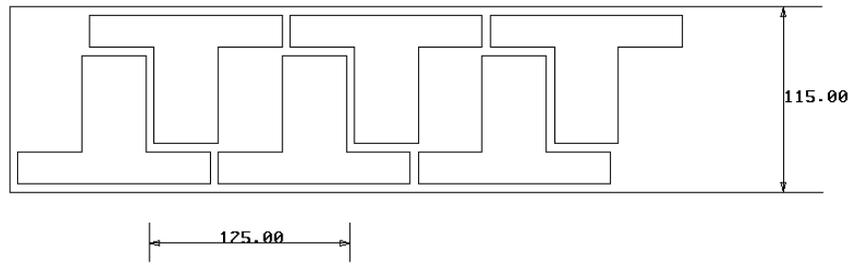


Figure 7.4.14 Two-pair Nesting

**MIRROR NESTING** 

Enables the user to generate the nesting result by mirroring the blank.

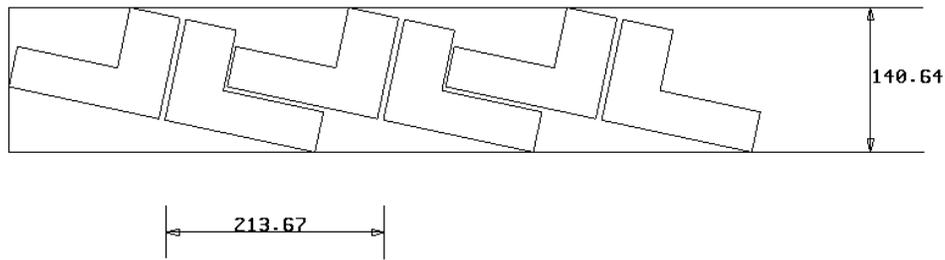


Figure 7.4.15 Mirror Nesting

**MULTI BLANKS NESTING** 

Enable the user to generate the nesting result using multiple blanks with different size and shape.

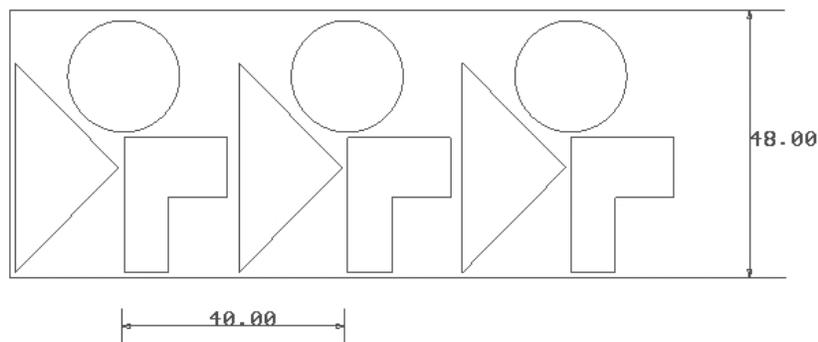


Figure 7.4.16 Multi blanks Nesting

Examples of blank nesting are illustrated in Figures 7.4.12 to 7.4.16. A detailed description of each function is given in the following section.

### 7.4.6.1 ONE-UP NESTING



This function enables the user to generate the nesting result in one row. The dialog box is shown in Figure 7.4.17.

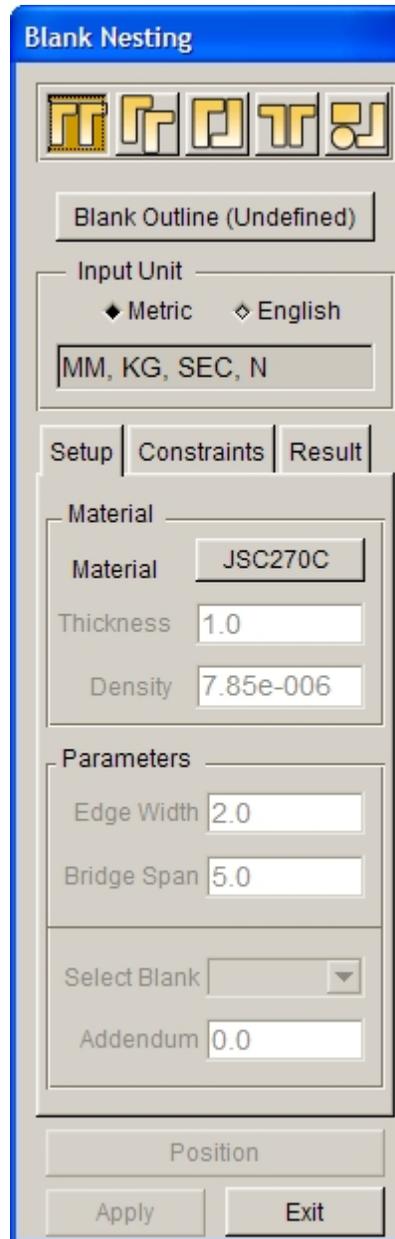


Figure 7.4.17 One-up Nesting dialog box

- **Blank Outline (Undefined)**

This parameter is used to define the Blank Outline. Click on the **Blank Outline (Undefined)** button in Figure 7.4.17, 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**

Enable the user to define the nesting setup parameters in metric unit.

**English**

Enable the user to define the nesting setup parameters in inch unit.

**SETUP**

In SETUP tab, the user can define the basic parameters such as material, edge width, bridge span and addendum size.

**MATERIAL**

The user can define blank material using two methods. In first method, the user can click on the MATERIAL button to open the MATERIAL dialog box. The user may choose to create a NEW material or IMPORT a specific material from the MATERIAL dialog box. Once the material is defined, the density of material is automatically displayed in the nesting window. The user proceeds to define part thickness. Another method is by entering the THICKNESS and DENSITY directly in the corresponding input data fields.

**Thickness**

Enable the user to define blank thickness.

**Density**

Enable the user to define density of material.

**PARAMETERS**

The user may define additional parameters to control the layout of blank nesting. These parameters include Edge Width, Bridge Span and Addendum. Figure 7.4.18 shows these parameters related to the blank nesting application.

**Edge Width**

Allow the user to define the gap between blank outline and strip edge.

**Bridge Span**

Allow 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**

Allow the user to expand the selected blank outline.

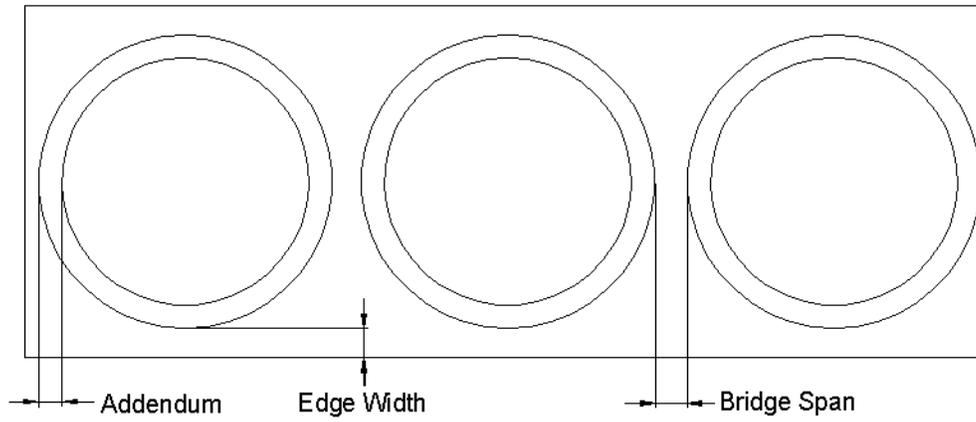


Figure 7.4.18 Edge Width, Bridge Span and Addendum

### 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 7.4.19.

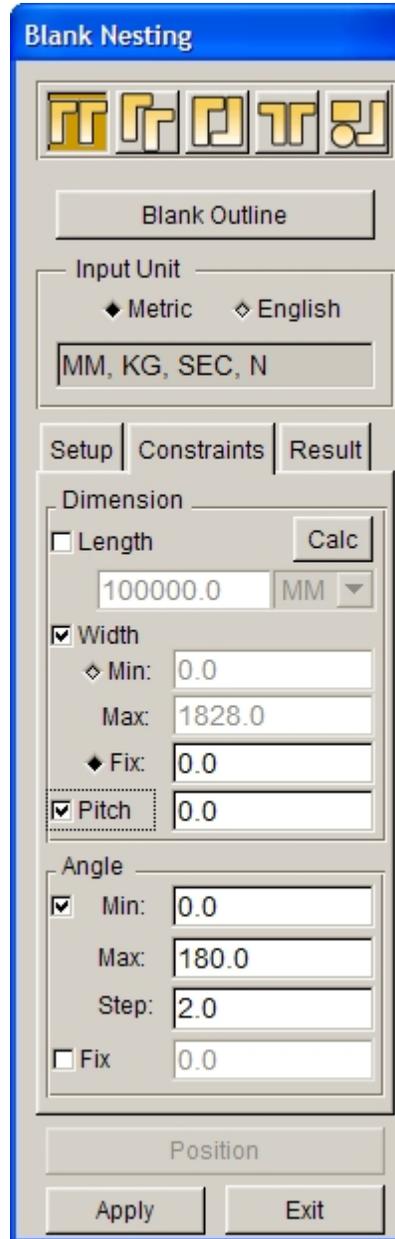


Figure 7.4.19 Constraints

## DIMENSION OPTION

### LENGTH

Fix Length is used to restrict the strip length. The user can click on  to calculate the strip length. After CALC button is hit, the Calculator dialog box illustrated in Figure 7.4.20 is displayed. The UNIT in the dialog box depends on the UNIT selected in blank nesting. The user can select ID and OD value, which is configured by coilinfo.txt file in installation directory. After these two values are selected, the strip length will be displayed in the LENGTH field.

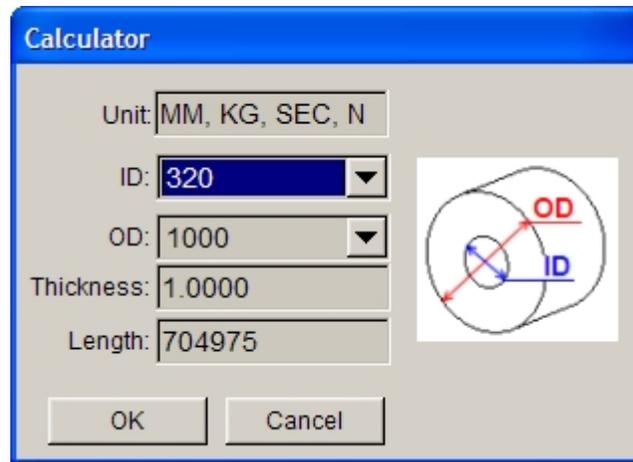


Figure 7.4.20 Calculator dialog box

**WIDTH**

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). Blank 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**

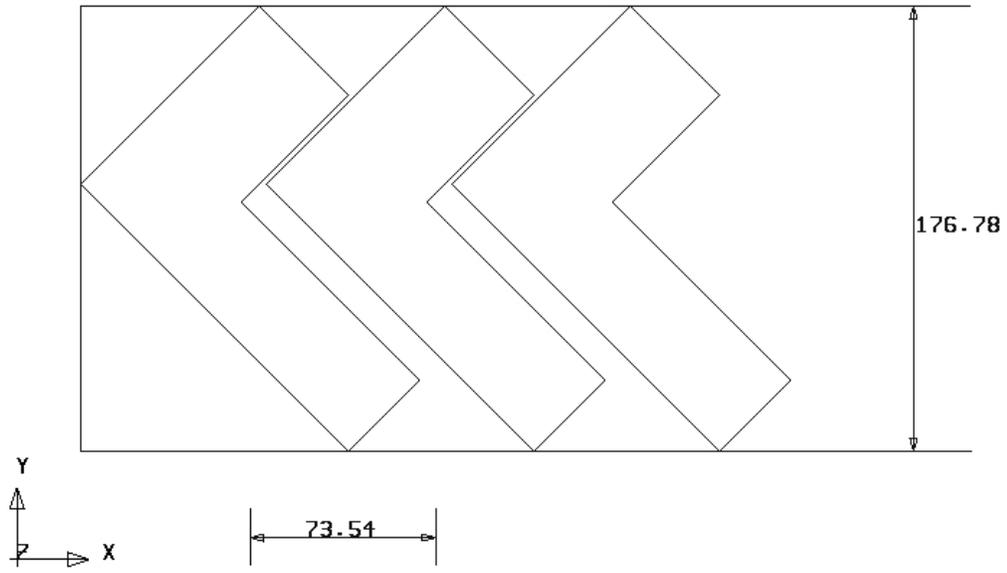
Constrain the feeding of coil.

**ANGLE OPTION**

- The angle range is used as the angle constraint. The user defines the allowable range of blank rotation (Min and Max). Blank nesting calculates a series of results according to the defined range of the rotation angle.
- Fix angle is used to specifically define a constant rotation angle.

**APPLY**

Click on the Apply button illustrated in Figure 7.3.19 to run the blank nesting and show the optimal material utilization, as illustrated in Figure 7.4.21.



ETA/DYNAFORM

Figure 7.4.21 The Result of One-up Nesting

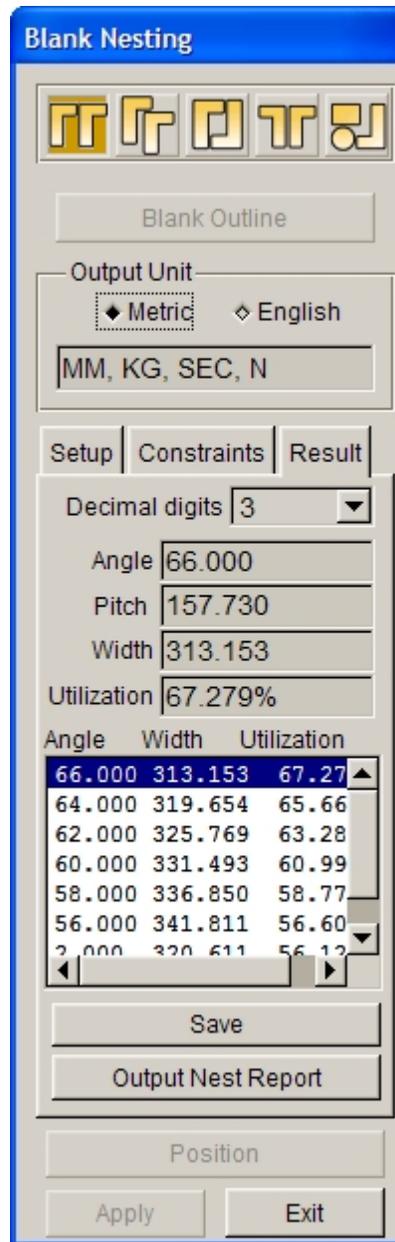


Figure 7.4.22 Result of Blank Nesting

Blank nesting calculates a series of results according to input constraints. The user may edit the number of decimal digit by choosing the number from the Decimal digits combo box. The user may select any result from the list to display the corresponding constraints in the Result tab illustrated in Figure 7.4.22.

**NOTE:** The user can not change type of nesting or unit system when nesting result is obtained.

- **Output Nest Report**

This function enables the user to output the nesting result in HTML format. The interface of the Output Nesting Report is illustrated in Figure 7.4.23.

**Nest Report**

Date:  
Jun 27 2006

Filename:  
test.htm

Blank Outline Color ■

Output Unit  
MM, KG, SEC, N

Production Volume / Per Coil  
Production Volume

Production Volume  
0

Coil Length  
0.000 MM

Base Material Cost  
0.000 \$/kg

Extra Material Cost  
0.200 \$/kg

Scrap Value Cost  
0.024 \$/kg

Consumables Cost  
0.350 \$/blank

Comments  
A sample of Nesting report

Apply Cancel

Figure 7.4.23 The Output Nesting Report dialog box

When Output Nesting Report dialog box is displayed, a highlighted frame appears in the display area. The image displayed inside this frame is captured saved to the nesting report.

**DATE**

Display 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 color of the blank outline.

**UNIT**

Display current unit system in 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 production volume.

**COIL LENGTH**

Display coil length defined in Constraints.

**BASE MATERIAL COST**

Enable the user to define base material cost.

**EXTRA MATERIAL COST**

Enable the user to define extra material cost.

**SCRAP VALUE COST**

Enable the user to define scrap value cost.

**CONSUMABLES COST**

Enable the user to define consumable cost.

**COMMENTS**

Enable the user to write comments in the nesting report.

Click on the Apply button to save the current nesting report as.htm file. Then, the file automatically displayed in the default web browser (PC with Windows operating system). A sample nesting report is illustrated in Figure 7.4.24.

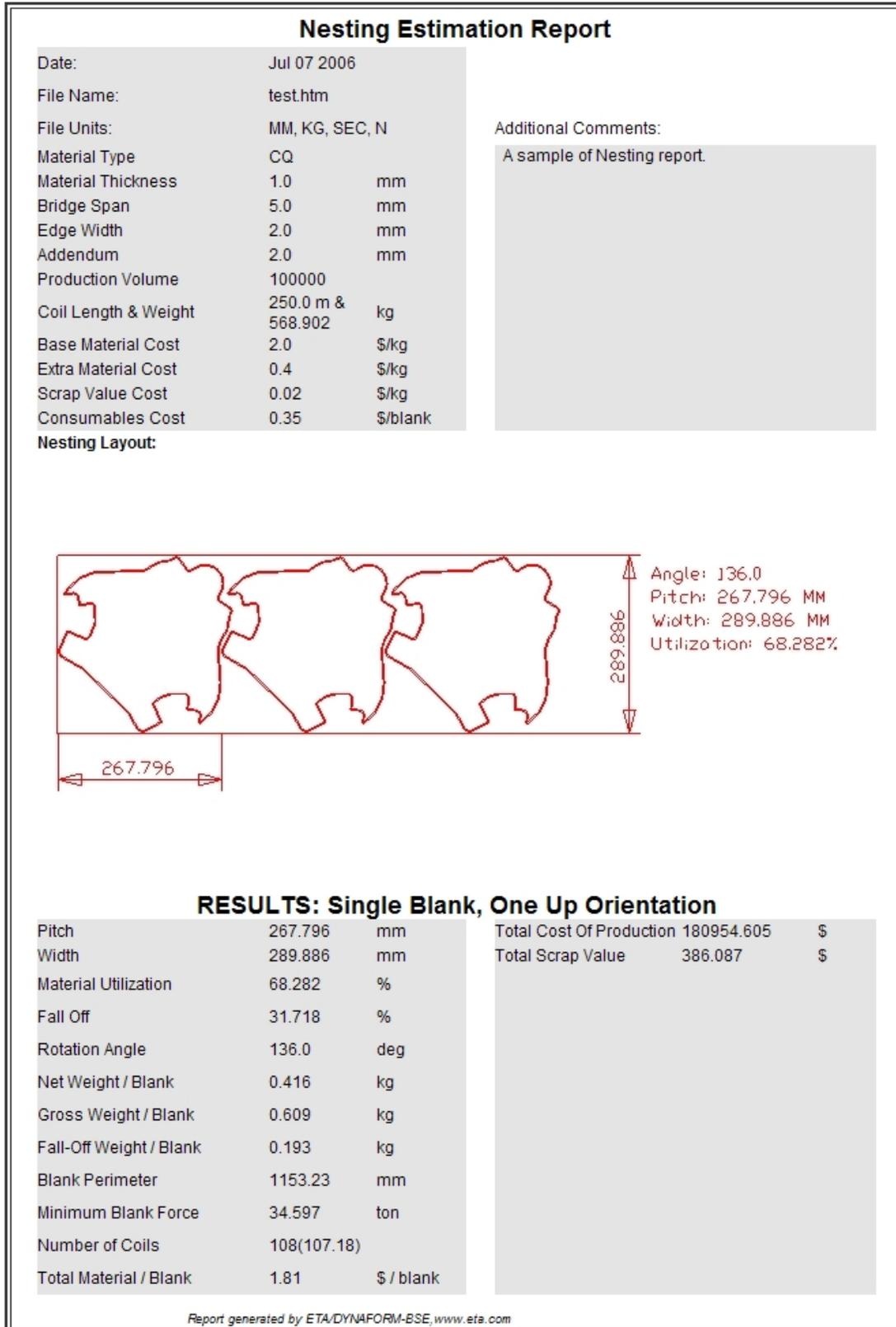


Figure 7.4.24 A typical nesting report

### 7.4.6.2 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 those described in the One-up Nesting. Refer to Section 7.4.6.1 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 adjust blank layout according to desired X- or Y-axis translations, and/or Z-axis rotations. Click on the Position button to open the Blank Position dialog box illustrated in Figure 7.4.25.

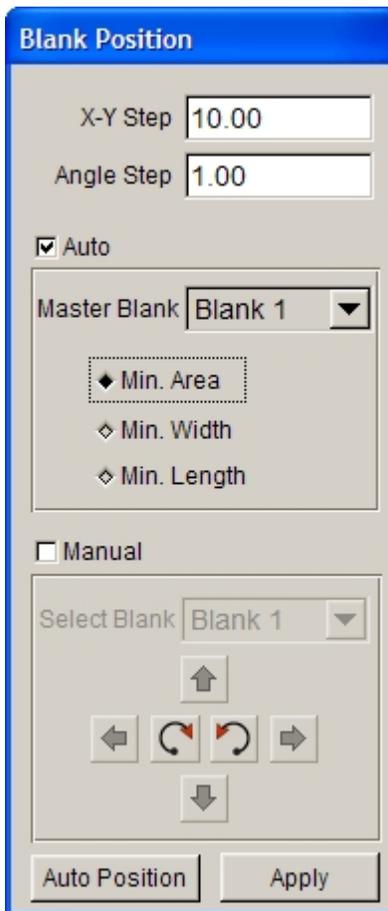


Figure 7.4.25 Auto Position dialog box

#### X-Y STEP

This function allows user to set 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, the blank is moved along the X/Y direction by the incremental value each time the translation button is hit. 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 for blank rotation. If the user adjusts the position of the blank through Manual method, the blank is rotated by the incremental angle each time the rotation button is hit. 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 optimized algorithm listed below.

- **Min. Area**  
Position blanks to minimize the surface area of the strip.
- **Min. Width**  
Position blanks to minimize the width of the strip.
- **Min. Length**  
Position blanks to minimize the length of progression.

### MANUAL

This function enables the user to manually position the blanks through translation and/or rotation.

### SELECT BLANK

This option enables the user to choose desired blank for positioning operation if more than one blank are available in nesting. The selected blank is highlighted in the display area.

### TRANSFORMATION



Move selected blank outline in positive X-axis direction according to incremental distance



Move selected blank outline in negative X-axis direction according to incremental distance



Move selected blank outline in positive Y-axis direction according to incremental distance



Move selected blank outline in negative Y-axis direction according to incremental distance

### ROTATION



Rotate selected blank outline clockwise by incremental angle.



Rotate selected blank outline counter-clockwise by incremental angle.

**AUTO POSITION**-- Enable the user to automatically obtain the positioned result.

**APPLY**--The program accepts the positioning result and exit the position dialog box.

### 7.4.6.3 TWO-PAIR NESTING



This function enables the user to generate nesting layout with two identical blanks with opposing orientation. The setting of Setup and Constraint in this function is similar to those described in the Two-Up Nesting. Refer to Sections 7.4.6.1 and 7.4.6.2 for description about all related parameters.

### 7.4.6.4 MIRROR NESTING



This function enables the user to generate nesting layout by 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 7.4.6.1. and 7.4.6.2 for description about all related parameters.

### 7.4.6.5 MULTI BLANKS NESTING



This function enables the user to generate nesting layout using more than one blank outline. When defining the blank outline, the user should click OK button or the middle button to accept each profile selection after selecting other blank outline. When all the blank outlines are selected, the user can click OK button or the middle button to exit the SELECT LINE dialog box.

The image shows a software dialog box titled "Blank Position". It contains several input fields and checkboxes. At the top, "X-Y Step" is set to 10.00 and "Angle Step" is set to 15.00. Below these, "Master Blank" is a dropdown menu currently showing "Blank 1". There are two main sections: "Auto" and "Manual". The "Auto" section is active, indicated by a checked checkbox. It contains two sub-options: "Rotate Master Blank" (unchecked) and "Rotate Slave Blank" (checked). The "Manual" section is inactive, indicated by an unchecked checkbox. It contains a "Select Blank" dropdown menu also showing "Blank 1", and a set of navigation buttons: a central "up" arrow, "left", "right", "counter-clockwise rotation", and "clockwise rotation" arrows, and a "down" arrow. At the bottom of the dialog, there is a "Result" section with three fields: "Pitch" (779.16), "Width" (2997.80), and "Utilization" (82.42%). At the very bottom are two buttons: "Auto Position" and "Apply".

Figure 7.4.26 Blank Position dialog box

**MASTER BLANK**

This option enables the user to select the master blank. It is active when the Multi-blanks nesting type is chosen.

*NOTE: Since the program uses the defined X-Y and Angle steps to calculate nesting layout in auto nesting, the user may use the default parameters to avoid long calculation time. These parameters can be reduced gradually to obtain more accurate nesting result.*

**AUTO**

This function enables the user to automatically position the blanks through the selected optimized scheme.

**ROTATE MASTER BLANK**

Allow the user to control whether the master blank is rotated during auto nesting.

**ROTATE SLAVE BLANK**

Allow the user to control whether the slave blanks are rotated during auto nesting. This option is activated for multiple blank nesting.

## Chapter 8

# SETUP

The functions provided in the SETUP user graphic interface (GUI) help the user to quickly set up a model for forming simulation. The SETUP menus provide two types of setup scheme: 1. The QUICK SETUP consists of several standard forming operations as shown in Figure 8.1; 2. The 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

#### ROTARY DRAW BENDING

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	Automated interface limits	Provides built-in templates

duplicate any tooling configuration: pads, multiple tools, etc.	flexibility.	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 13.1).

**EXIT** allows the user to exit the QS interface.

A typical QS GUI is illustrated in Figure 8.1.1.

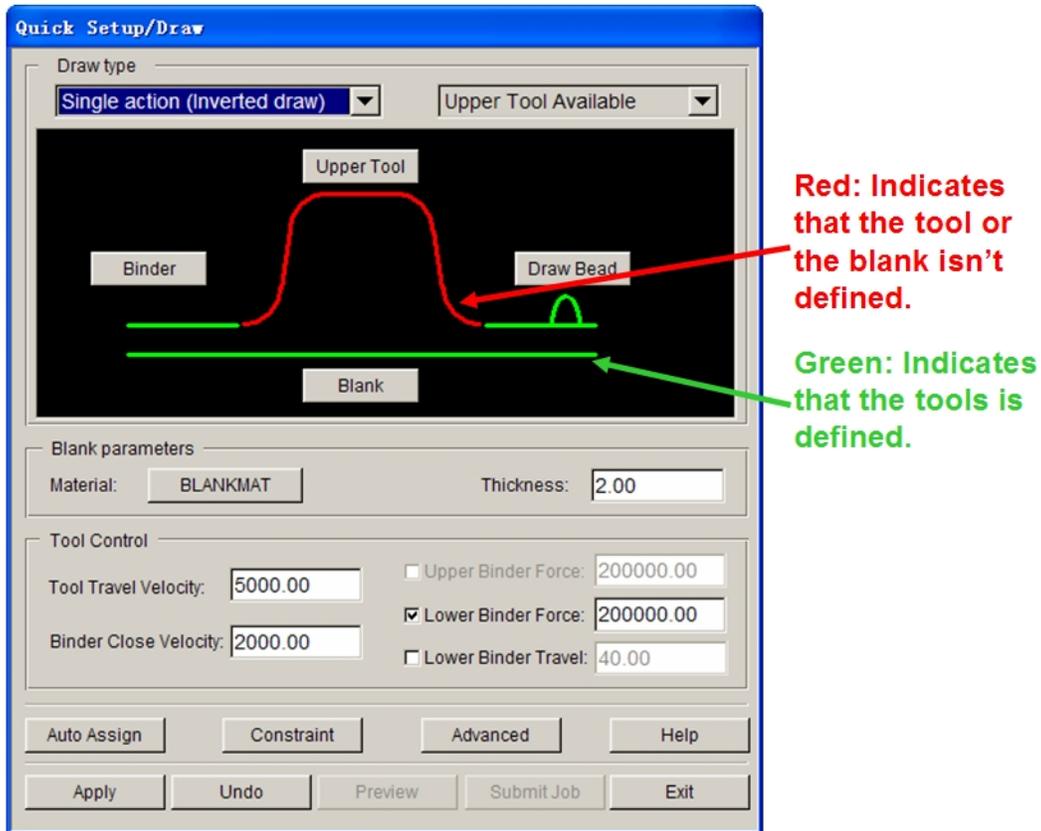


Figure 8.1.1 A typical Quick Setup GUI

### 8.1.1 GRAVITY LOADING

The QUICK 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.

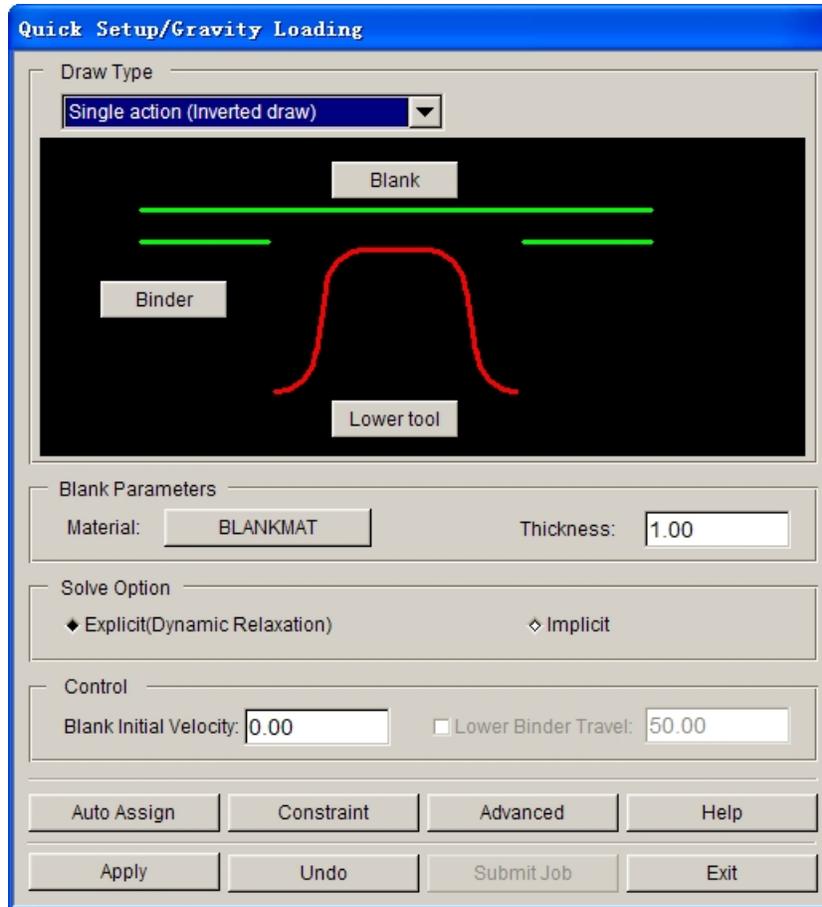


Figure 8.1.2 A typical gravity loading GUI for single action draw operation

### **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 better solution, it is difficult to achieve converged non-linear equilibrium iterations. 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.

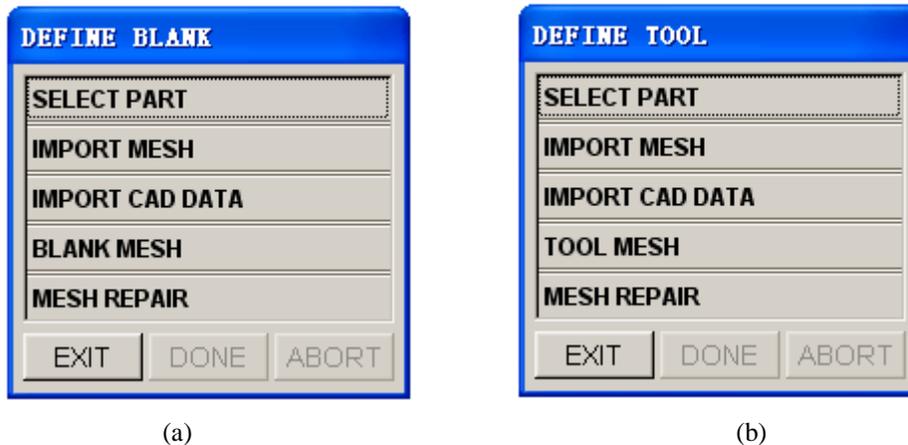


Figure 8.1.3 The Tool Definition dialog box (a) Define Blank (b) Define Tool

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 is shown in Figure 8.1.5.

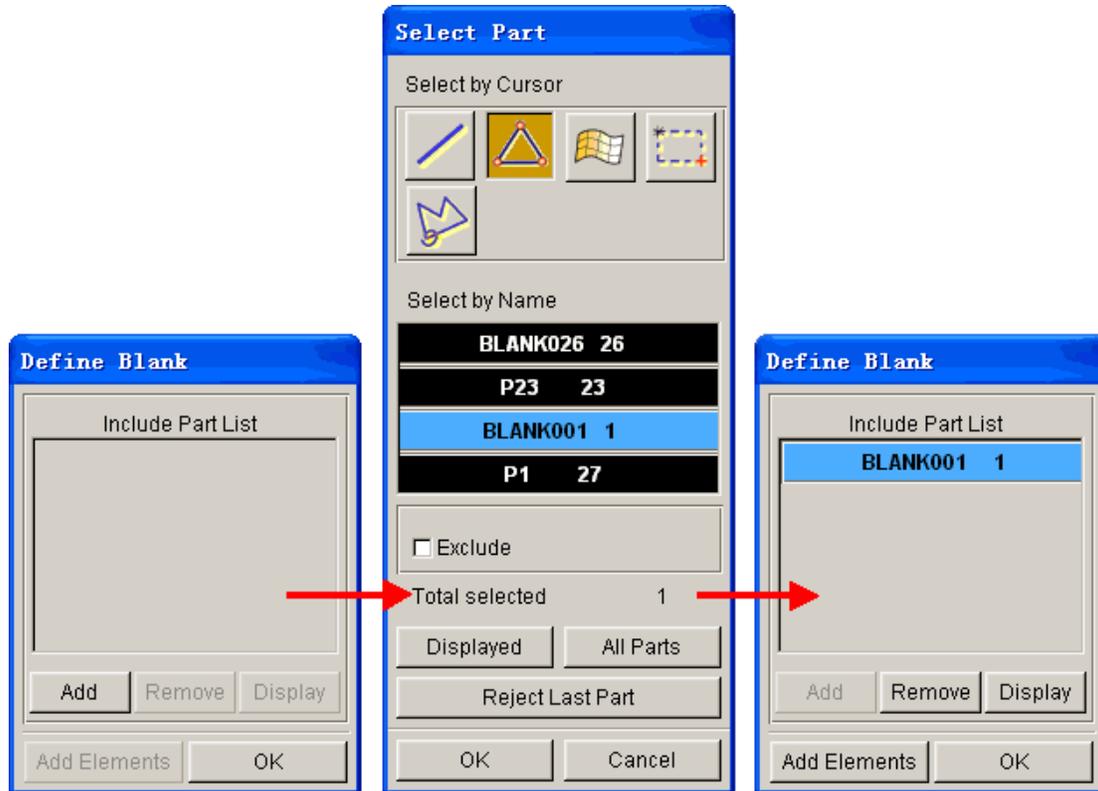


Figure 8.1.4 A Schematic Illustration Showing the SELECT PART Operation for Blank Definition in QS

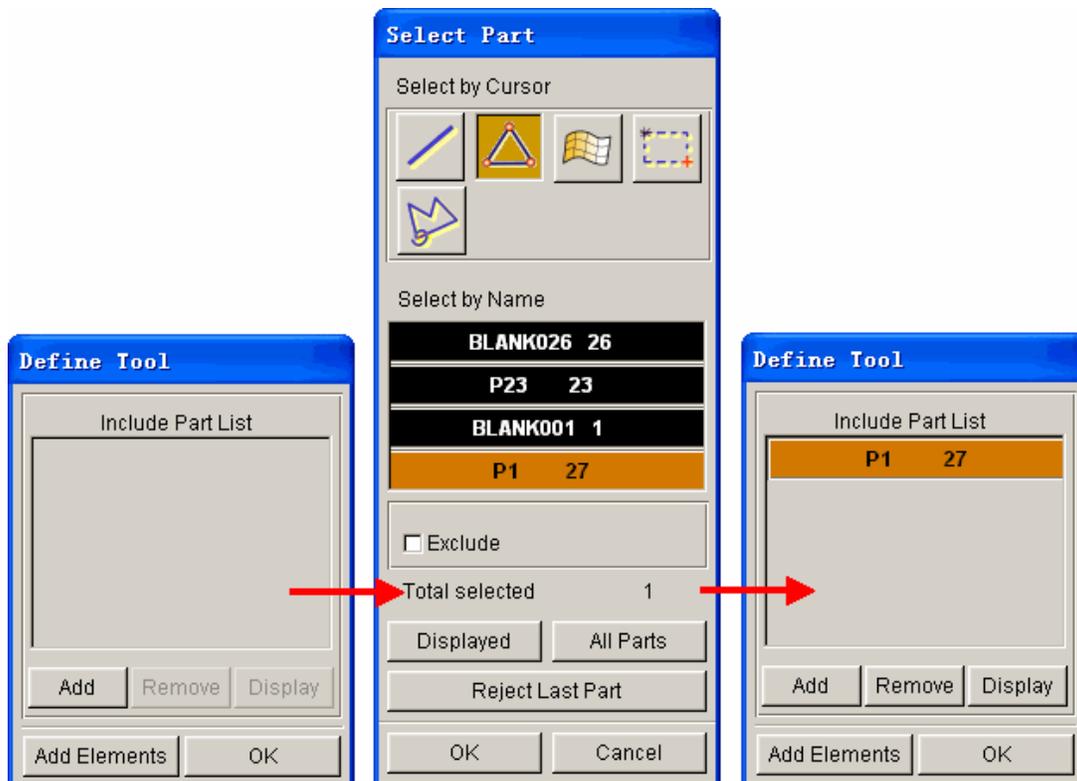


Figure 8.1.5 A Schematic Illustration Showing the SELECT PART Operation to Define the Lower Tool in QS

### 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 9.9. The NONE button is disabled if no blank is defined in the tool GUI.

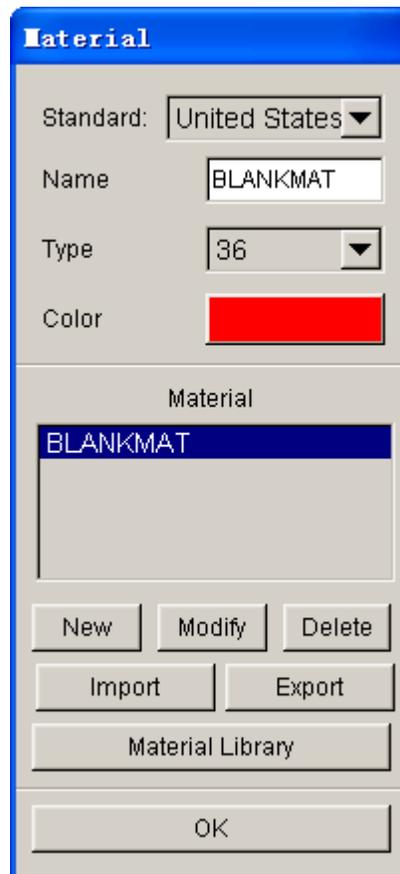


Figure 8.1.6 The Material dialog box

### 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 button to auto-position the tools and blank.
8. Click SUBMIT JOB to submit the analysis. Refer to Section 13.1 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.

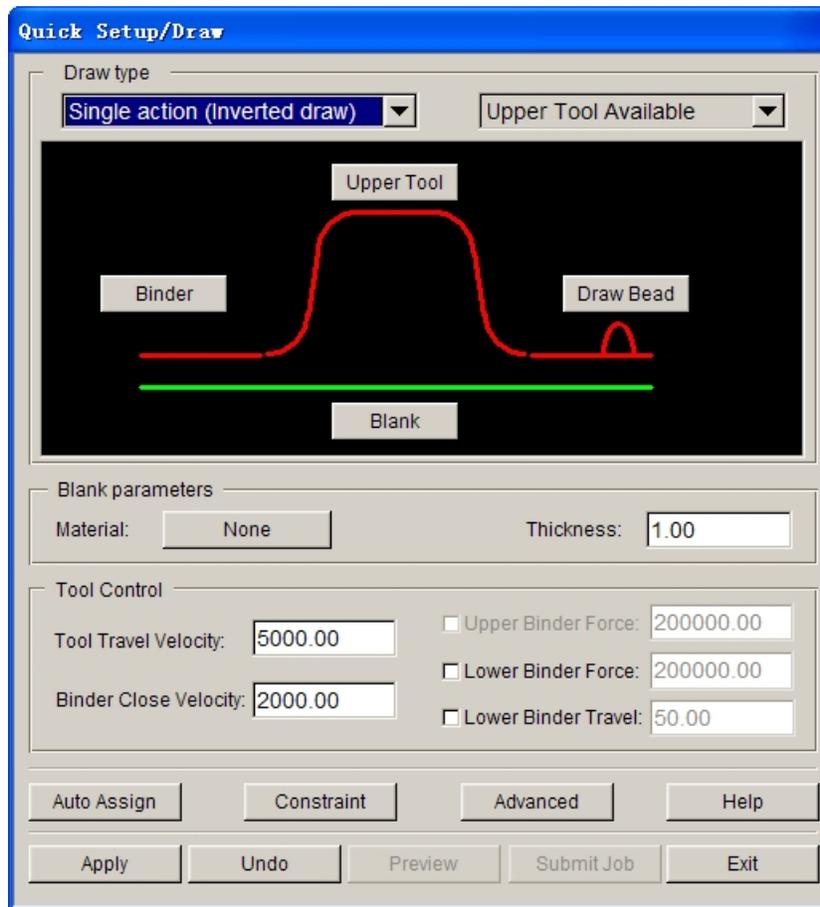


Figure 8.1.7 A typical draw die GUI for single action draw operation

#### 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.

#### 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.

## 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.

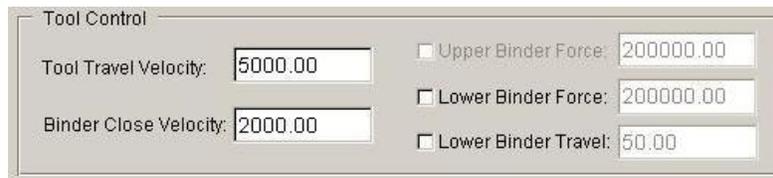


Figure 8.1.8 The Tool Control interface

Depending on the draw type, the tool velocity and force are applied according to the following control scheme:

<b>CRASH FORM</b>	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/s** 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:

- a. Provide punch travel velocity (default is 5000 mm/s).
- b. Provide binder velocity (default is 2000 mm/s).
- c. 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 die gap, load curve, output frames and drawbead control. The dialog box is shown in Figure 8.1.9.

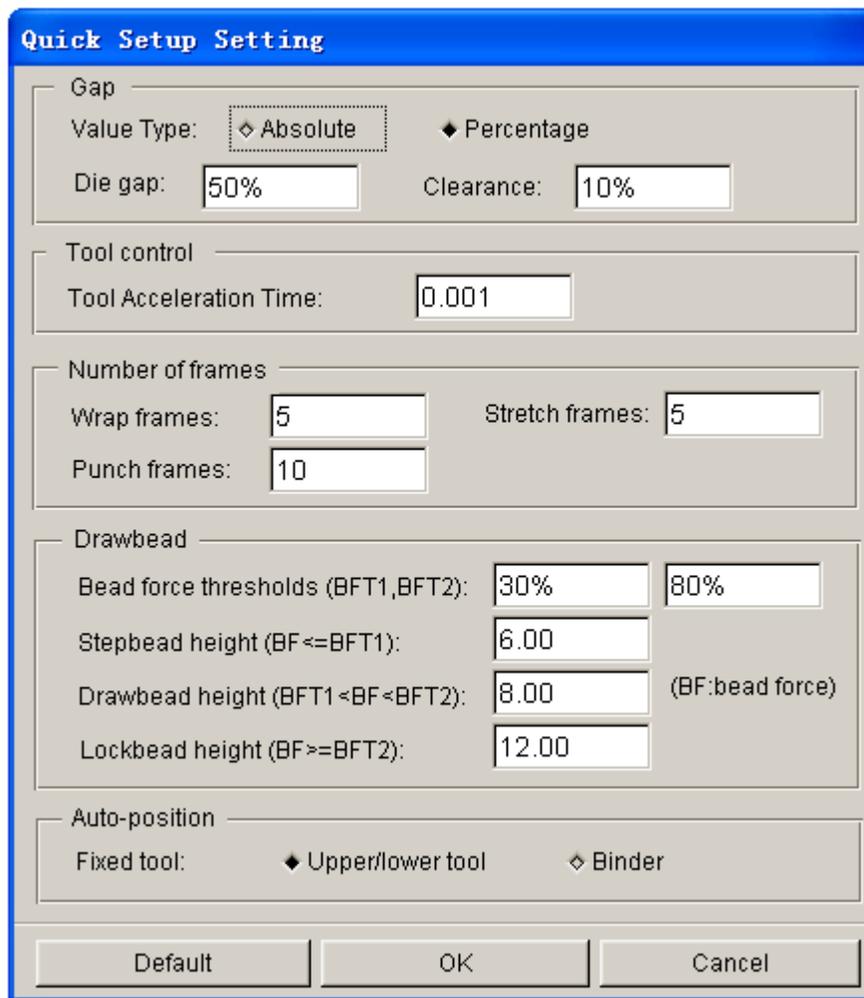


Figure 8.1.9 Quick Setup Setting dialog box

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 9.4.

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.
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 13.1).

#### **8.1.3 SPRINGBACK**

The QS/SPRINGBACK GUI (as shown in Figure 8.1.10) allows the user to easily setup a springback analysis.

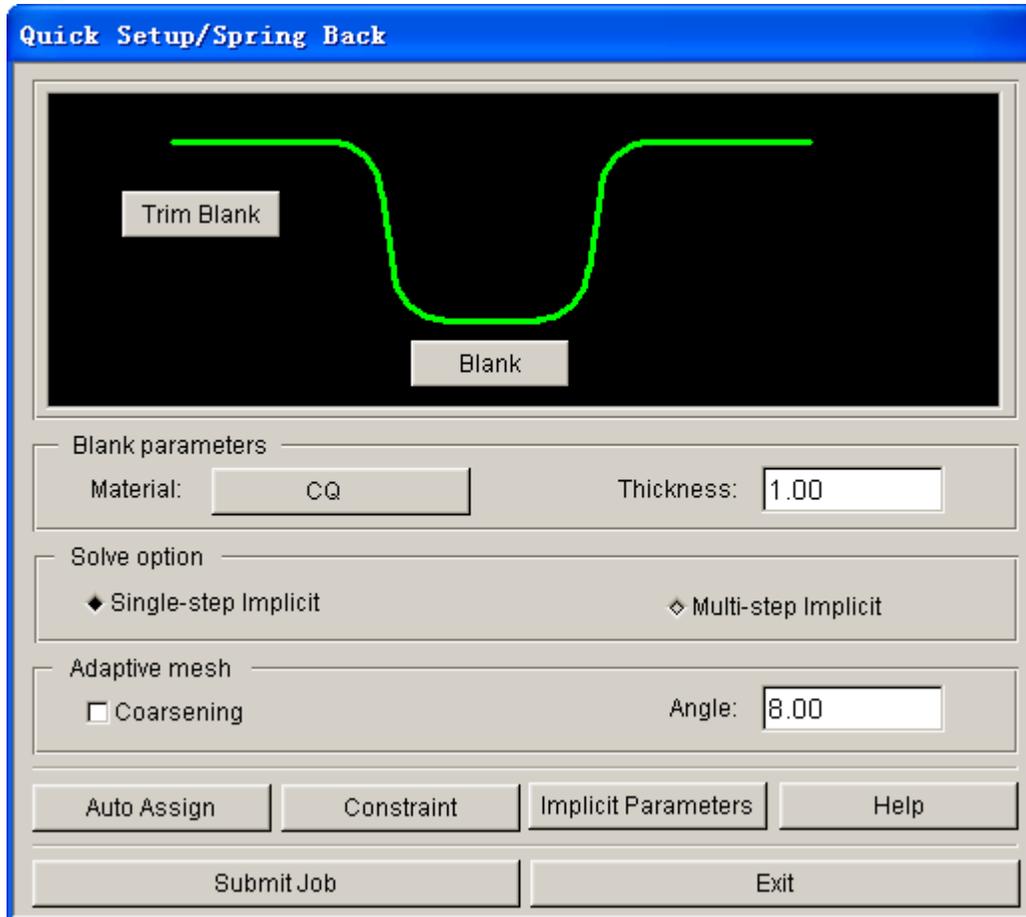


Figure 8.1.10 The QS Springback GUI

### 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 a 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 9.8.5 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 analysis. 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).

### 8.1.4 ROTARY DRAW BENDING

Rotary Draw Bending is a newly developed Quick Setup module in eta/DYNAFORM for performing rotary draw bending of tubular products. This is a process oriented module with user-friendly GUI. Furthermore, it provides functions to enable the user to easily generate tube and toolings of rotary draw bending models.

The Rotary Draw Bending interface is illustrated in Figure 8.1.12. The user can obtain the complete analysis model using the default parameters. The program requires the user to confirm the thickness of tube blank and define the tube blank first and then define tools (refer to the location and size of tube blank).

Note: Rotary bending simulation only supports global coordinate system and the rotation axis is +Z axis.

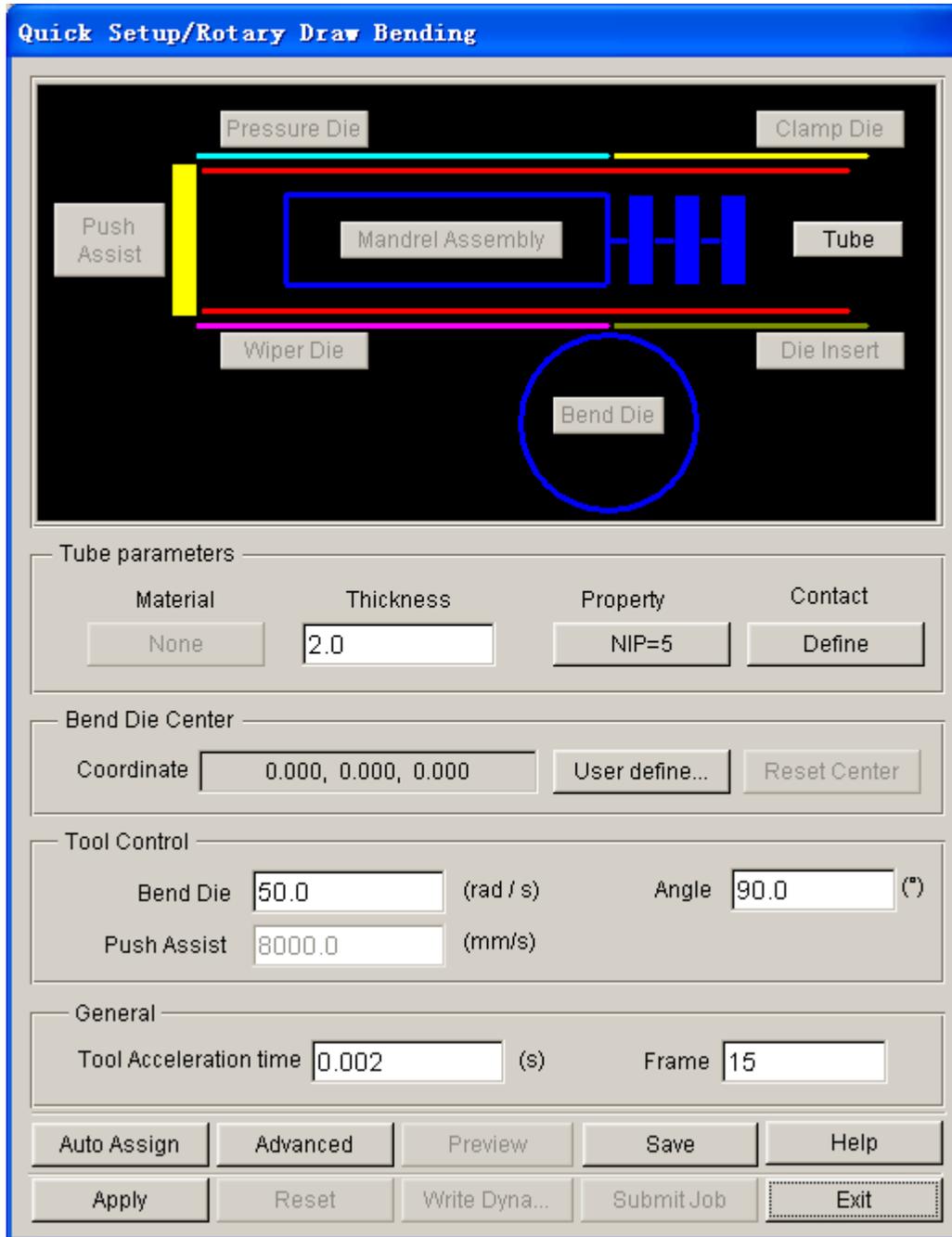


Figure 8.1.12 Quick Setup/ Rotary Draw Bending GUI

The following options are included in the Rotary Draw Bending interface:

**Auto Assign:** Assign the part with standard name as the corresponding tool, e.g. TUBE, BEND, CLAMP, INSERT, PRESS, WIPER, PUSH, MANDREL, BALL and LINK.

**Advanced:** Modify the setting of default parameters of rotary draw bending.

**Preview:** Check the motion of tool by animation.

**Save:** Save the database without exiting the Rotary Draw Bending interface.

**Help:** Provide useful information about simulation process.

**Apply:** Activate the program to define tool motion curve, etc.

**Reset:** Restore to the original setting.

**Write Dyna:** Generate input deck.

**Submit Job:** Submit input deck to solver for analysis.

**Exit:** Exit the Rotary Draw Bending interface.

### 8.1.4.1 TOOL DEFINITION GUI

This function allows the user to assign or create tools and tube blank mesh for rotary draw bending. After all the tools are defined, the outline of tools is displayed in green.

#### TUBE

The Define Tube dialog box is illustrated in Figure 8.1.13.

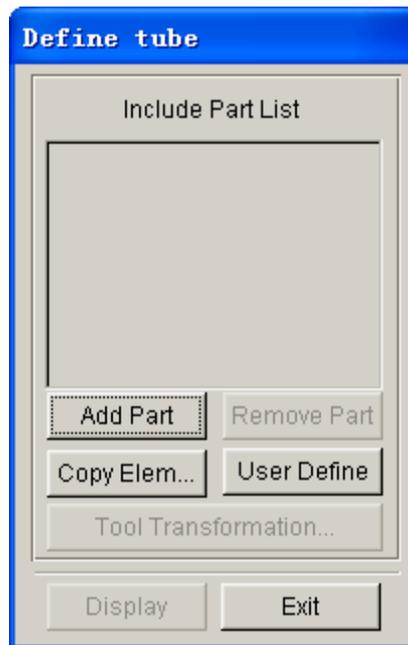


Figure 8.1.13 Define Tube dialog box

#### ADD PART...

Select a part from the Select Part dialog as tube. The selected part will be displayed in the Part List.

#### REMOVE PART

Remove the selected part from the Part List.

**COPY ELEM...**

Generate elements by duplicating the existing elements.

**USER DEFINE**

Generate tube geometry. The dialog box is illustrated in Figure 8.1.14.

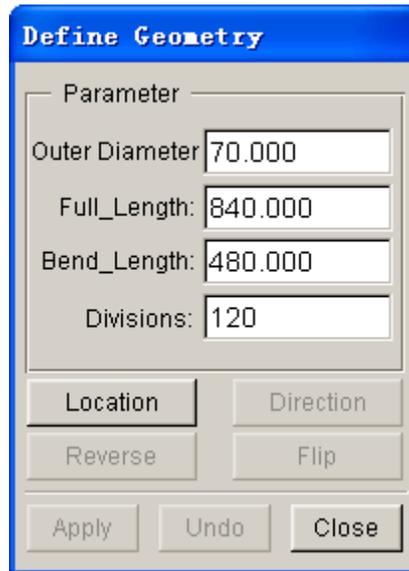


Figure 8.1.14 Define Geometry dialog box

<b>OUTER DIAMETER</b>	Outer diameter of tube.
<b>FULL_LENGTH</b>	Length of tube.
<b>BEND_LENGTH</b>	Bend length of tube.
<b>DIVISIONS</b>	Number of mesh divisions along tube parameter.
<b>LOCATION</b>	Define the starting location of tube blank bending, which also locates on the tube blank axis.
<b>DIRECTION</b>	Define the direction of tube centerline.
<b>REVERSE</b>	Reverse direction of bend length after generation of mesh.
<b>FLIP</b>	Reverse direction of bend length prior to generation of mesh.

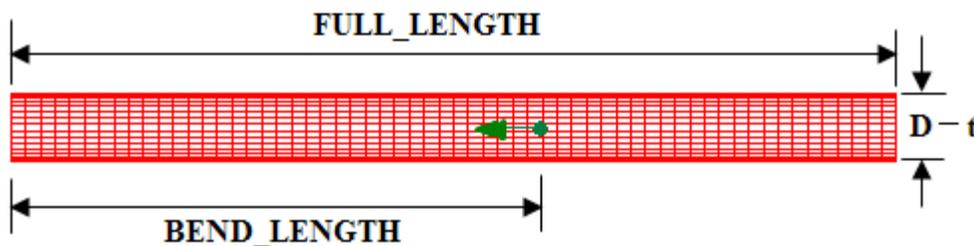


Figure 8.1.15 Tube Blank Geometry

**APPLY:** Generate mesh according to the specified parameters.

**UNDO:** Cancel the operation of generating mesh.

**CLOSE:** Exit the current dialog box.

**TOOL TRANSFORMATION**

This function allows the user to translate and/or rotate the defined tube.

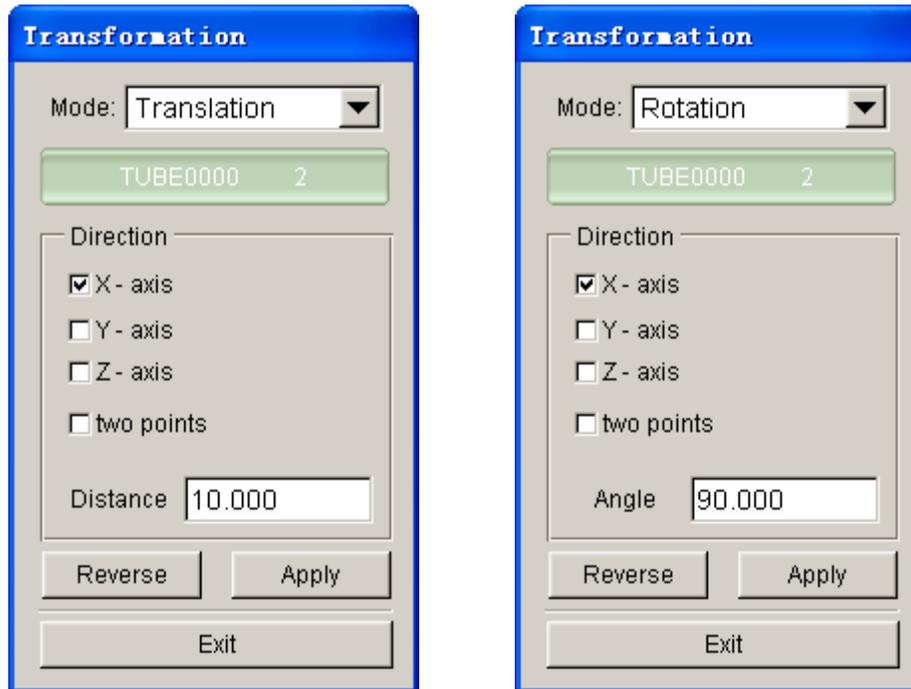


Figure 8.1.16 Transformation dialog box

- X-AXIS** Translate or rotate along (about) X axis.
- Y-AXIS** Translate or rotate along (about) Y axis.
- Z-AXIS** Translate or rotate along (about) Z axis.
- TWO POINTS** Translate or rotate along (about) the vector direction defined by two points.
- DISTANCE** Translation distance.
- ANGLE** Rotation angle.

**REVERSE:** Translate or rotate along (about) the reverse direction.

**APPLY:** Execute translation or rotation.

**EXIT:** Exit the current dialog box.

**DISPLAY**

This function allows the user to only display the current defined object on the screen. Click on this button again to display the other objects.

**EXIT**

Exit the current dialog box.

**BEND DIE**

The definition of BEND DIE is similar to definition of tube. As shown in Figure 8.1.17, the user can generate geometry of BEND DIE by defining the bend die diameter (D) and radius (R) illustrated in Figure 8.1.18.

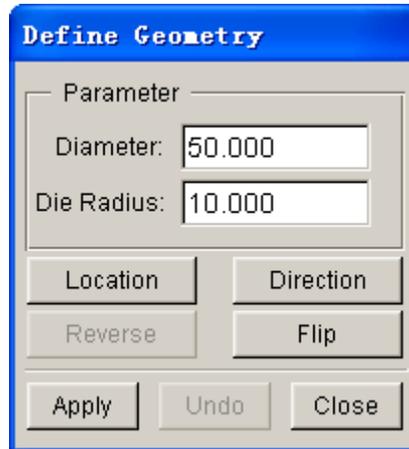


Figure 8.1.17 Define Geometry

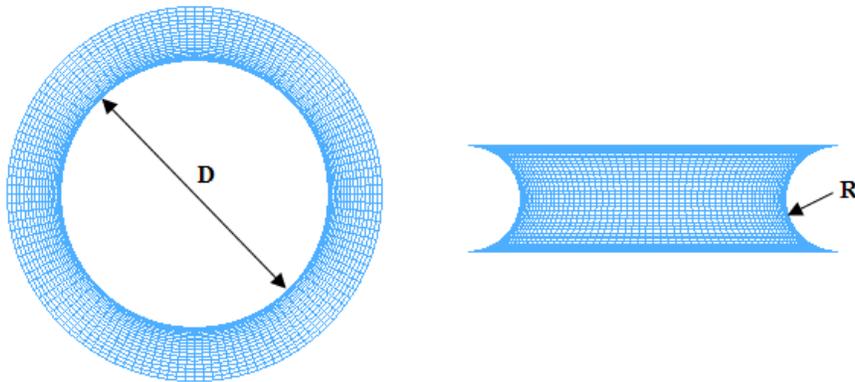


Figure 8.1.18 Parameters defining geometry of Bend Die

**MANDREL ASSEMBLY**

The mandrel Assembly Definition dialog box is illustrated in Figure 8.1.19. A typical mandrel assembly consists of mandrel shaft, balls and links between balls. The user can generate the mandrel assembly by entering the related dimensions. After defining the mandrel shaft and balls, links are automatically generated and connects the mandrel shaft and balls. The user can click on the Advanced... button to define desired contact type and parameters, as shown in Figure 8.1.20.

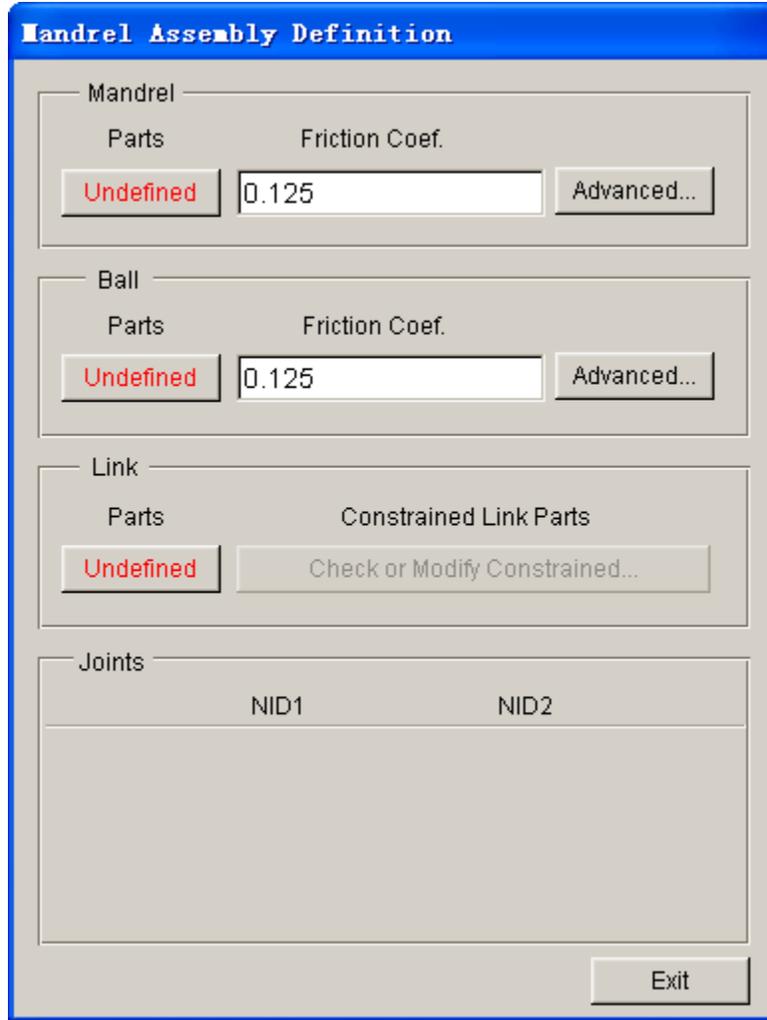


Figure 8.1.19 Mandrel Assembly Definition dialog box

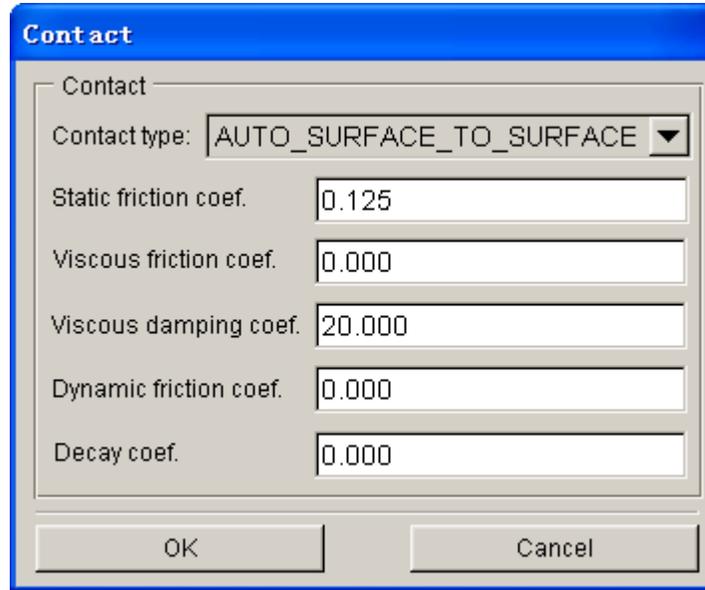


Figure 8.1.20 Contact dialog box

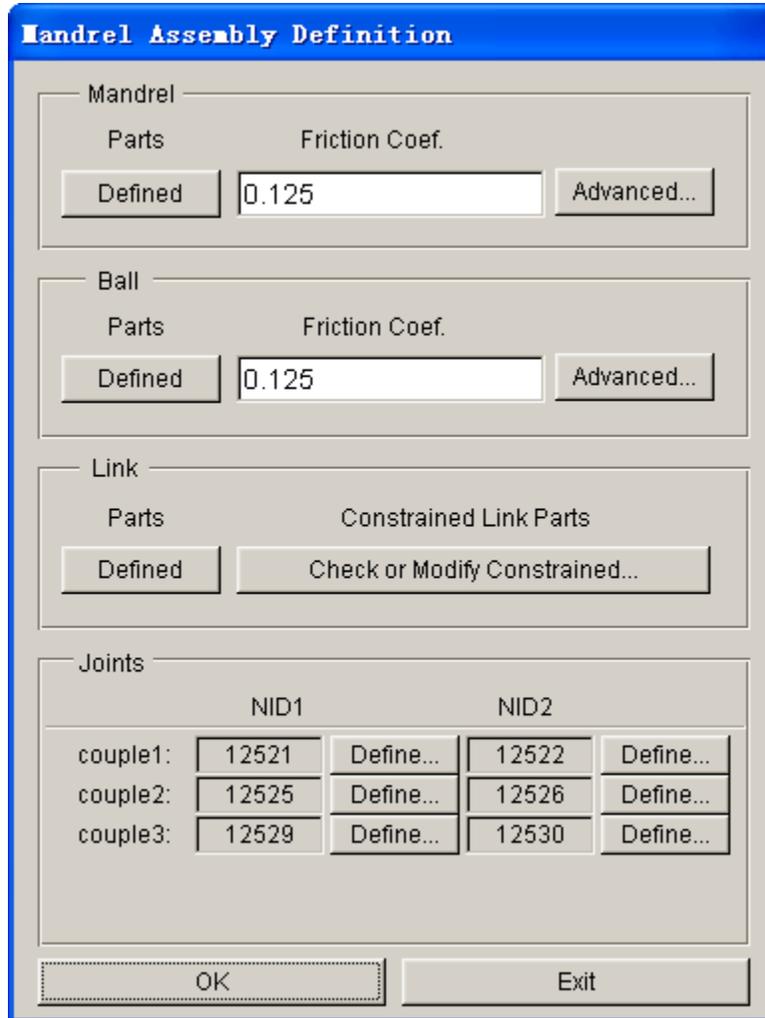


Figure 8.1.21 Defined Mandrel Assembly dialog box

After the mandrel assembly is defined, an interface as illustrated in Figure 8.1.21 is displayed. In the Link field, the user can click on Check or Modify Constrained...button to check of the definition of constraints. In the Joints field, the user can click on Define... button to modify the NID of connected nodes. The coupled nodes indicate the connection between links.

As shown in Figure 8.1.22, the tool font color and background color are displayed with the color of tool mesh. The user can click on the buttons in the dialog window to edit the relationship of constraint.

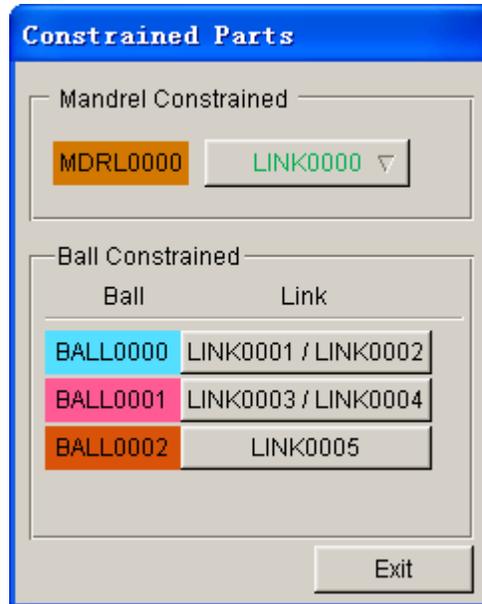


Figure 8.1.22 Constrained Parts dialog box

**CLAMP DIE**

The Define Geometry dialog box for CLAMP DIE is illustrated in Figure 8.1.23.

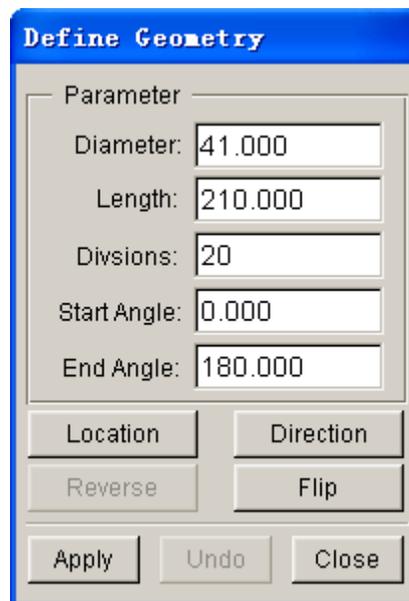


Figure 8.1.23 Define Geometry

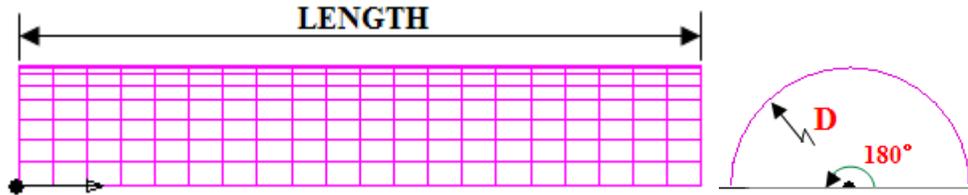


Figure 8.1.24 Parameters defining geometry of Clamp Die

The definition of other tools are similar to the one described above. Usually, the user adopts the default base point without assigning one for the user-defined tool. The user must check the correctness of direction when generating tool mesh.

#### 8.1.4.2 TUBE PARAMETERS

Tube Parameters includes MATERIAL, THICKNESS, PROPERTY and CONTACT types and parameters.

##### MATERIAL

Refer to Section 10.9 for description about tube material definition.

##### THICKNESS

The user must define the tube wall thickness prior to defining the tube geometry. By default, the thickness is set as 2.0 (mm).

##### PROPERTY

Property includes selection of element formulation, number of integration points and the setting of shear correction factor.

#### 8.1.4.3 BEND DIE CENTER

The center of bend die defines the center of rotation for rotary draw bending. If the bend die is created by the program, the center of rotation is identical to the geometry center. After the bend die is defined, the coordinate of rotation center is displayed. The user can click on USER DEFINE... button to define new rotation center. The user may click on the RESET CENTER button to restore the original rotation center.

#### 8.1.4.4 TOOL CONTROL

Tool control field enables the user to define the velocity and rotation angle of bend die, as well as the motion of push assist. The motion of push assist can be defined using one of the 3 options: AUTOMATIC, VELOCITY and DISPLCMNT. If AUTOMATIC option is chosen, the velocity curve is automatically assigned according to the pre-defined velocity. For VELOCITY and DISPLCEMNT option, click on DEFINE... button to define the velocity or displacement curve. Refer to Section 8.2.4.2 for description about curve definition.

#### 8.1.4.5 GENERAL

This function allows the user to set the Tool Acceleration Time and output Frame number. The default acceleration time is set as 0.002(s), while the default output frame number is set as 15.

## 8.2 AUTO SETUP

The Auto Setup module provides a complete set of solutions for vast range of sheet metal forming. It supports most basic functions of traditional setup and new capabilities such as hydroforming, superplastic forming, laminated sheet and thermal forming. In addition, simple multi-stage forming simulations and other new features such as user defined stamping direction and tailor welded blank are also supported by Auto Setup. In Auto Setup module, the user simply defines the closing between two pieces of tools in each process, while the program computes simulation time for each process. It greatly helps the user to save time in model setup and reduces risk of making mistakes.

A typical Auto Setup interface is illustrated in Figure 8.2.1. The Auto Setup interface consists of menu bar, stage setup bar and setup pages, which includes General, Blank, Boundary Condition, Tools, Process and Control. The Blank page is shared by all stages.

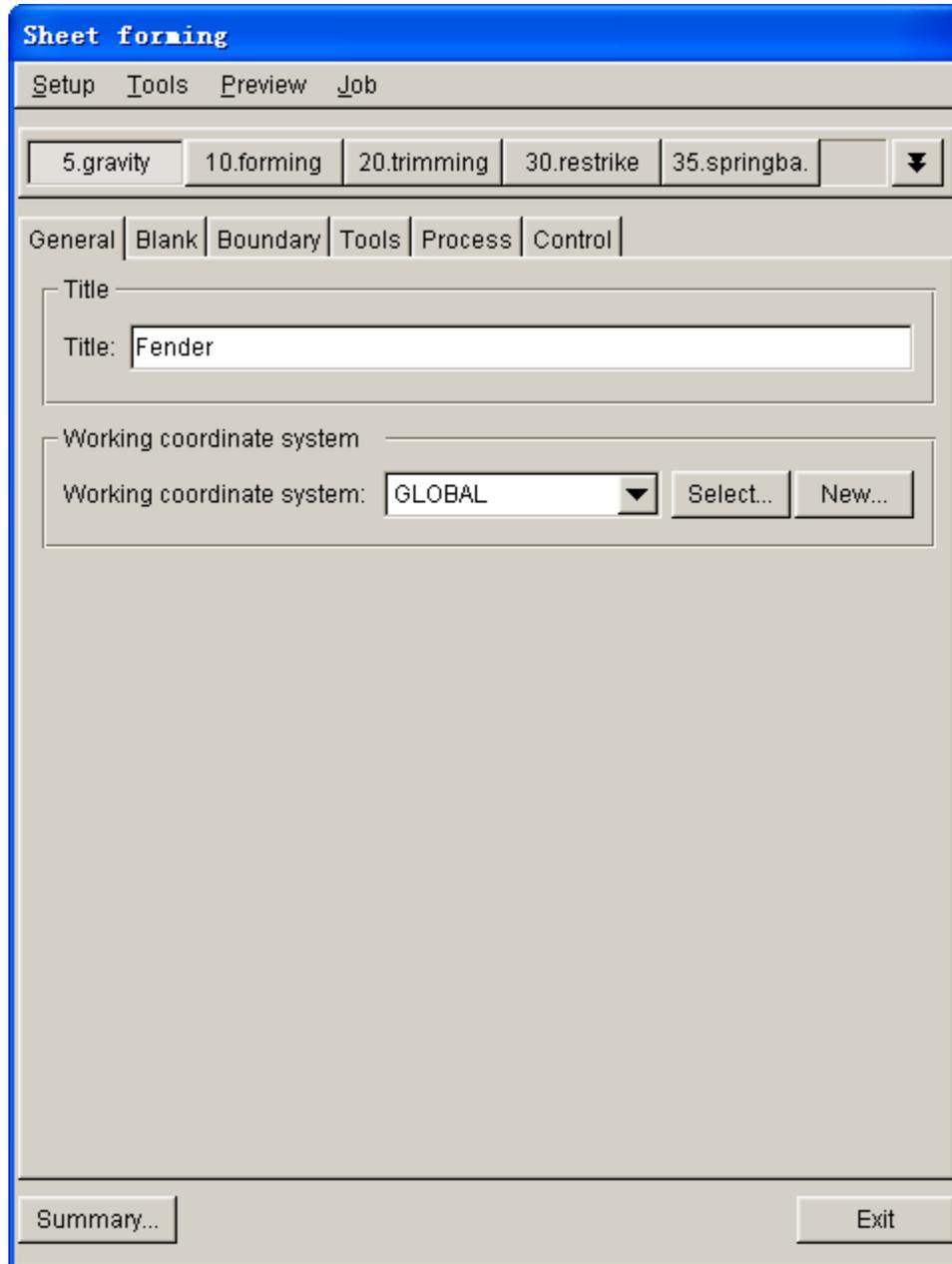


Figure 8.2.1 A typical Auto Setup Interface

## 8.2.1 NEW SIMULATION

This dialog enables the user to define a new simulation prior to displaying the Auto Setup interface. First, select AUTOSSETUP from the SETUP menu. If Auto Setup is defined in the existing database, the program will automatically display the Auto Setup interface. Otherwise, the program will automatically display the New Simulation window illustrated in Figure 8.2.2. The user must define the setting in New Simulation window.

**NOTE:** The user is only allowed to set one Auto Setup project in each database.

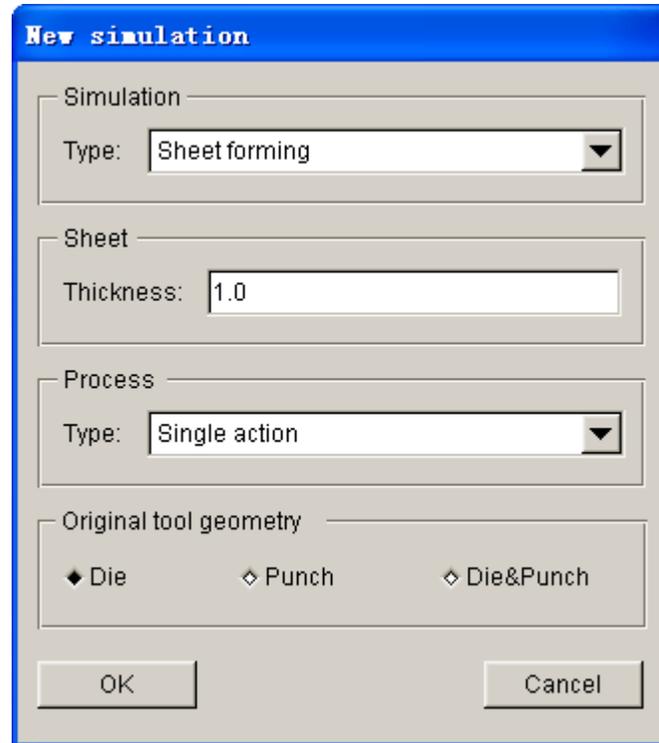


Figure 8.2.2 New Simulation window

### 8.2.1.1 SIMULATION TYPE

There are two simulation types supported in the Auto Setup. One is sheet forming, which includes sheet forming and sheet hydroforming. The other one is tubular hydroforming.

### 8.2.1.2 BLANK THICKNESS

At the beginning of each new setup, the user shall key in the desired blank thickness. This value will be automatically set in Blank page. The user may modify the blank thickness in Blank page. Default blank thickness is 1.0 (mm).

### 8.2.1.3 PROCESS

As shown in Figure 8.2.3, the Auto Setup includes 6 ordinary process templates. The Trimming and Superplastic templates are added in eta/DYNAFORM 5.6.



Figure 8.2.3 Process

**GRAVITY:** gravity loading simulation

**CRASH FORM:** forming process using upper and lower tools.

**SINGLE ACTION:** air draw or inverted toggle draw process using die, punch and binder.

**DOUBLE ACTION:** toggle draw process using die, punch, upper and lower binders.

**TRIPLE ACTION:** forming process using multiple pieces of tooling.

**SPRINGBACK:** springback analysis.

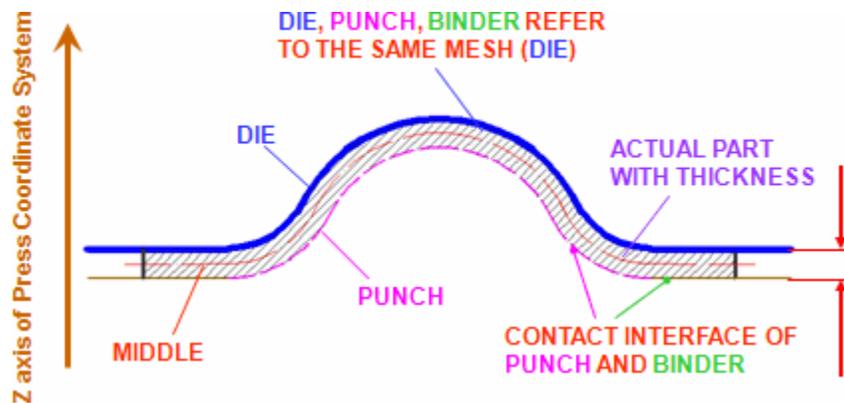
**TRIMMING:** trimming simulation.

**SUPERPLASTIC FORMING:** superplastic forming simulation

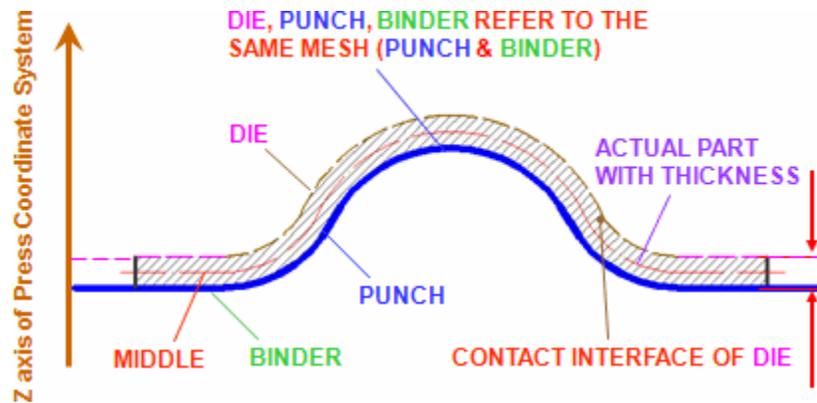
### 8.2.1.4 ORIGINAL TOOL GEOMETRY

In this field, the user chooses to perform stamping simulation using either CONTACT OFFSET algorithm or physical tool geometry. The user may turn on the radio button of either DIE (upper die) or PUNCH (lower die) option if he/she selected the CONTACT OFFSET algorithm. As illustrated in Figure 8.2.4a, the contact interfaces of both punch and binder are offset by  $(2 \times \text{blank thickness}) + \text{gap}$  if the DIE option is chosen. Similarly, the contact interfaces of die is offset by  $(2 \times \text{blank thickness}) + \text{gap}$  if the PUNCH option is chosen, as illustrated in Figure 8.2.4b.

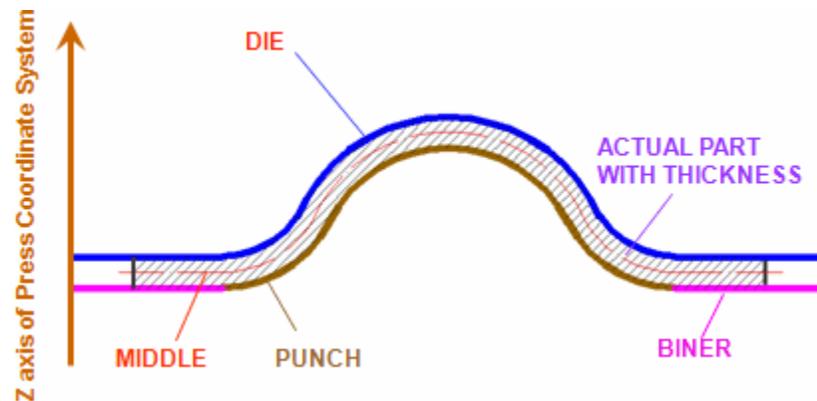
*Note: The geometry of punch and binder are duplicated from the die geometry if CONTACT OFFSET algorithm is adopted in stamping simulation.*



a) Using DIE geometry as reference for contact offset



b) Using PUNCH geometry as reference for contact offset



c) Using both DIE &amp; PUNCH physical geometry

Figure 8.2.4 Blank Reference Surface

If the user chooses to perform the stamping simulation using physical tool geometry, he/she should turn on the radio button of DIE&PUNCH illustrated in Figure 8.2.2. It is recommended to perform stamping simulation using physical tool geometry if springback is primary concern.

## 8.2.2 AUTO SETUP MENU

In the Auto Setup interface, the menu bar includes four menus illustrated in Figure 8.2.5. The functions provided in each menu are described in subsequent sections.



Figure 8.2.5 Auto Setup Menu Bar

### 8.2.2.1 SETUP

The functions listed in Setup menu includes New, Save, Clear and Exit.

**NEW** — Create a new simulation setup. After this option is selected, a dialog box is displayed to allow the user to perform new simulation setup.

**SAVE** — Save current setup information in existing database.

**CLEAR** — Delete all the setup information in AUTO SETUP. Prior to performing Clear operation, the user must confirm the operation using the Confirmation dialog box.

**EXIT** — Exit Auto Setup dialog box and return to eta/DYNAFORM main interface.

### 8.2.2.2 TOOLS

The Tools menu provides functions such as Trimming, Display, Positioning, Drawbead and POP Line.

**TRIMMING** — Trim the blank. Both blank and trimline must be defined prior to trimming operation. For detailed description of trimming operation, refer to Section 5.3.10.

**DISPLAY** — Manage the display of tools via Tool On/Off dialog window illustrated in Figure 8.2.6.



Figure 8.2.6 Tool On/Off

**POSITIONING** — This function is used to adjust the initial position of tools and blank in stamping simulation. This function supports auto and manual positioning. For detailed description of positioning function, refer to Section 8.2.4.4.

**DRAWBEADS** — This function allows the user to setup drawbeads. Detailed descriptions of drawbead are provided in Section 10.4.

**POP LINES** — This function allows the user to define the POP line of punch and print such information into the index file. Therefore, the user can read in the POP line information in Post-processor. For detailed description of POP line, refer to Section 14.1.5.

### 8.2.2.3 PREVIEW

This function enables the user to validate the motion of tools and review setup information. The

ANIMATION function allows the user to preview motion of each tool assigned with either velocity or displacement curve. The SUMMARY function allows the user to review the setup information.

**ANIMATION** — Animate the movement of tools. After selecting this function, the Animate dialog box as shown in Figure 8.2.7 is displayed. Detailed description of the animation control is provided in Section 10.11.

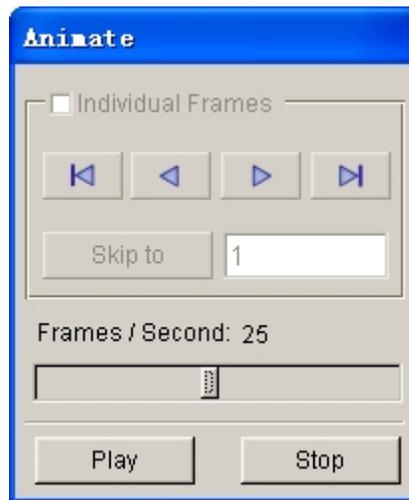


Figure 8.2.7 Animate dialog box

**SUMMARY** — This function lists information of the current setup, which includes stamping direction, tool, blank, numbers of process, process name and contact type, etc. A typical Summary output is illustrated in Figure 8.2.8.

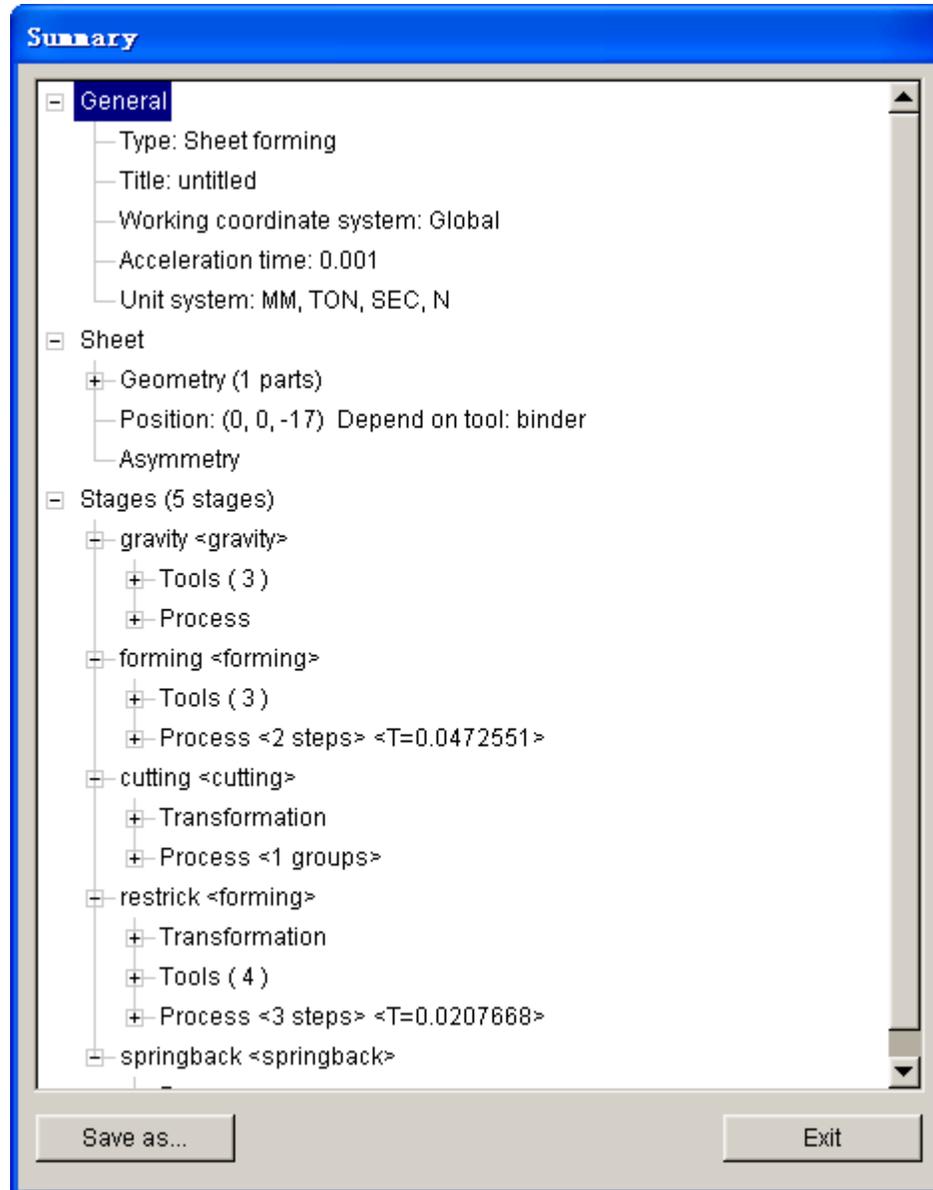


Figure 8.2.8 Summary output

### 8.2.2.4 JOB

The functions provided in this menu include LS-DYNA INPUT FILE, FULL RUN DYNA and JOB SUBMITTER.

**LS-DYNA INPUT FILE** — This function enables the user to output LS-DYNA input deck to the directory for the current database.

**FULL RUN DYNA** — This function enables the user to directly run a job on LS-DYNA Solver. The Submit Job dialog box illustrated in Figure 8.2.9 is displayed.

**NOTE:** The user may define the appropriate path for the LS-DYNA solvers from the *TOOLS/ANALYSIS SETUP* menu.

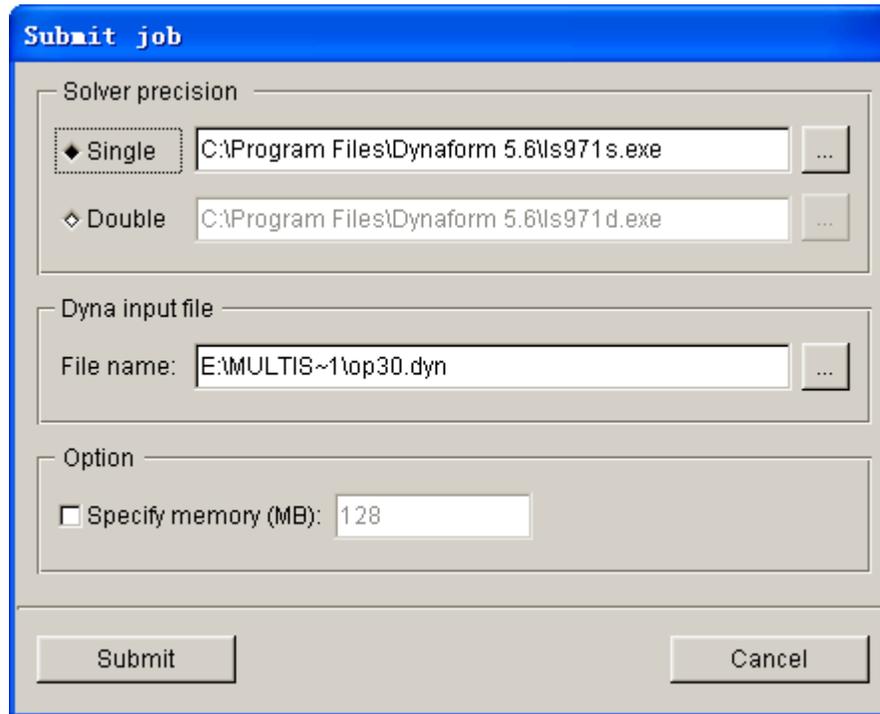


Figure 8.2.9 Submit Job dialog box

**Solver precision:** Enable the user to select either Single or Double Precision solver for running the current job. The user may click on  button to set the paths for Single and Double Precision solvers.

**Dyna input file:** Define the path and file name of the output DYNA input file. The default path is the current working directory. The user may click  button to change the output path and file name.

**Specify memory (MB):** Specify memory allocation for current job. The default value is 128MB. The user may specify different memory allocation according to the size of model.

**Submit:** Allow the user to submit the current job. Click on this button to start LS-DYNA solver for calculation.

**Cancel:** Exit Submit Job interface and return to Auto Setup interface.

**JOB SUBMITTER** — This function allows the user to submit and run the stamping simulation model. The Submit Job dialog box is illustrated in Figure 8.2.10.

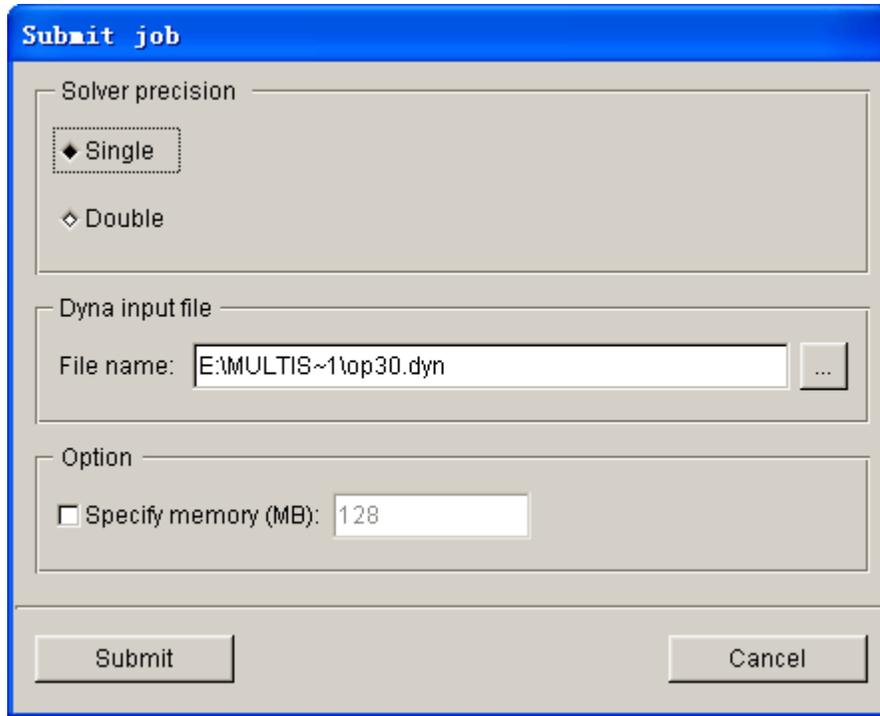


Figure 8.2.10 Submit Job dialog box

To submit a job, click the Submit button in Submit Job dialog window. The job is submitted to the Jobs Submitter window illustrated in Figure 8.2.11.

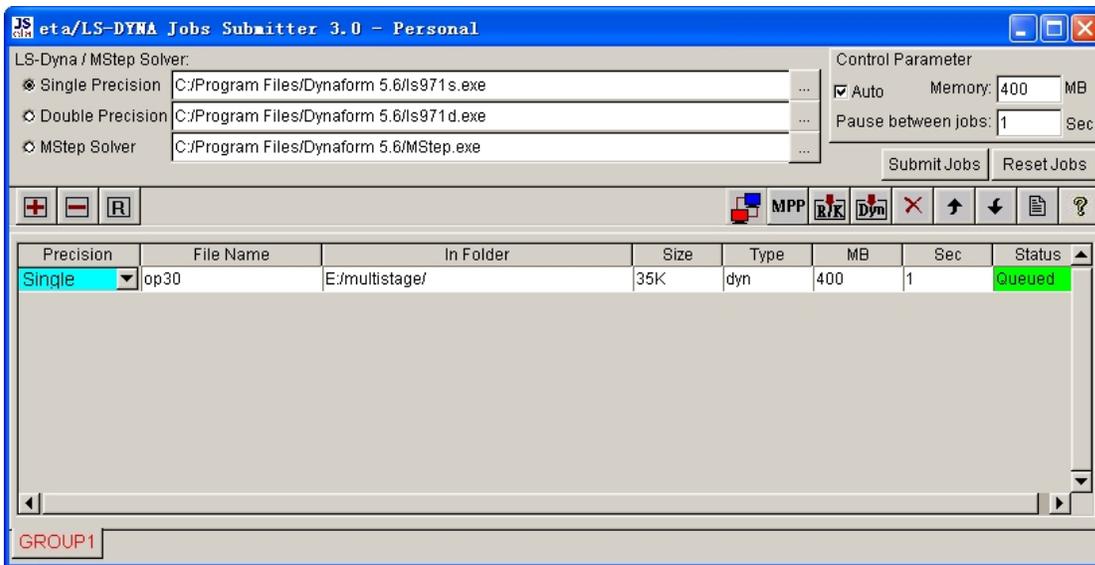


Figure 8.2.11 An illustration of Job Submitter GUI

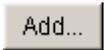
### 8.2.3 STAGE SETUP

Stage Setup bar is located below the menu bar. The incomplete stages are marked with red labels. The user may switch between stage pages to edit the setting of setup such as stage number and name, boundary

condition, process parameters, etc. The functions provided in Stage Setup bar are described as following:



Figure 8.2.12 An illustration of Stage Setup

-  : Click on these buttons to display hidden stages.
-  : Click on this button to display or hide the stage edit tools.
-  : Click this button to frozen current stage. The stages created after current stage will not be output or submitted for calculation and it is marked with dark font.
-  : Click on this button to move forward the current stage. The stage number remains unchanged.
-  : Click on this button to move backward the current stage. The stage number remains unchanged.
-  : Click on this button to delete the current stage.
-  : Click on this button to add new stage. The dialog box illustrated in Figure 8.2.13 is displayed.

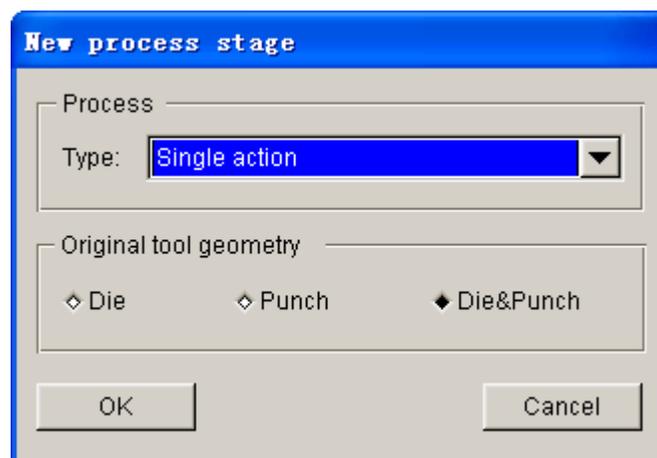


Figure 8.2.13 New Process Stage dialog box

## 8.2.4 FORMING

Each forming stage consists of one of the following processes: Crash Form, Single Action, Double Action and Triple Action. There are of six basic setup pages in each forming stage. Description of each pages are listed in the following sections.

### 8.2.4.1 GENERAL

The General page enables the user to define information such as Title and Working coordinate system for the current stage. A typical General page is illustrated in Figure 8.2.14.

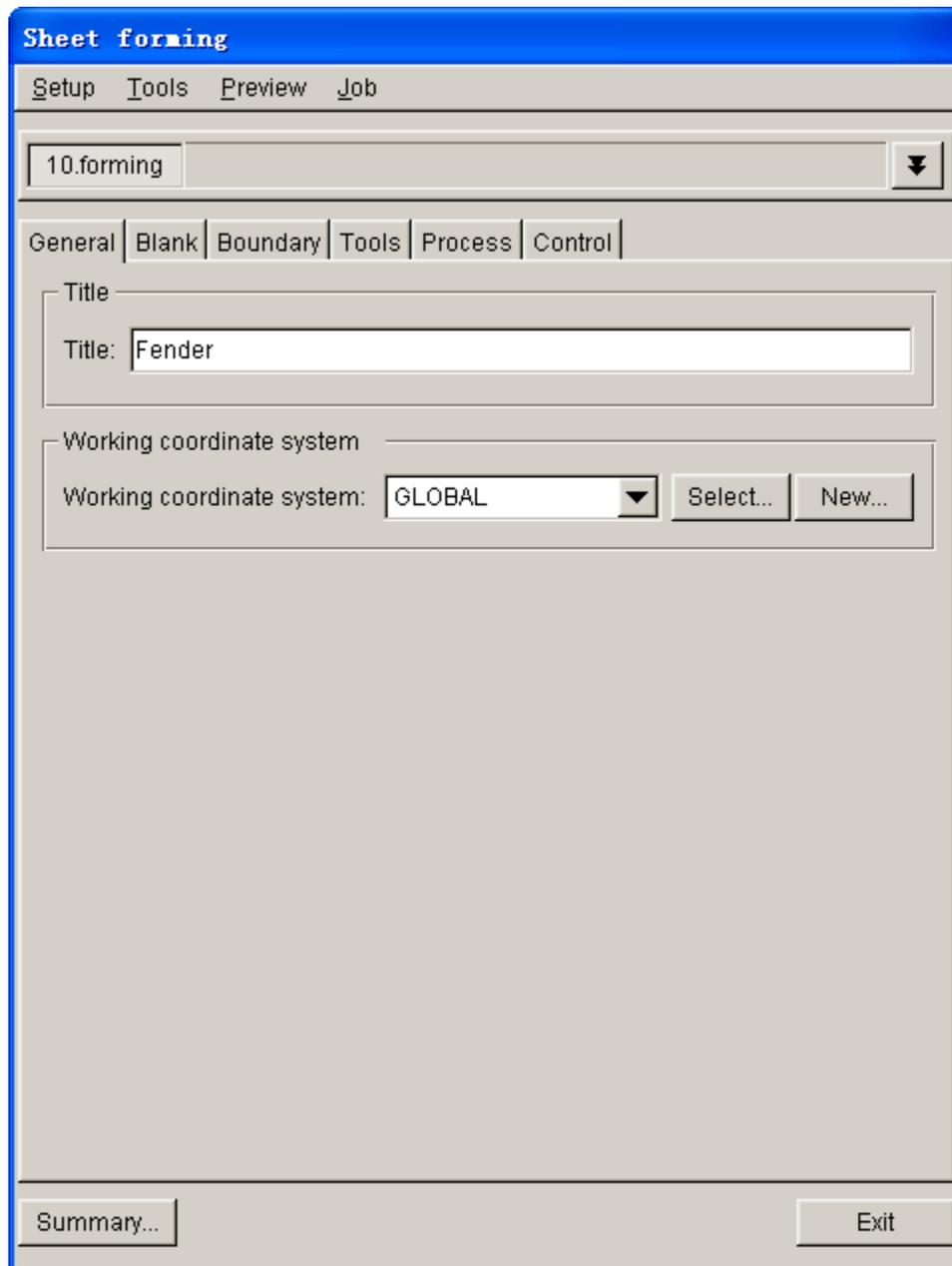


Figure 8.2.14 A typical General page

**TITLE**

This function allows the user to key in the title for each forming stage. For multi-stage stamping simulation, the user can define different titles for each stage.

**WORKING COORDINATE SYSTEM**

This function allows user to set the working coordinate system for each stage. Auto Setup supports arbitrary stamping direction. Default working coordinate system is set as Global Z direction. The user may choose a CS from the list of coordinate system as Working Coordinate System of the current stage, or click on Select button to select from the display screen. If local stamping direction is needed, the user can click New button to define LCS for the current stage. The local W direction is chosen as the stamping direction. The newly created LCS will be automatically set as the working CS of the current stage. Refer to Section 2.5 for description about creating LCS.

**8.2.4.2 BLANK**

In Blank page, the user can define Part, Material, Thickness and Property of blank. Furthermore, the user may define weld line for tailor-welded blank application, contact interface for laminated sheets and symmetry condition. The interface of Blank page is illustrated in Figure 8.2.15.

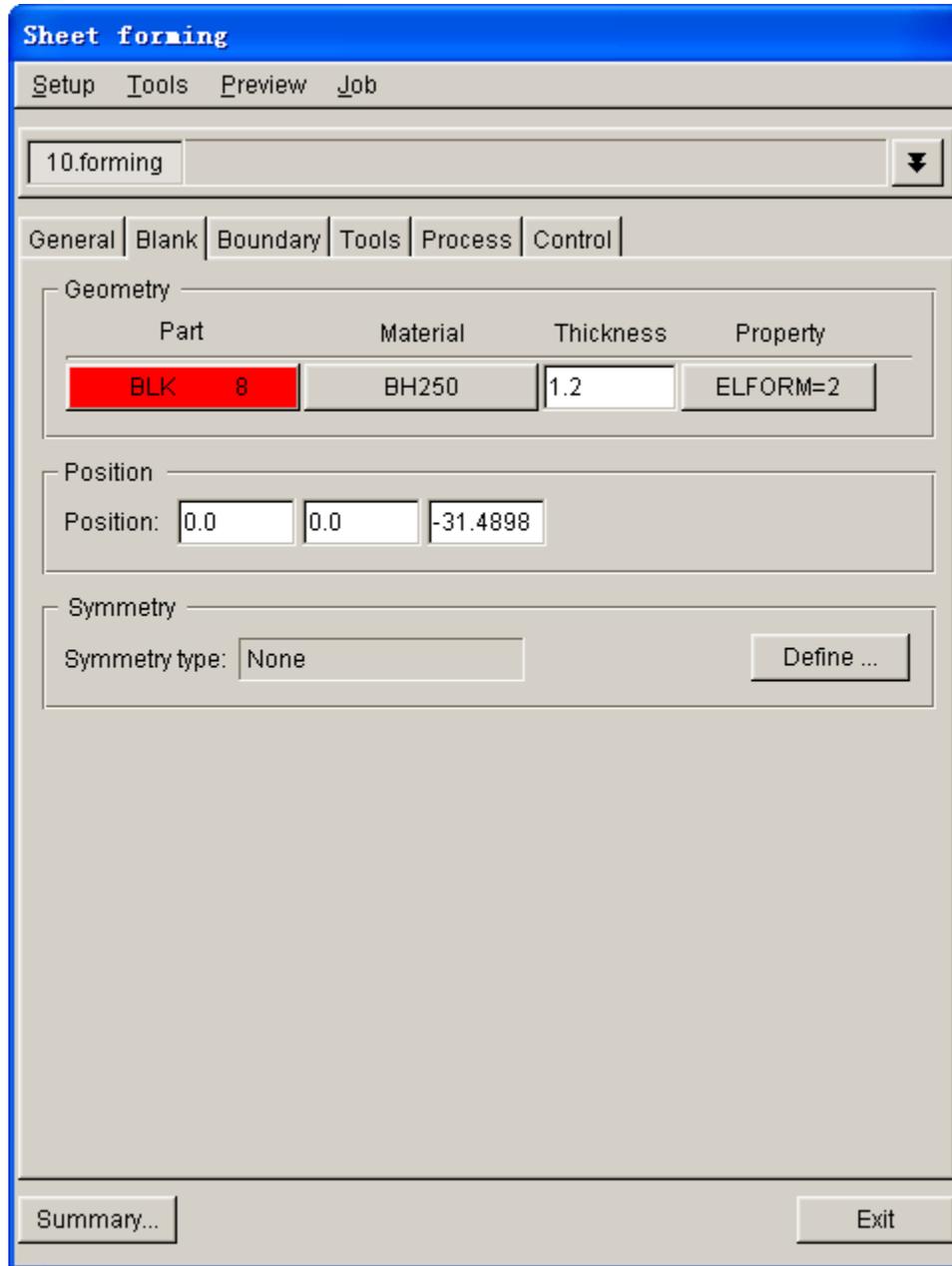


Figure 8.2.15 Interface of Blank Page

## GEOMETRY

### PARTS

As shown in Figure 8.2.16, the Part field allows the user to assign a part as blank for stamping simulation. After clicking the Define geometry... button, the DEFINE GEOMETRY dialog box illustrated in Figure 8.2.17 is displayed.

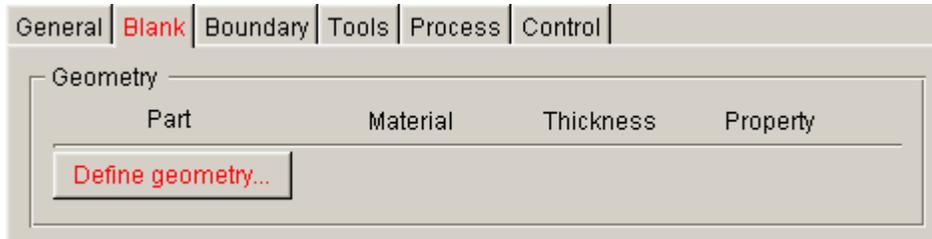


Figure 8.2.16 Geometry field

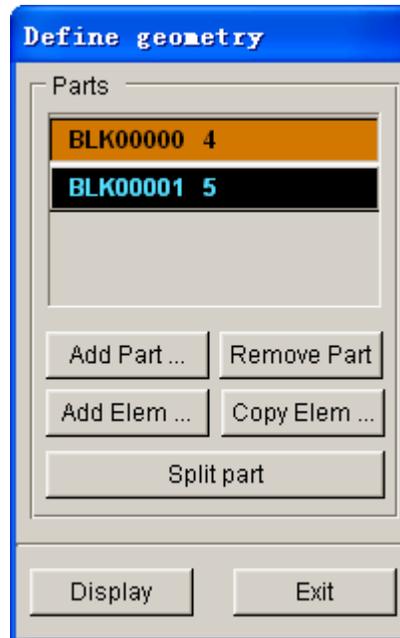


Figure 8.2.17 Define Geometry dialog box

**ADD PART...:** This function allows user to define blank by selecting one or more parts from the part list. Once selected, the parts are highlighted on the screen, as well as in the Parts List of SELECT PART dialog box.

**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 as blank.

**COPY ELEM...:** This function allows the user to copy or offset elements from existing elements. The elements will be added to a new part, which is then defined as blank.

**SPLIT PART:** This function allows user to split the blank into several blanks. Different material, thickness and properties can be assigned to each blank. The user can easily generate the blanks for stamping simulation using tailor-welded blank.

**DISPLAY:** This function allows the user to display all the parts which are defined as blank. The user may easily check and repair the blank mesh without displaying other parts on the screen.

**EXIT:** This function allows the user to dismiss the DEFINE GEOMETRY dialog box.

After blank parts are defined, the names of each part are displayed in the Part field illustrated in Figure 8.2.18. The default material, thickness and property are assigned for the part. The user may reassign material, thickness and property for each part.

**MATERIAL**

The Auto Setup defines a default type 36 material for blank part. The user may click **BLANKMAT** button under Material to modify or create a material. After clicking the button, the Material window illustrated in Figure 8.2.19 is displayed.

Part	Material	Thickness	Property
BLK00000 4	BLANKMAT	0.7	ELFORM=2
BLK00001 5	BLANKMAT	0.7	ELFORM=2

Figure 8.2.18 Define material

**NOTE:** If no material is assigned to the Part, the BLANKMAT is listed in material button. After defining the material, the material name is displayed on the button.

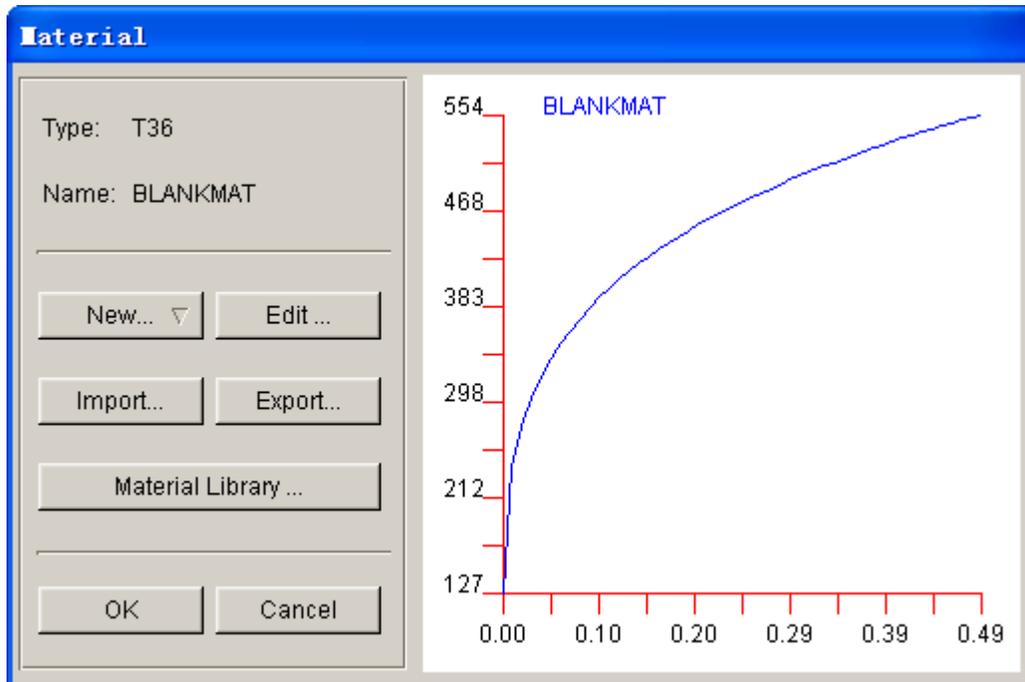


Figure 8.2.19 Material Define dialog box

**TYPE:** Display defined material type. Auto Setup currently supports the following material types: 1, 4, 18, 24, 36, 37, 39, 64, 106 and 125.

**NAME:** Display the defined material name.

**NEW...:** Allow the user to create new blank material. After clicking this button, the list of material types illustrated in Figure 8.2.20 is displayed. The user may select one of the material types from the list. Once a material type is chosen, the MATERIAL dialog box illustrated in Figure 8.2.21 is displayed.

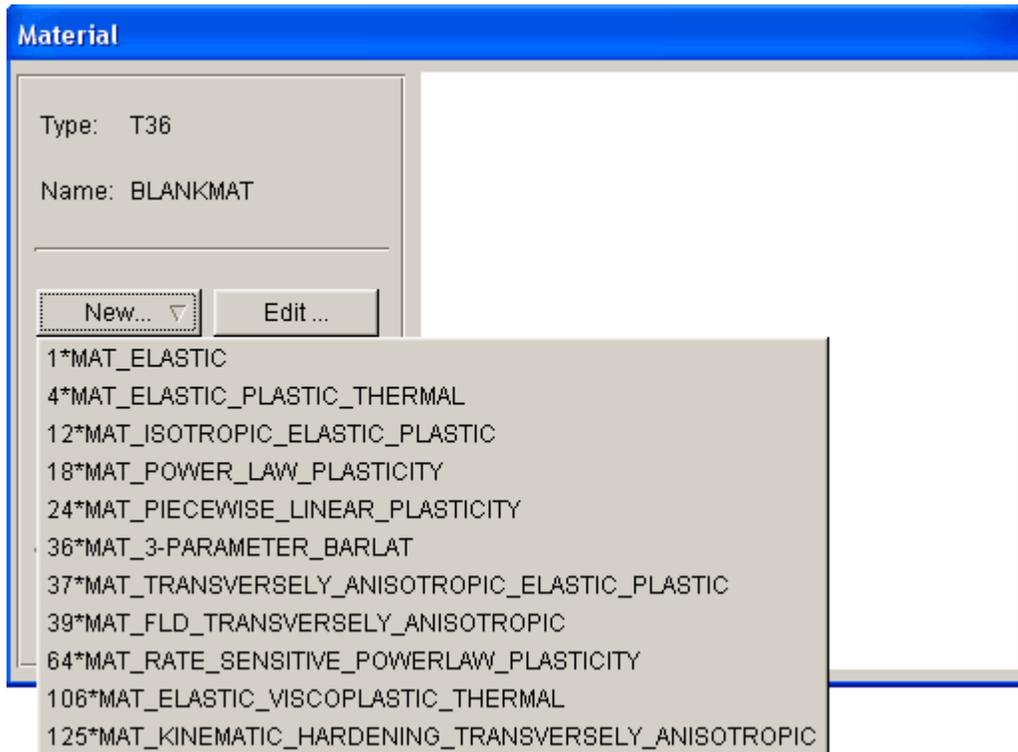


Figure 8.2.20 Select material type

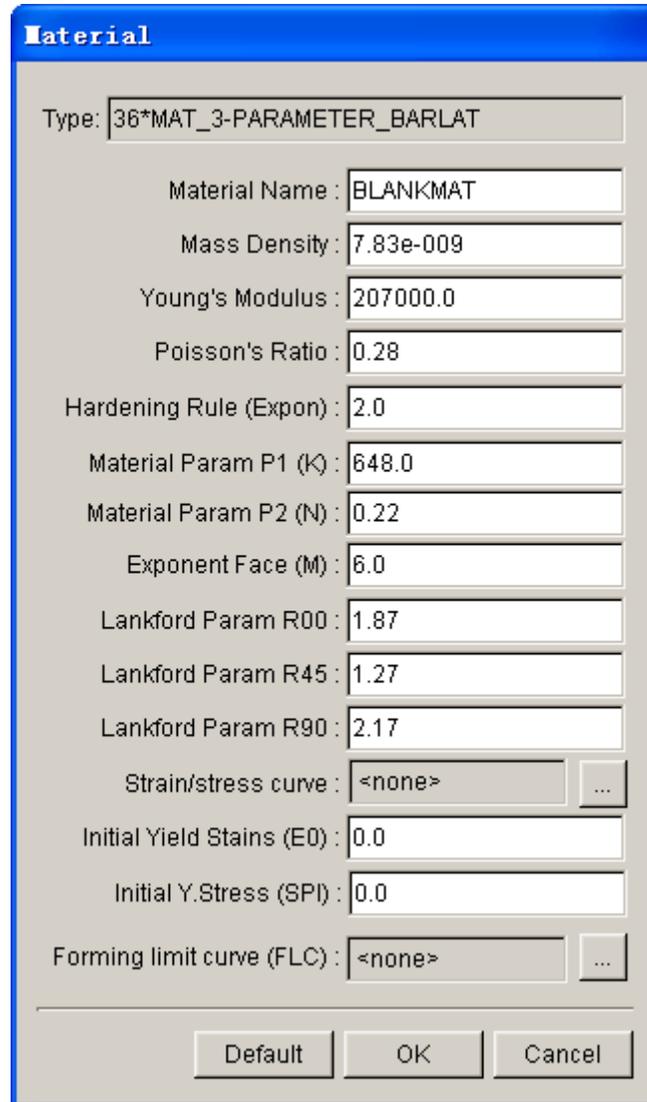


Figure 8.2.21 Material dialog box

**Forming Limit Curve (FLC):** This parameter should be defined if material type 39 is chosen. If no FLC curve is defined, <NONE> is displayed at the FLC input field. To define FLC curve, click  button to display the FLC curve edit dialog box illustrated in Figure 8.2.22.

**NOTE:** *The FLC curve defined in Auto Setup is represented in engineering strain.*

In the FLC curve edit dialog box, the user may enter the FLC data points directly in INPUT page. Alternatively, the user may define FLC curve using KEELER’s FLC empirical equation provided in the FORMULA page. After entering the n and t values, click the APPLY button to generate the FLC. The FLC curve is displayed in the show curve interface illustrated in Figure 8.2.22. Detailed description of curve edit operation is provided in Section 8.4.6.5.

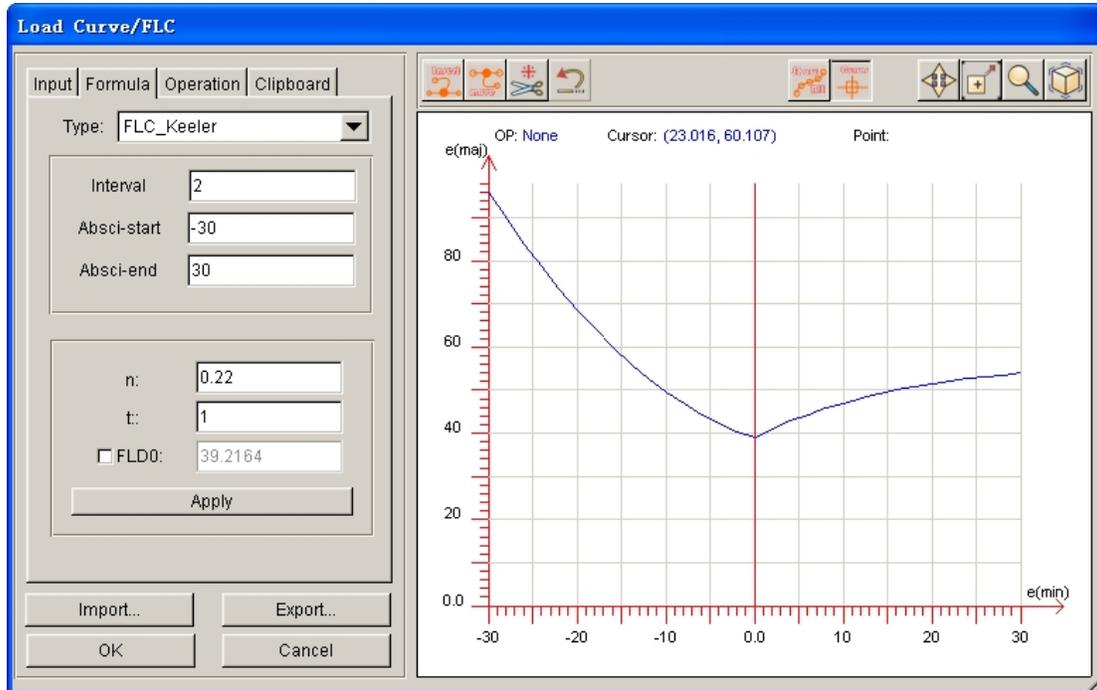


Figure 8.2.22 FLC Curve

**EDIT...:** Allow the user to edit the material parameters listed in the MATERIAL dialog box illustrated in Figure 8.2.23.

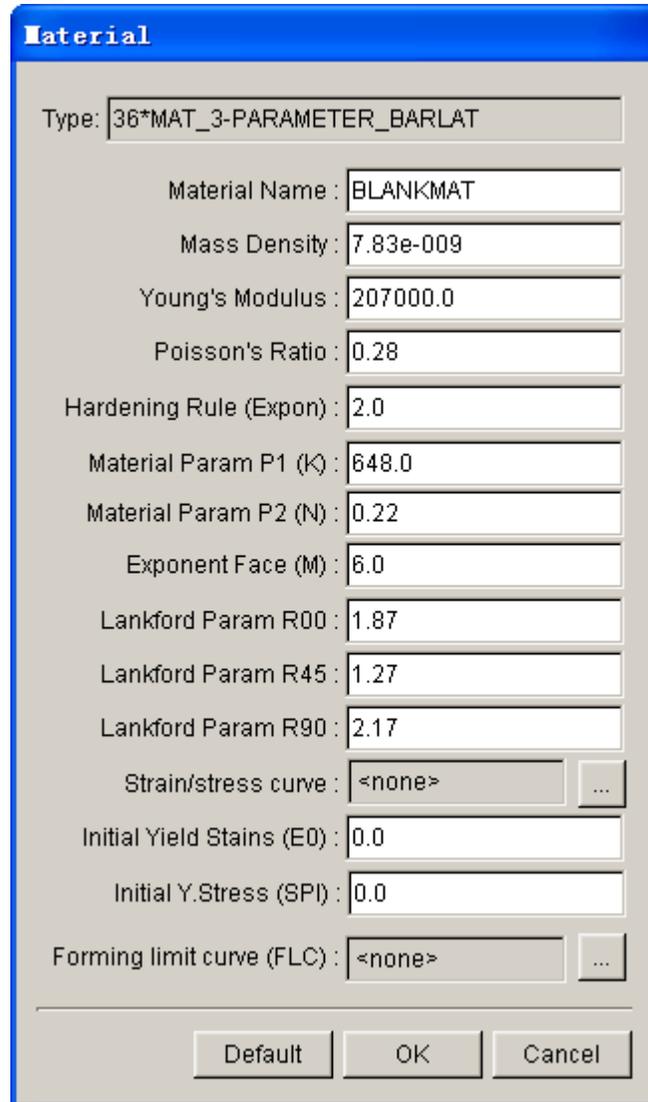


Figure 8.2.23 Material Edit dialog box

**NOTE:** *Edit Material and Define Material dialog boxes are basically identical. 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 create another the material properties.*

**IMPORT:** Allow the user to import a material file. The extension of the file name is “.mat”. By default, Import Material window illustrated in Figure 8.2.24 is displayed when the button is hit.

**NOTE:** *For PC users, the extension of material file may be associated with Microsoft Office Access. The “.mat” extension will not be displayed, but the extension of material files may be viewed from the console model. To display the “.mat” extension, the user must delete the association between the “.mat” extension and Microsoft Office Access.*

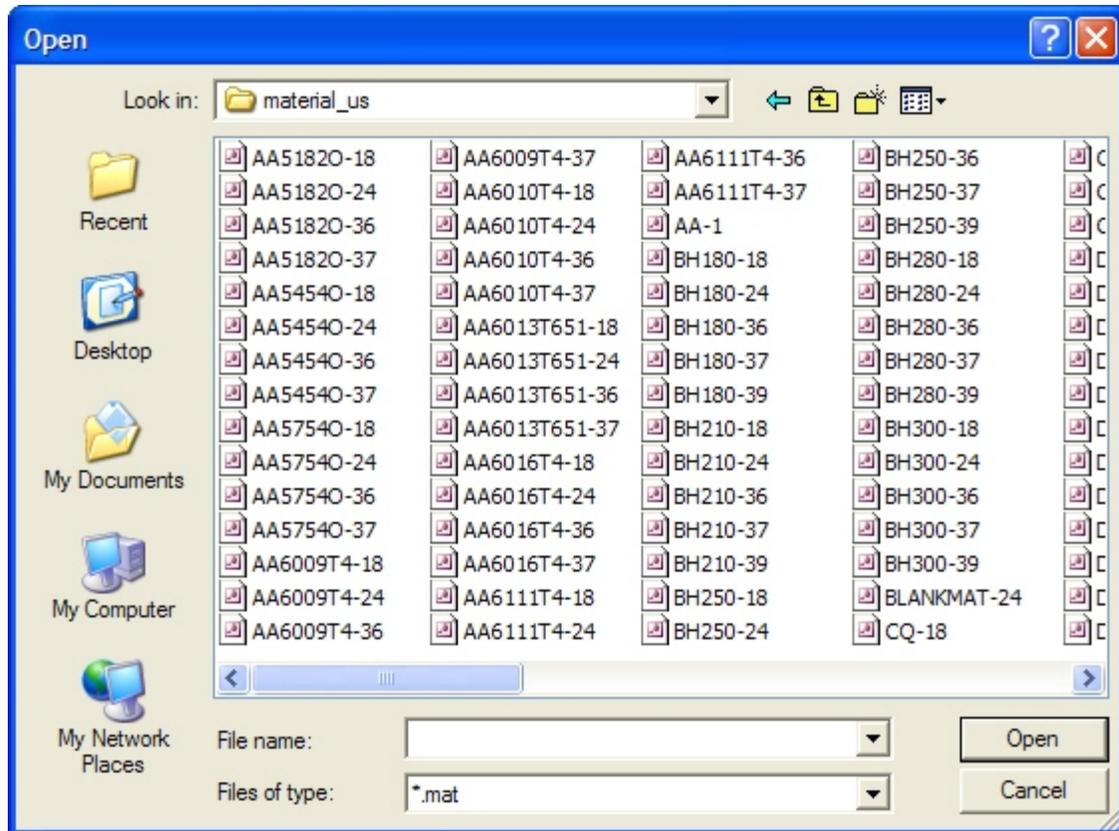


Figure 8.2.24 Import Material window

After selecting a file, the user may click IMPORT button to import selected material and assign it to the selected blank part.

**EXPORT:** Allow the user to export and write the current material into a file. The file is saved with an extension “.mat”. The Export Material window is illustrated in Figure 8.2.25.

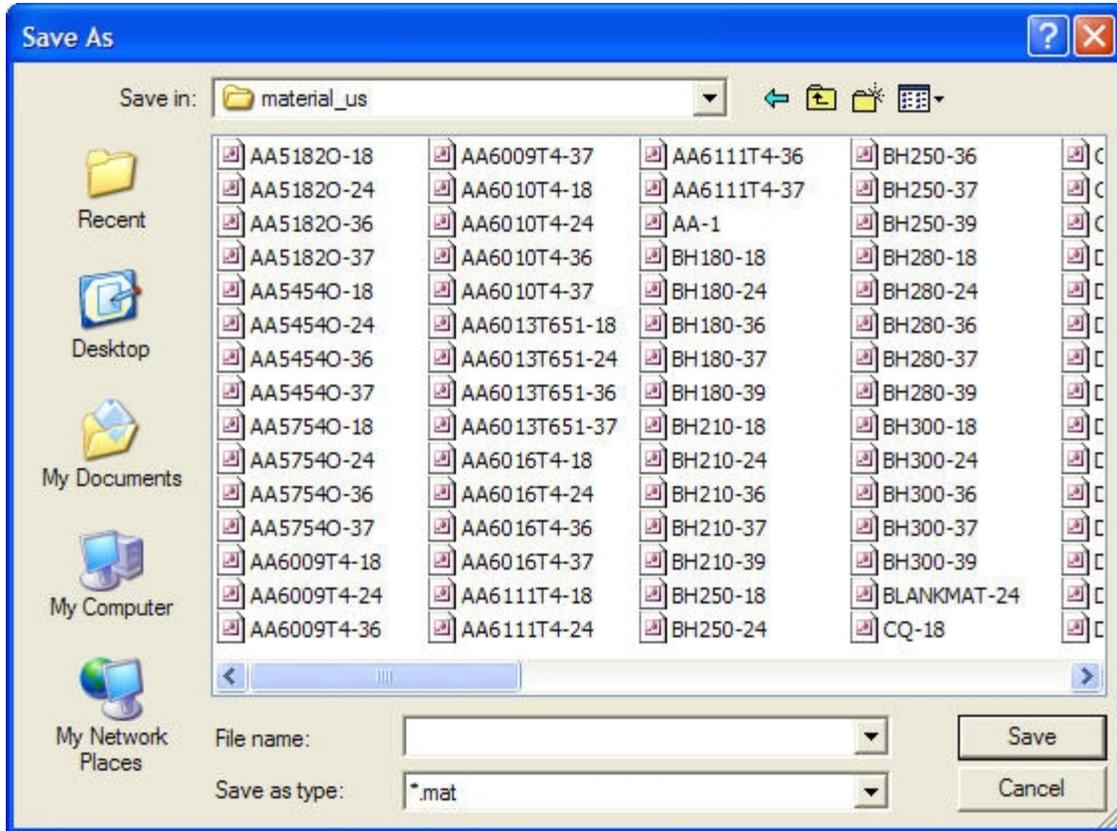


Figure 8.2.25 Export Material window

**MATERIAL LIBRARY:** This function enables the user to select the type of generic material library in eta/DYNAFORM. There are four types of generic material library supported by eta/DYNAFORM. They are United States, JAPAN, CHINA and EUROPE. The User Defined material library option is provided to enable users to build their own material library.



Figure 8.2.26 Material Library Standard

After selecting type of material library, the material library GUI illustrated in Figure 8.2.27 is displayed. The user proceeds by selecting type of material from library to be assigned as the blank material.

The Material Library interface allows the user to conveniently select a type of material. The right key menu enables the user to expand each node of tree structure. The number listed in the parentheses next to each material indicates the material type. The previously selected material type number will be displayed in the parentheses.

*Note: The user-defined material is saved in the user-defined directory. The user can set the default path of material library from Option/Material Library menu.*

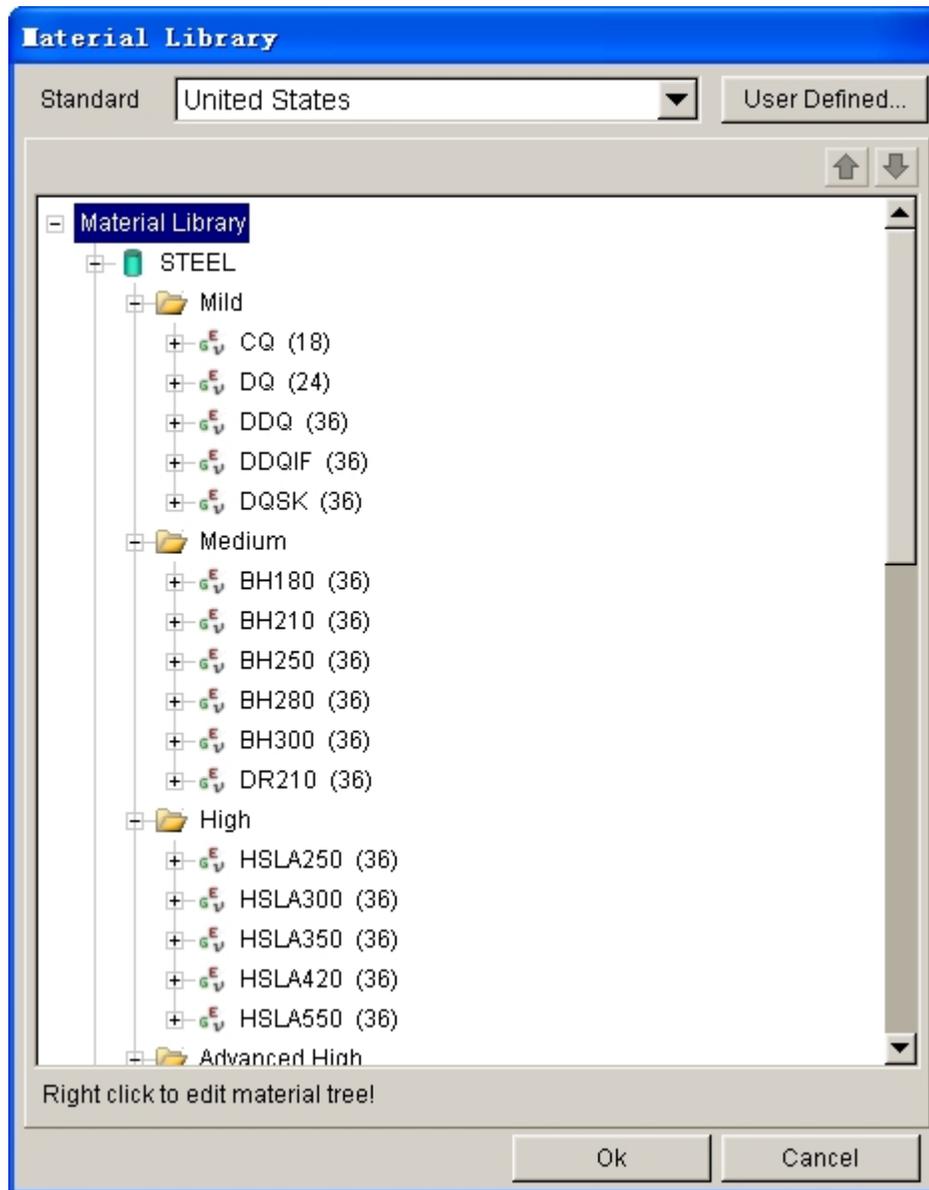


Figure 8.2.27 GUI of Material Library

- **User Defined**

This function allows the user to add, delete and rename the user-defined material library. The added material library will be shown in the list of material library. The User Defined Standard dialog box is illustrated in Figure 8.2.28.

**ADD:** Add and define the name of user-defined material library.

**RENAME:** Edit the name of user-defined material library.

**DELETE:** Remove the user-defined material library.

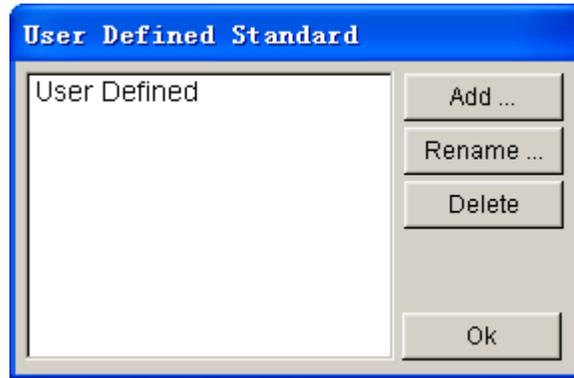


Figure 8.2.28 User Defined Standard dialog box



This function allows the user to jump between the types of material model within the material strength level. If the material strength level is highlighted, the user can click the button to jump between the types of material strength level within the material category.

- **Material Category**

Click right mouse button at the root node of tree structure to create a new material category, strength level and type of material. Default material categories are Steel and Aluminum. The user can assign name to the new material category, and rename or delete material category. As illustrated in Figure 8.2.29, the list of each node is displayed if the right mouse button is hit.

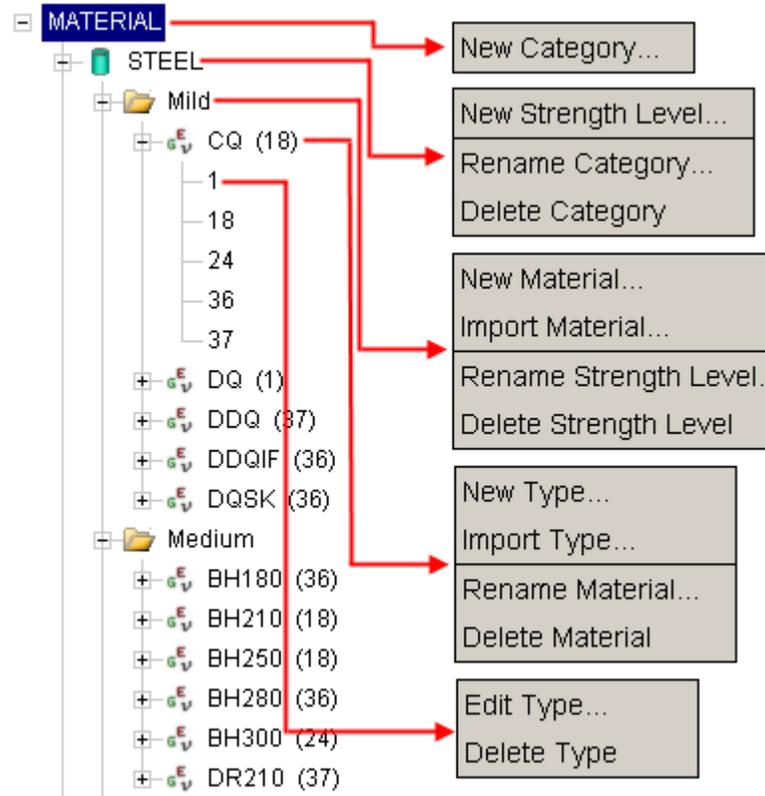


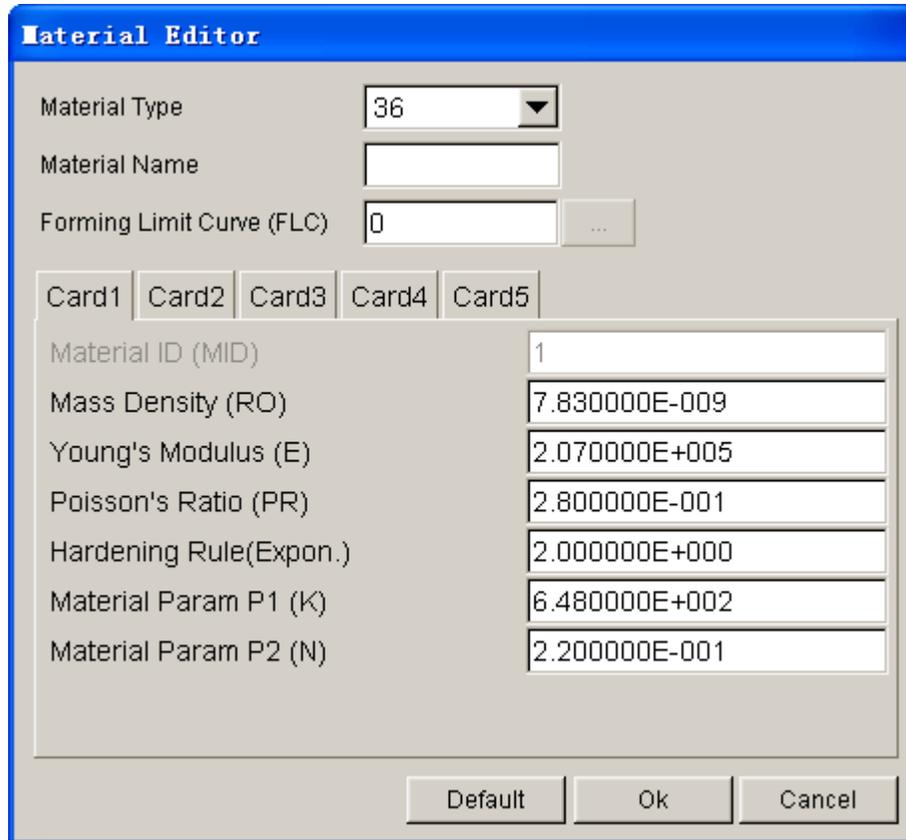
Figure 8.2.29 The corresponding list of each node

- **Strength Level**

The material library allows the user to classify the following steel grades: Low, Medium, High, Advanced High, Hot Roll and Cold Roll. Users can easily manage their own material library.

- **Material Name**

The user can click the right mouse button of Strength Level to define a new material. The Create Material dialog box illustrated in Figure 8.2.30 is displayed. The user can select material type and edit material parameters in this dialog box. The default material name is similar to its title. The user must specify the path of the new material prior to saving it.



The Material Editor dialog box is shown with the following fields and values:

Field	Value			
Material Type	36			
Material Name				
Forming Limit Curve (FLC)	0			
Card1	Card2	Card3	Card4	Card5
Material ID (MID)	1			
Mass Density (RO)	7.830000E-009			
Young's Modulus (E)	2.070000E+005			
Poisson's Ratio (PR)	2.800000E-001			
Hardening Rule(Expon.)	2.000000E+000			
Material Param P1 (K)	6.480000E+002			
Material Param P2 (N)	2.200000E-001			

Buttons: Default, Ok, Cancel

Figure 8.2.30 Create Material dialog box

- **Material Type**

The user can click the right mouse button of Material Name to define or import the new material type. The user can also click right mouse button of Material Type to edit or delete material type. The Create Material Type dialog box is illustrated in Figure 8.2.31. After the material is imported, the type of material number is automatically assigned.

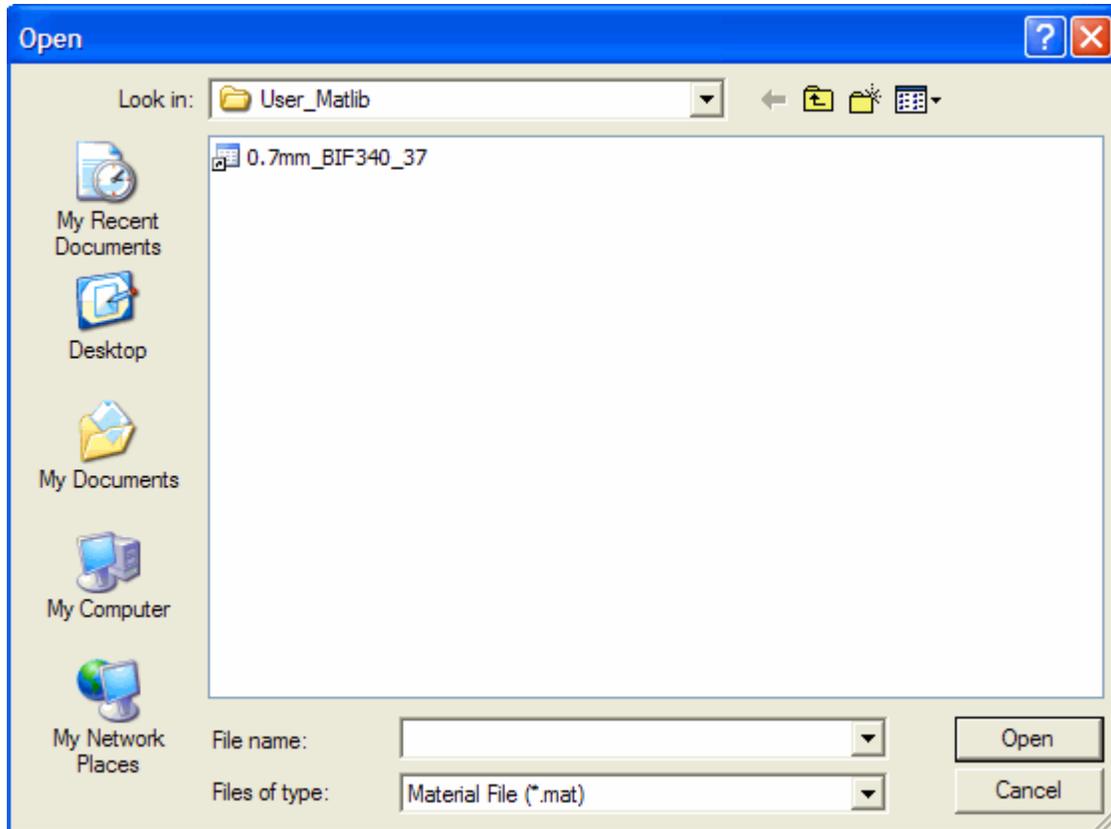


Figure 8.2.31 Import Material dialog box

**THICKNESS**

By default, the thickness value for blank is defined in the New Simulation dialog box. The user can edit thickness of blank in Thickness field.

**PROPERTY**

By default, the properties of blank are set as illustrated in Figure 8.2.32. These blank properties are good for most stamping simulation.

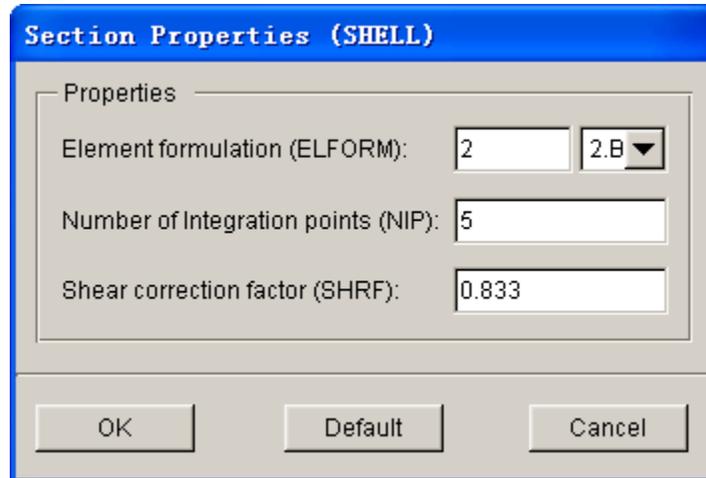


Figure 8.2.32 Define Blank Properties

**ELEMENT FORMULATION:** This option allows the user to choose type of element formulation. Currently, 16 shell element formulations are supported in eta/DYNAFORM. In addition, thick shell and solid element formulation are available. The default element formulation is No. 2 BELYSTSCHKO-TSAY shell element formulation. This element formulation is the most popular and economical one for sheet metal forming simulation. For springback analysis, it is recommended to select No. 16 Full Integration element formulation in both draw simulation and springback analysis.

**NUMBER OF INTEGRATION POINTS:** The default number of integration points is 5. Increasing through thickness integration points may improve the precision of simulation result. On the other hand, the simulation time will drastically increase. For draw simulation without considering springback, the default through thickness integration points is recommended. If there is draw forming simulation with subsequent springback analysis, through thickness integration points of 7 or greater is suggested.

**OK:** Save the current setting and exit Define Property dialog box.

**DEFAULT:** All parameters are restored to the default value.

**CANCEL:** Exit the dialog box without saving the modification of current setting.

**POSITION**

This function allows the user to move the position of blank along U, V and W direction illustrated in Figure 8.2.33. After auto positioning, the movement along W direction is listed in the Position field.

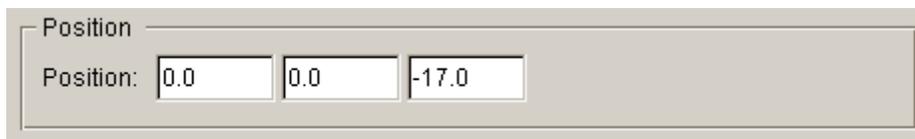


Figure 8.2.33 Position field

**SYMMETRY**

This function allows the user to create symmetry condition for the blank to reduce simulation time. The user may define either  $\frac{1}{2}$  or  $\frac{1}{4}$  symmetrical condition along the line of symmetry of blank.  $\frac{1}{2}$  and  $\frac{1}{4}$  blank mesh is required for applying  $\frac{1}{2}$  and  $\frac{1}{4}$  symmetrical conditions, respectively. By default, None is displayed in the field of Symmetry type illustrated in Figure 8.2.40, indicating no symmetry condition is applied to blank. The user may click **Define ...** button to display the SYMMETRY PLANE dialog box illustrated in Figure 8.2.34.

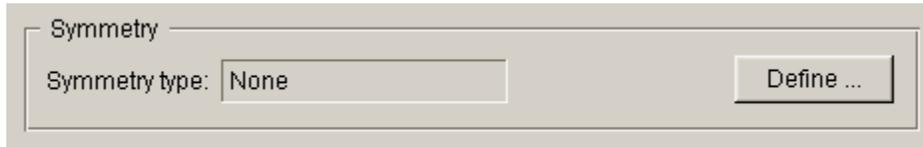


Figure 8.2.34 Symmetry field

### 1/2 SYMMETRY

This function allows the user to define  $\frac{1}{2}$  symmetrical blank. The interface is illustrated in Figure 8.2.35.

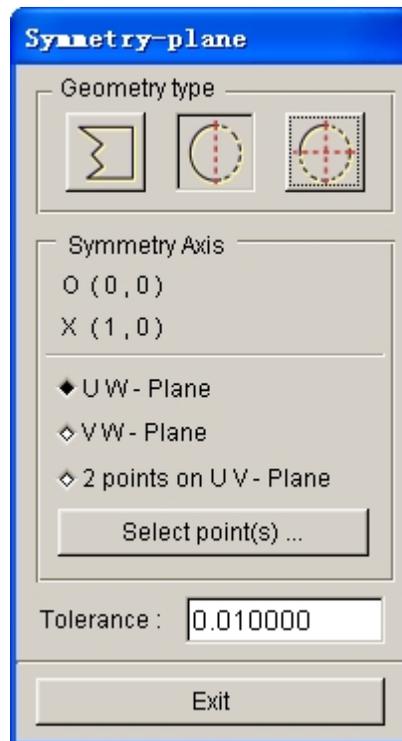


Figure 8.2.35 Define Symmetry Plane

**SYMMETRY AXIS:** The current user defined symmetry plane information is displayed in this field. The symmetry plane is specified by a point, a vector and WCS direction.

**O(0,0):** Indicate the coordinate of starting point of the vector defining symmetry plane.

**X(1,0):** Indicate either UW- or VW-plane chosen as symmetry plane.

**UW-PLANE:** A plane parallel to WCS's UW plane is chosen as symmetry plane. The user specifies the location of plane by using Select Points function illustrated in Figure 8.2.35. The position of symmetry plane is displayed in O (X, Y), while the direction of symmetry is displayed in X (1, 0). A vector is displayed in WCS's UW plane on the screen to show the symmetry plane. See Figure 8.2.36 for more details.

**VW-PLANE:** A plane parallel to WCS's VW plane is chosen as symmetry plane. The user specifies the location of plane by using Select Points function illustrated in Figure 8.2.35. The position of symmetry plane is displayed in O (X, Y), while the direction of symmetry is displayed in X (0, 1).

**2 POINTS ON UV-PLANE:** Enable the user to define the vector of symmetry plane by selecting two points. The symmetry plane is parallel to W axis. The first point of the vector specifies the position of the symmetry 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 option, only one point is required to specify the location of symmetry plane. If the user selects 2-Points on UV-plane option, then two points are required to define symmetry plane.

**TOLERANCE:** Default tolerance is 0.01. Nodes fall inside the tolerance are defined with symmetry boundary condition. The user may increase tolerance to extend the search range for nodes on symmetry plane.

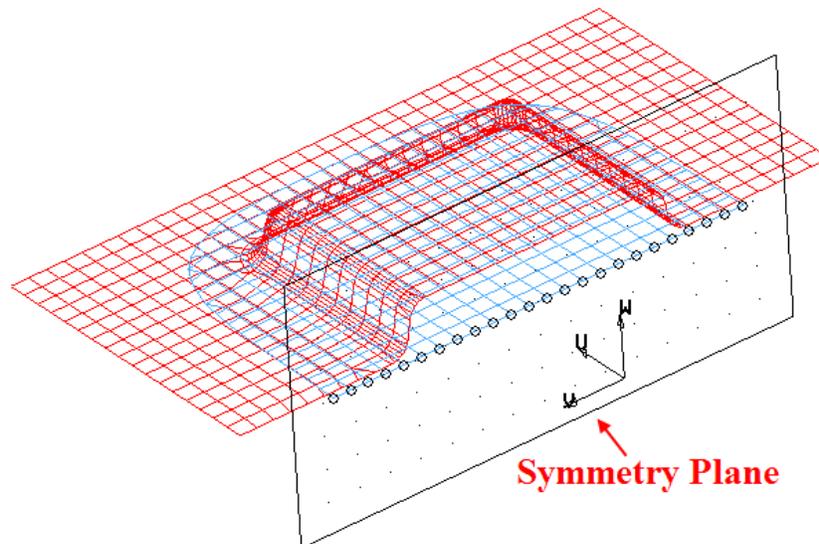


Figure 8.2.36 Define Symmetry

### 1/4 SYMMETRY

This function allows the user to apply symmetry boundary condition on 1/4 symmetry blank. After defining a vector, another vector normal to the defined vector is automatically created to specify the other symmetry plane. Figure 8.2.37 illustrated a typical definition of 1/4 symmetry planes.

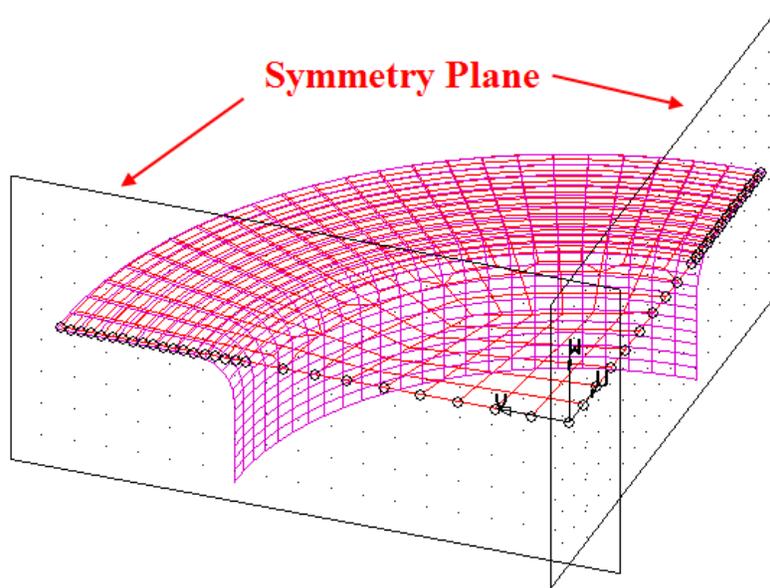


Figure 8.2.37 Define 1/4 Symmetry

### WELDS

The user can conveniently define tailor-welded and composite blanks. After multiple blank parts are defined, the Welds option is activated on Blank page illustrated in Figure 8.2.38. After the weld lines are defined, the information of weld lines is displayed in Welds list.

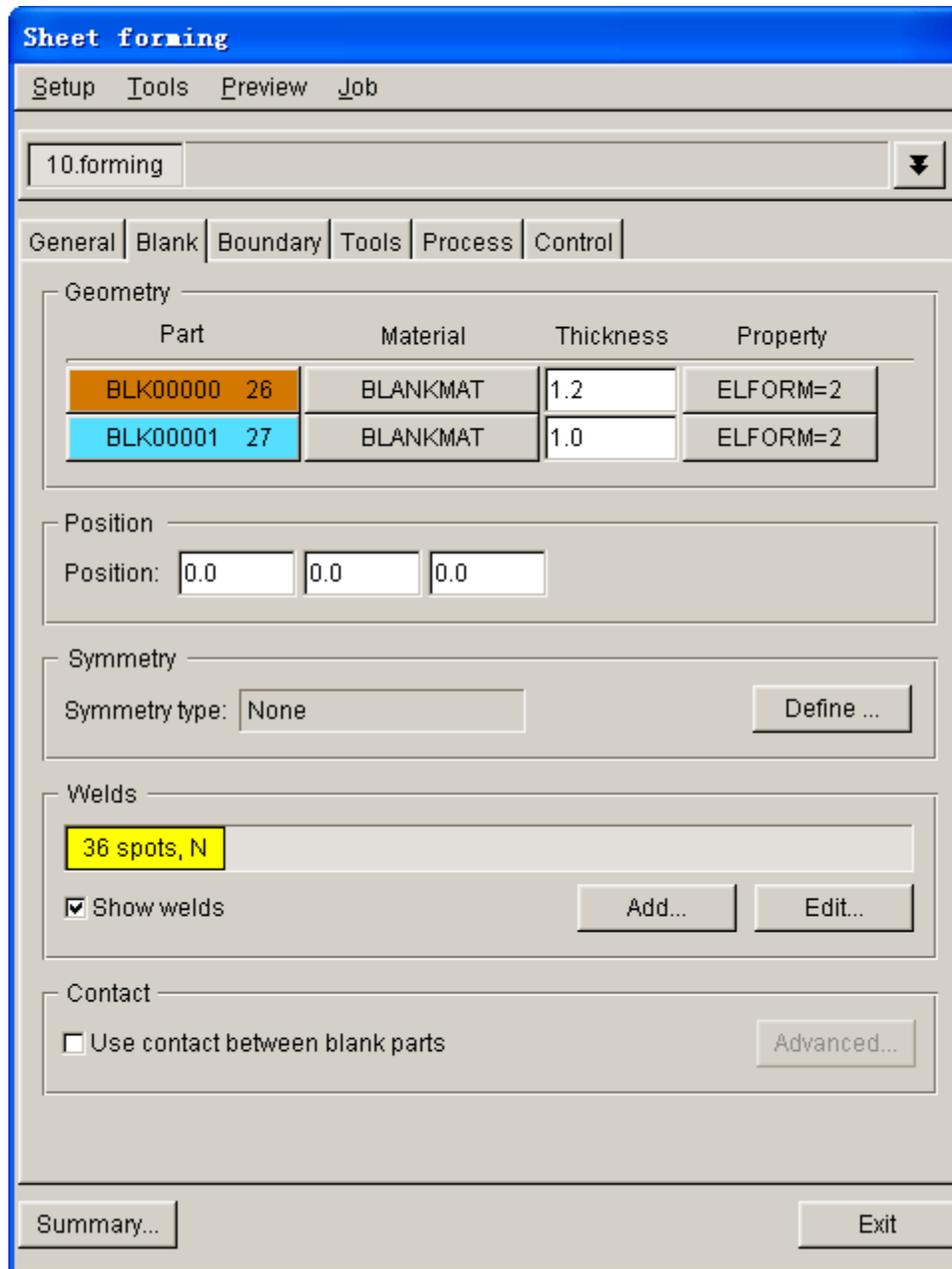


Figure 8.2.38 Define Tailor-welded Blank

**ADD...**

This function allows the user to select adjacent nodes to define weld element. After clicking this button, the DEFINE BLANK WELD dialog box illustrated in Figure 8.2.39 is displayed.

**NOTE:** In LS-DYNA, the pairing nodes of weld must consist of two separate nodes with different positions. In order to reduce the preparation time, eta/DYNAFORM will automatically separate the identical nodes which are used to define the weld. User only needs to add the corresponding elements to different parts.

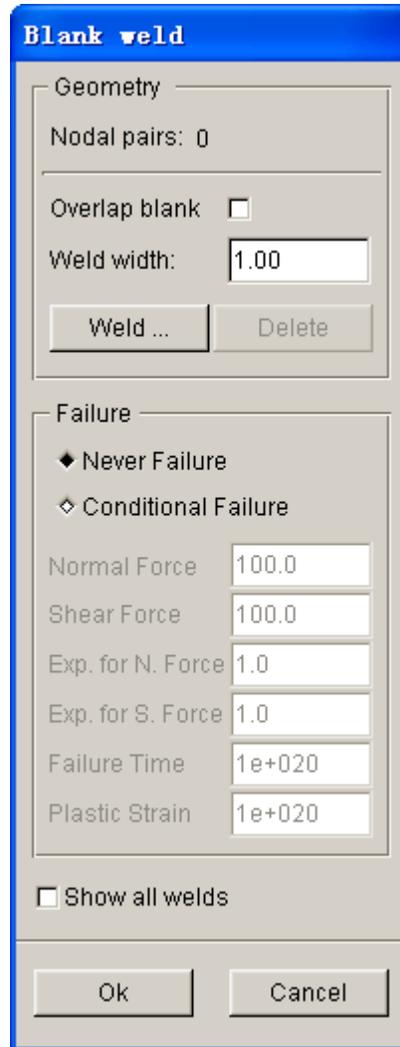


Figure 8.2.39 Define Weld dialog box

**NODAL PAIRS:** Display number of nodal pairs. After the user selecting nodal pairs, 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 is normal to its connecting element.

**WELD WIDTH:** Specifies the width of weld. Default width is 1.0 (mm). If the distance between selected node pair is smaller than default value, weld is defined for the selected node pair. The user may adjust the width to include the node pairs with larger gap.

**INPUT...:** The user may click this button to select node pair. From the displayed SELECT NODE PAIR dialog box, select two rows of nodes required for definition of weld line. After selecting the nodes, weld as illustrated in Figure 8.2.40 is defined between nodal pairs.

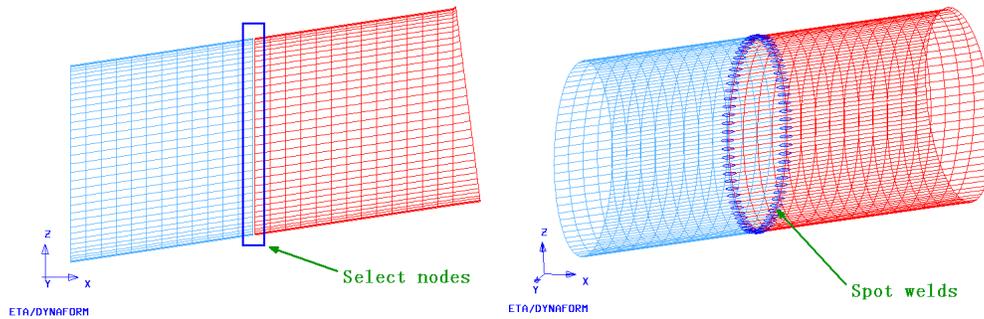


Figure 8.2.40 Define Weld

As illustrated in Figure 8.2.41, the nodes are separated and automatically offset if only one row of nodes is available.

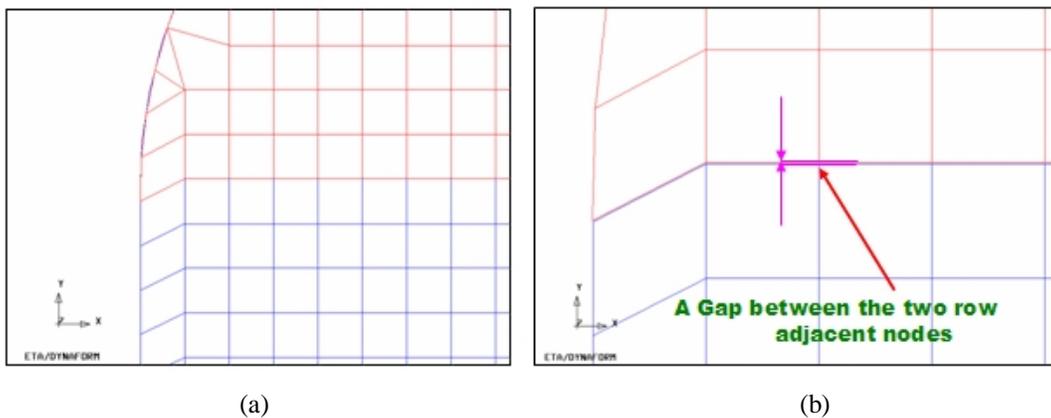


Figure 8.2.41 Offset Node (a) Before (b) After

**DELETE:** This button allows the user to remove all current node pairs.

**FAILURE:** The user may define failure criteria for the weld.

**NEVER FAILURE:** By default, the NEVER FAILURE option is chosen to ensure weld does not fail during the stamping simulation.

**CONDITIONAL FAILURE:** Allow the user to define conditional failure criteria and its relevant parameters. Detailed description is available in LS-DYNA User’s Manual.

**EDIT**

This function allows the user to edit the defined weld. The user may delete selected nodal pair, add new nodal pair, modify weld failure criteria, etc.

**DELETE**

After a weld line is defined, weld information such number of nodal pairs, failure criteria of weld is displayed in the weld list illustrated in Figure 8.2.42. The user may delete a weld by clicking the Edit button, following by clicking the Delete button to remove the current weld definition.

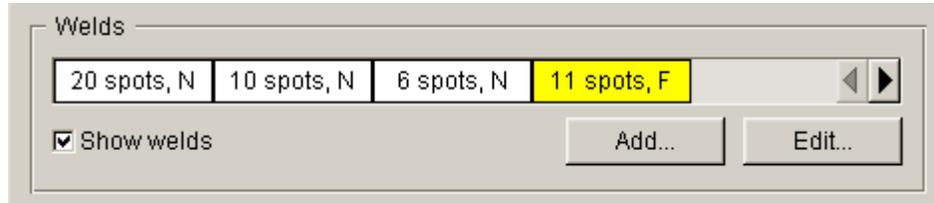


Figure 8.2.42 Welds List

## CONTACT

For overlap blanks, the user should define the contact interface between blank parts illustrated in Figure 8.2.43. The user may modify contact parameters, such as friction coef., contact coef. type, etc. for contact interface between blank parts. For tailor-welded blank, the user should toggle off the checkbox for the contact between the blank parts.

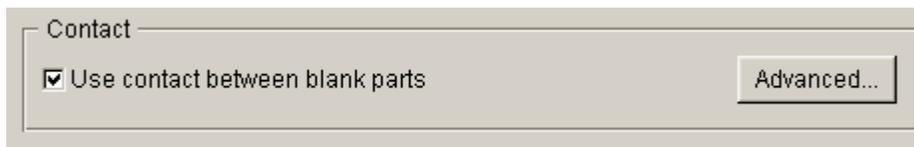


Figure 8.2.43 Define Contact Between Blanks

## USE CONTACT BETWEEN BLANK PARTS

This option should be toggled on for overlap blanks.

### ADVANCED...

The user may modify some contact parameters between blanks through advanced options illustrated in Figure 8.2.44.

Parameter	Value
Type	AUTOMATIC_SURFACE_TO_SURFACE_TIEE
Static friction coef. (FS)	0.125
Viscous friction coef. (VC)	0.0
Viscous damping coef. (VDC)	20.0
Dynamic friction coef. (FD)	0.0
Decay coef. (DC)	0.0
Normal failure stress (NFLS)	30.0
Shear failure stress (SFLS)	10.0

Figure 8.2.44 Define Advanced Contact Parameters

**CONTACT TYPE:** This function allows the user to select type of contact algorithm illustrated in Figure 8.2.45.

```
AUTOMATIC_NODES_TO_SURFACE
AUTOMATIC_NODES_TO_SURFACE_SMOOTH
AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE
AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_TIEBREAK
AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_SMOOTH
AUTOMATIC_SURFACE_TO_SURFACE
AUTOMATIC_SURFACE_TO_SURFACE_TIEBREAK
AUTOMATIC_SURFACE_TO_SURFACE_SMOOTH
FORMING_NODES_TO_SURFACE
FORMING_NODES_TO_SURFACE_SMOOTH
FORMING_ONE_WAY_SURFACE_TO_SURFACE
FORMING_ONE_WAY_SURFACE_TO_SURFACE_SMOOTH
FORMING_SURFACE_TO_SURFACE
FORMING_SURFACE_TO_SURFACE_SMOOTH
NODES_TO_SURFACE
NODES_TO_SURFACE_SMOOTH
ONE_WAY_SURFACE_TO_SURFACE
ONE_WAY_SURFACE_TO_SURFACE_SMOOTH
SURFACE_TO_SURFACE
SURFACE_TO_SURFACE_SMOOTH
```

Figure 8.2.45 Define type of contact algorithm

**STATIC FRICTION COEF:** Allow the user to define static coefficient of friction. Default value is 0.125.

**VISCOUS FRICTION COEF:** Allow the user to define viscous coefficient of friction. Default value is 0.

**VISCOUS DAMPING COEF:** Allow the user to define viscous coefficient of damping. Default value is 20.0.

**DYNAMIC FRICTION COEF:** Allow the user to define dynamic coefficient of friction. Default value is 0.

**DECAY COEF:** Allow the user to define coefficient of decay. Default value is 0.

**NORMAL FAILURE STRESS:** Allow the user to define the normal failure stress.

**SHEAR FAILURE STRESS:** Allow the user to define the shear failure stress.

### 8.2.4.3 BOUNDARY CONDITION

This page allows the user to define single point constraints, acceleration time and advanced parameters for thermal analysis. The boundary condition interface is illustrated in Figure 8.2.46.

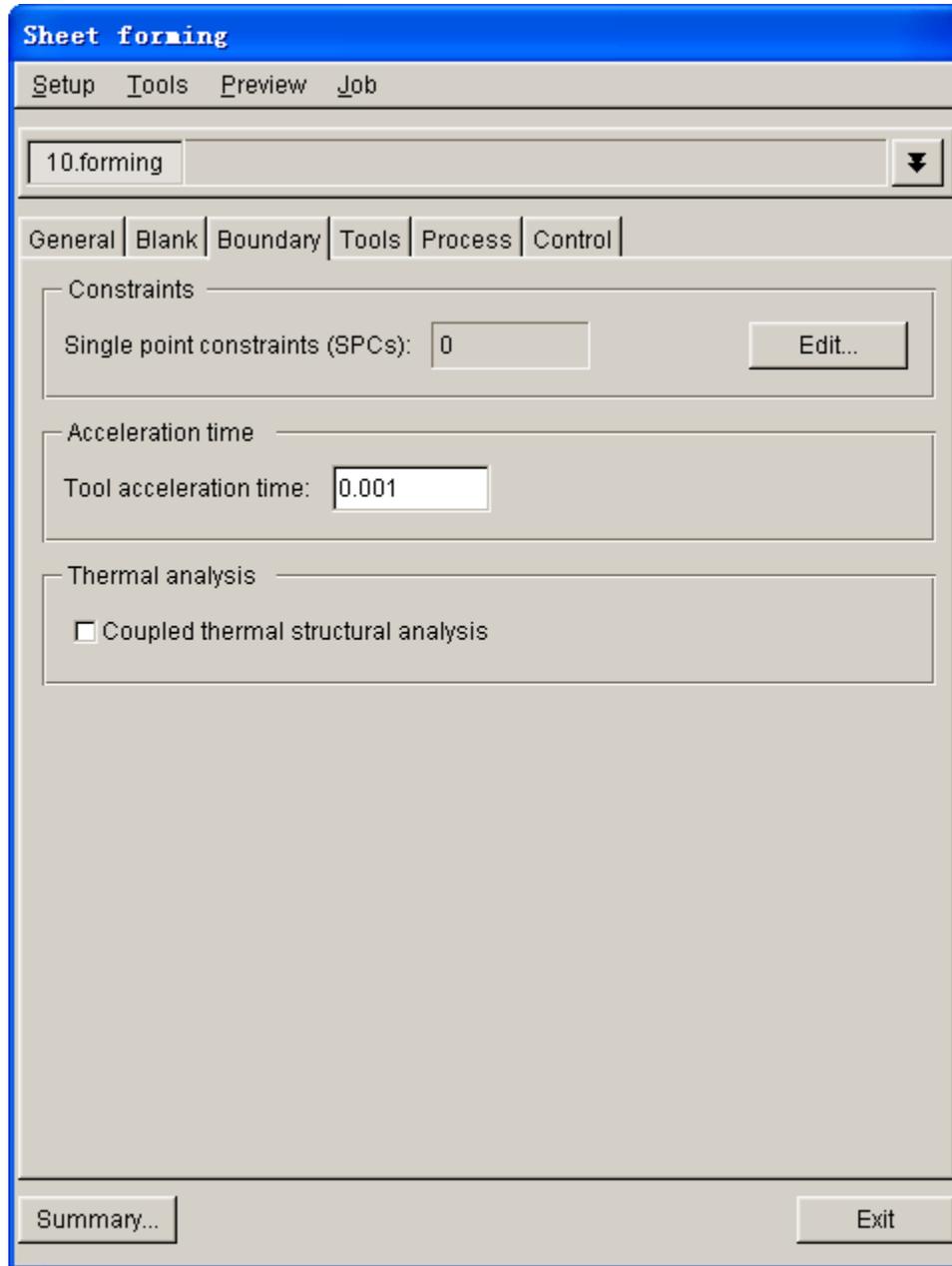


Figure 8.2.46 Boundary Condition page for draw simulation

**SINGLE POINT CONSTRAINTS (SPCs)**

This function allows the user to define single point constraint. Click on EDIT...button to display the dialog box as shown in Figure 8.2.47. The user should select the constraint type prior to clicking the SELECT... button to apply constraint to selected nodes. The node IDs with defined constraint are listed in the box illustrated in Figure 8.2.47. The highlighted node with defined constraint is marked with red label on the display screen.

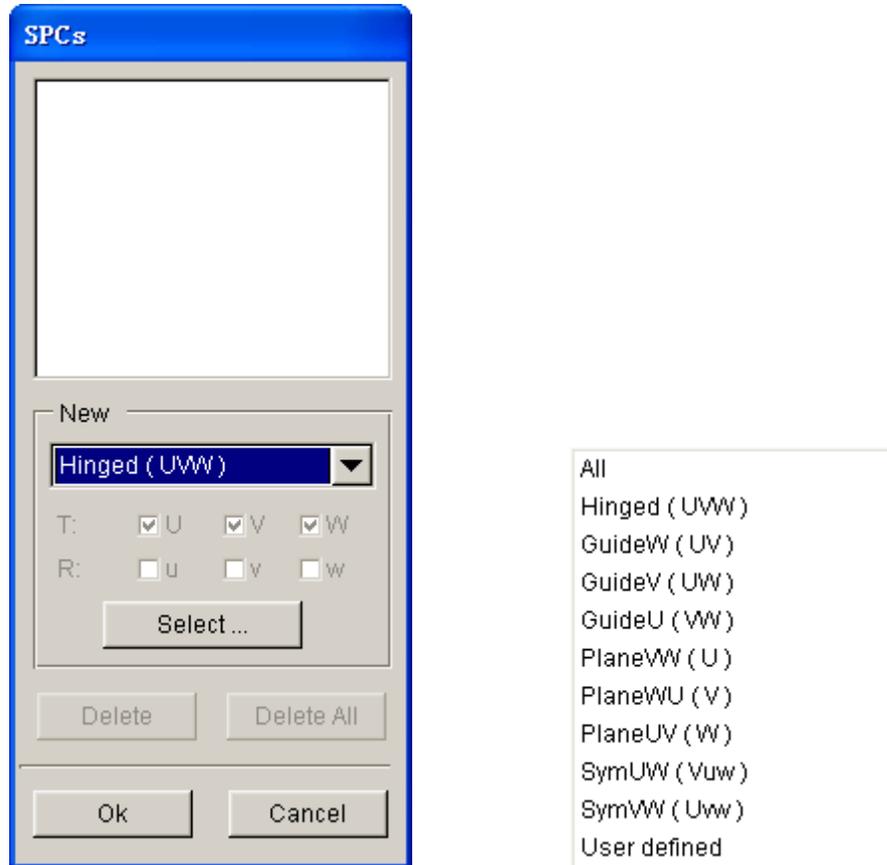


Figure 8.2.47 Single Point Constraints dialog box

The types of constraint include:

<b>All:</b>	Constrain all the degrees of freedom.
<b>Hinged (UWV):</b>	Hinged constraint. Constrain translation of all directions.
<b>GuideW (UV):</b>	Constrain translation along U and V direction.
<b>GuideV (UW):</b>	Constrain translation along U and W direction.
<b>GuideU (VW):</b>	Constrain translation along V and W direction.
<b>PlaneVW (U):</b>	Define part symmetry about VW plane.
<b>PlaneWU (V):</b>	Define part symmetry about WU plane.
<b>PlaneUV (W):</b>	Define part symmetry about UV plane.
<b>User defined:</b>	Allow the user to customize constraints.

If User defined option is selected, the user can toggle on any checkbox to define translation and/or rotation constraints. U, V and W indicate translation constraints in X, Y and Z, respectively. u, v and w indicate rotation constraints in X, Y and Z, respectively.

<b>T:</b>	Translation constraints.
<b>R:</b>	Rotation constraints.
<b>SELECT...:</b>	Select nodes.
<b>DELETE:</b>	Delete selected constraints.
<b>DELETE ALL:</b>	Delete all constraints.
<b>OK:</b>	Accept defined constraints and exit SPCs dialog box.
<b>CANCEL:</b>	Reject defined constraints and exit SPCs dialog box.

In setup of multi stage forming simulation, the user may translate and/or rotate blank in the subsequent stage using the Transformation function illustrated in Figure 8.2.48. It is recommended to perform rotation operation prior to the translation operation.

**ADD...:** Add translation and/or rotation operations.

**EDIT...:** Edit the defined translation and/or rotation.

**DELETE:** Remove defined translation and/or rotation.

**TRANSLATION:** Define distance and direction of translation. The user may define translation using one of the following approaches:

- Enter the distance along U, V and W directions. The direction of translation and distance is automatically calculated according to the input. If **DIRECTION** button is hit, the direction of translation is displayed in the dialog box and marked with an arrow in the display screen. The distance is displayed in the **DISTANCE** field.
- Click on **DIRECTION** button to define direction of translation. Then, exit the **DIRECTION** dialog box and key in the desired distance in the data input field of **DISTANCE**. The direction of translation and distance is automatically calculated according to the input.
- Click on **2 Points/Nodes** button, following by selecting two points/nodes from the display screen. The direction of translation and distance is automatically calculated according to the input.

**ROTATION:** Define rotation axis and angle. Rotate the model about any axis of current coordinate system with any input angle.

**LCS2LCS:** Conversion of local coordinate system. The user is allowed to convert the model from one coordinate system to another. The translation and rotation information automatically output.

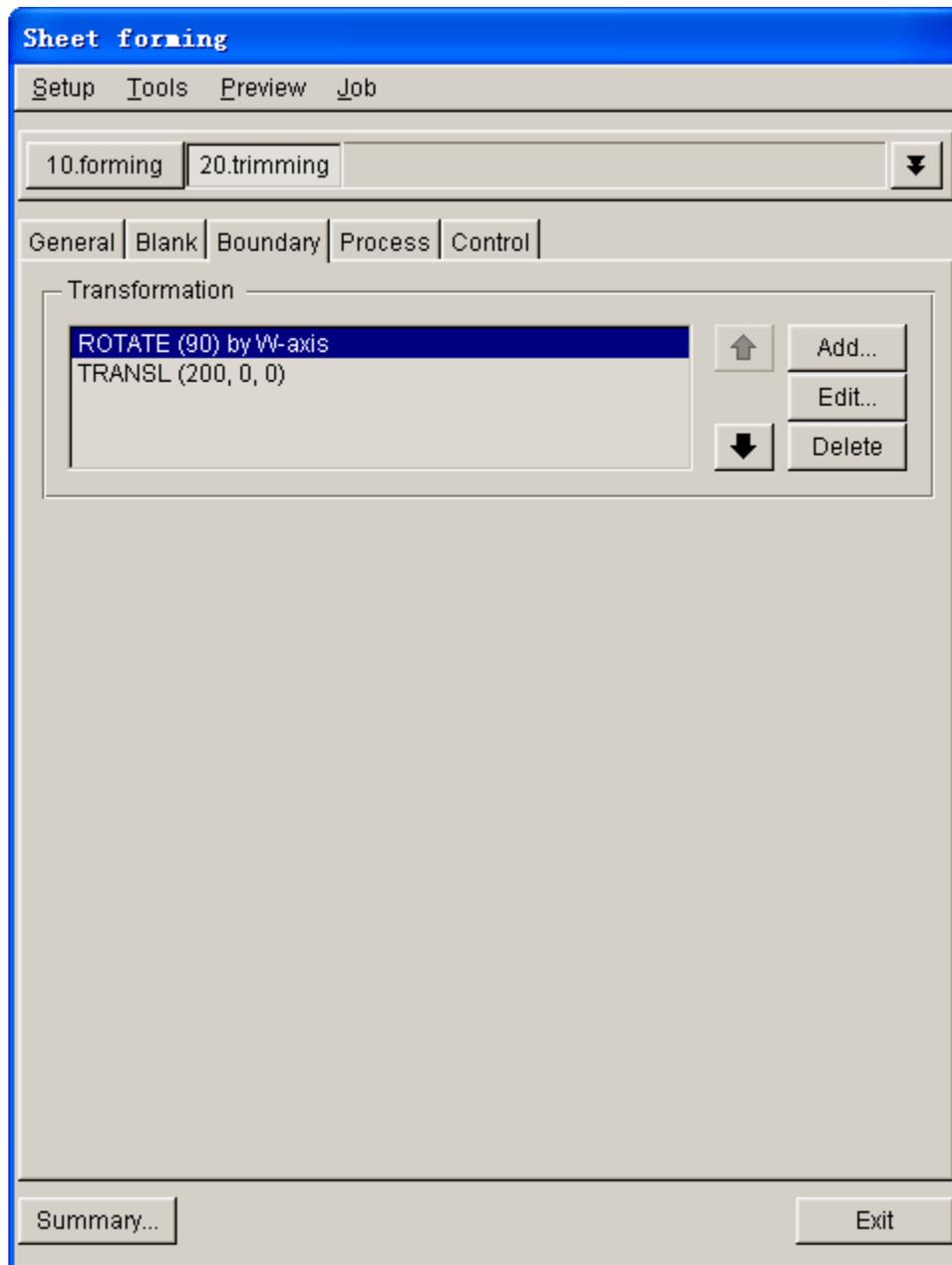
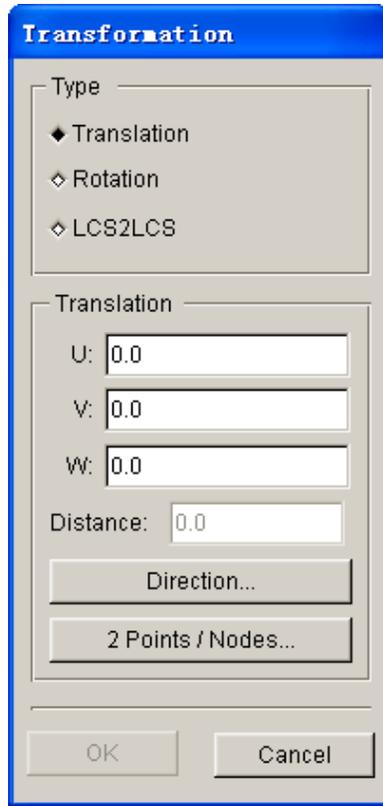
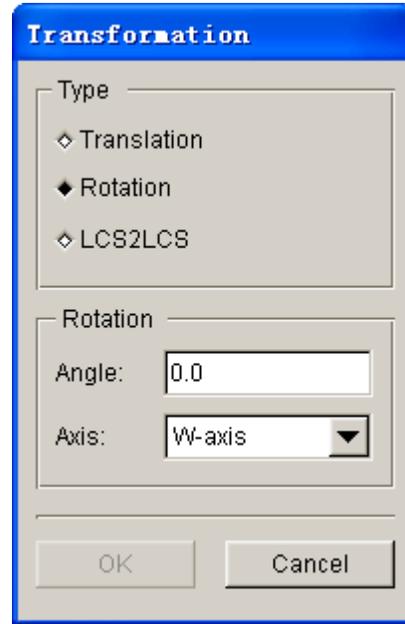


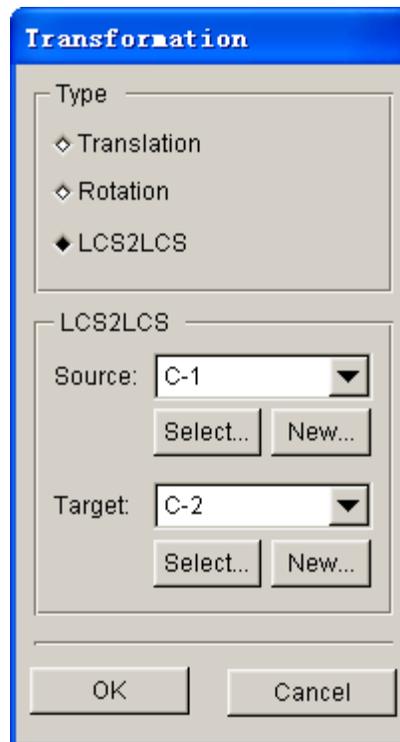
Figure 8.2.48 Define Boundary Condition



a) Define Translation



b) Define Rotation



c) LCS2LCS

Figure 8.2.49 Transformation dialog box

## TOOL ACCELERATION TIME

This function allows the user to specify the tool acceleration/ramp time from 0 to peak value. The ramp time should be defined to guarantee the stability of calculation. Default ramp time is 0.001 second, as shown in Figure 8.2.50.

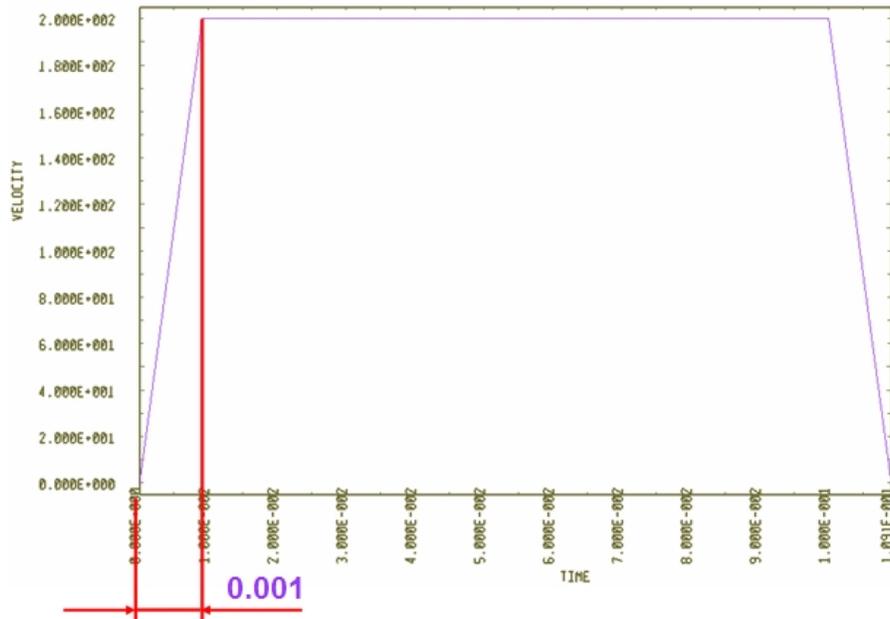


Figure 8.2.50 Acceleration/ramp time

## THERMAL ANALYSIS

If COUPLED THERMAL STRUCTURAL ANALYSIS option is selected, the ADVANCED... button (this option is toggled off by default) is toggled on. Click on ADVANCED... button to display THERMAL ANALYSIS dialog box illustrated in Figure 8.2.51. Refer to LS-DYNA User's Manual for detailed description about parameters for coupled structure-thermal analysis.

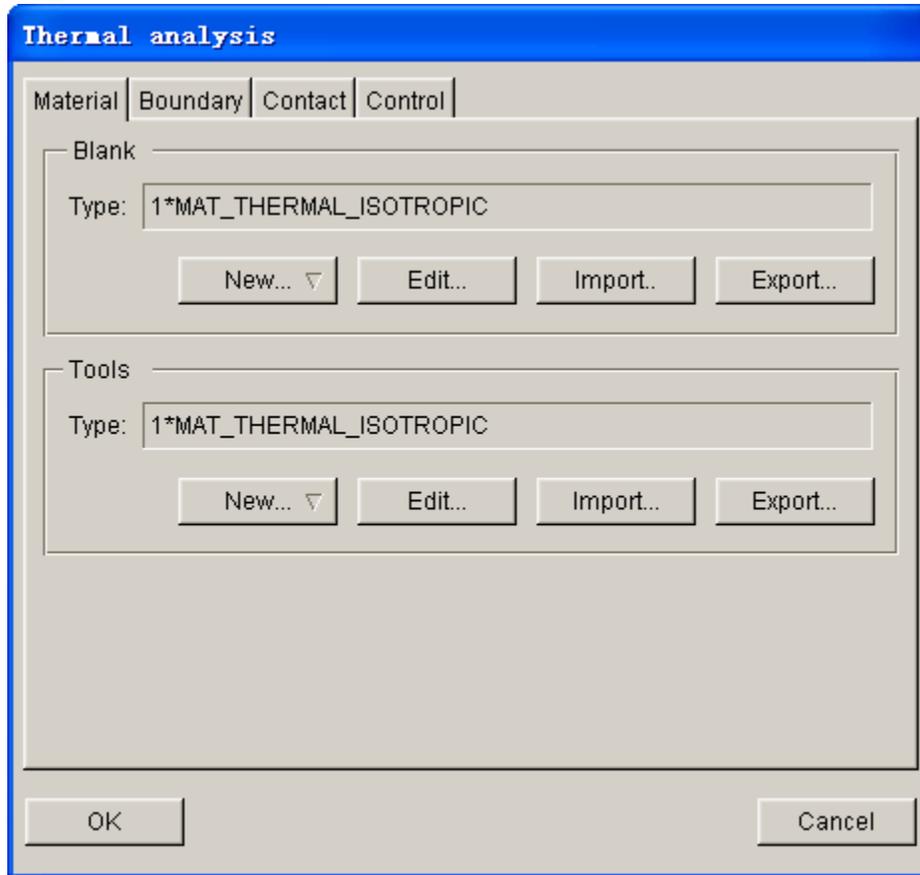


Figure 8.2.51 Thermal Analysis interface

The Thermal Analysis interface enables the user to define parameters such as Material, Boundary, Contact and Control parameters, which are used to perform the steady and transient coupled structure-thermal analysis.

### Material

In coupled structure-thermal analysis, the user must define thermal properties of sheet material, as well as tools.

**NEW:** Click to select a thermal material model from the list illustrated in Figure 8.2.52. After the material is selected, the user may define the thermal material properties in the THERMAL MATERIAL dialog box illustrated in Figure 8.2.53.

```

1*MAT_THERMAL_ISOTROPIC
2*MAT_THERMAL_ORTHOTROPIC
3*MAT_THERMAL_ISOTROPIC_TD
4*MAT_THERMAL_ORTHOTROPIC_TD
5*MAT_THERMAL_ISOTROPIC_PHASE_CHANGE
6*MAT_THERMAL_ISOTROPIC_TD_LC

```

Figure 8.2.52 Thermal material models

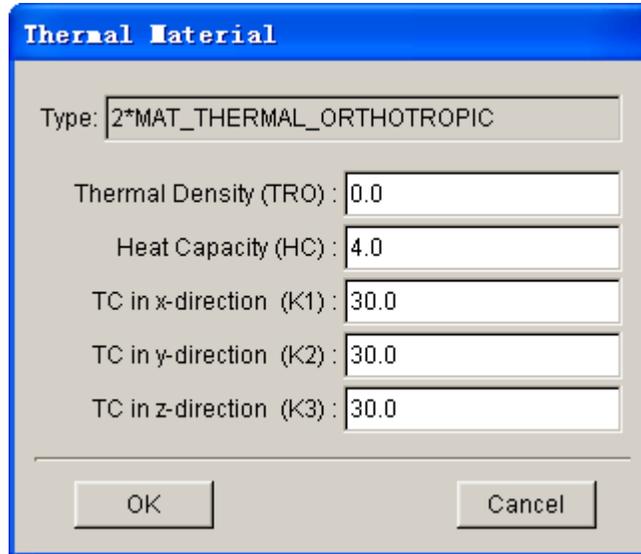


Figure 8.2.53 Thermal Material dialog box

- EDIT:** Edit defined material parameters.
- IMPORT:** Enable the user to import material from any material file with “.mat” extension. Select a material file, following by clicking OK button to close the IMPORT MATERIAL dialog box.
- EXPORT:** Enable the user to save any defined material in the file with “.mat” extension. Click on Export...button to display the EXPORT MATERIAL dialog box.

### Boundary

This page enables the user to define thermal boundary conditions such as initial temperature (blank and tools), radiation, convection and final temperature, as shown in Figure 8.2.54.

- BLANK TEMPERATURE:** Set initial temperature for blank.
- TOOLS TEMPERATURE:** Set initial temperature for tools.
- RADIATION FACTOR:** Thermal radiation factor.
- CURVE MULTIPLIER:** Curve multiplier. Click on the button to define a curve. The number of points on curve is displayed on the button.
- HEAT TRANSFER COEF.:** Coefficient of heat transfer.
- FINAL TEMPERATURE:** Final temperature.

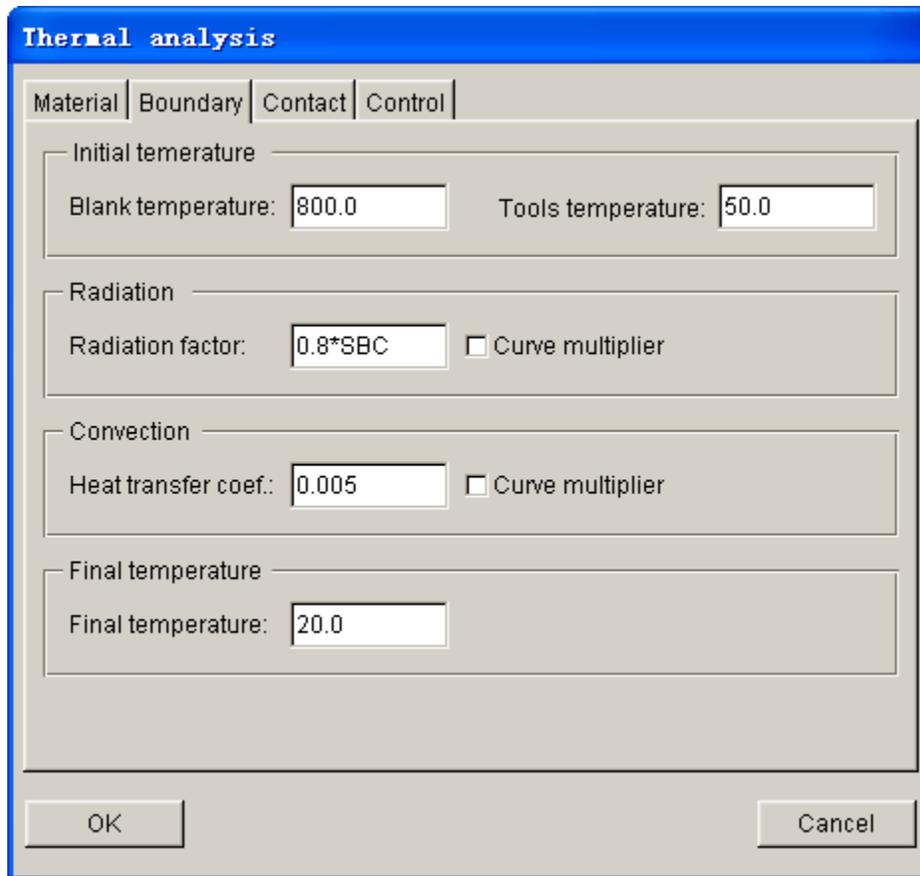


Figure 8.2.54 Boundary page

### Contact

Contact page enables definition of thermal contact and friction, as shown in Figure 8.2.55.

<b>THERMAL CONDUCTIVITY:</b>	Thermal conductivity between gaps.
<b>RADIATION FACTOR:</b>	Thermal radiation factor.
<b>HEAT TRANSFER COEF.:</b>	Coefficient of heat transfer for closed ring.
<b>MIN. GAP:</b>	The minimum gap. If the gap is smaller than MIN. GAP, use htc (heat transfer coefficient) value.
<b>MAX. GAP:</b>	The maximum gap. If the gap is greater than MAX. GAP, thermal contact does not occur.
<b>CONTACT ALGORITHM:</b>	Thermal contact algorithm.
<b>THERMAL BOUNDARY CONDITIONS ARE OFF WHEN PARTS ARE IN CONTACT.</b>	
<b>THERMAL FRICTION:</b>	Thermal friction factor. Curve defining the relation between dynamic/static coefficient of friction and temperature.

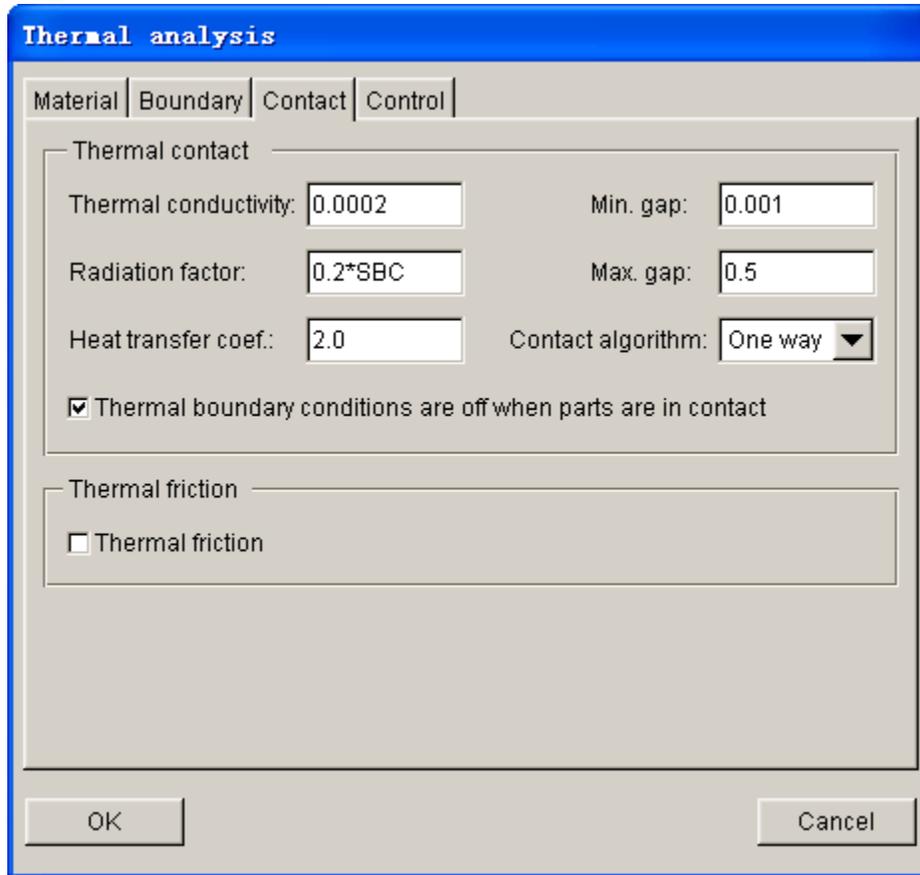


Figure 8.2.55 Contact page

### Control

Control page illustrated in Figure 8.2.56 enables the user to define thermal control parameters. For thermal or coupled structure-thermal analysis, these control parameters are used to control the type of solver for steady or transient analysis, linear or nonlinear analysis and time step size for thermal analysis.

#### **THERMAL ANALYSIS TYPE (ATYPE):**

- 0: steady analysis
- 1: transient analysis

#### **THERMAL PROBLEM TYPE (PTYPE):**

- 0: linear problem
- 1 or 2: nonlinear problem. If chosen, please define  
\*CONTROL\_THERMAL\_NONLINEAR

#### **TIME STEP CONTROL (TS):**

- 0: fixed time step
- 1: variable time step

#### **TIME INTEGRATION PARAMETER (TIP):**

- 0: set to 0.5 and use Crank-Nicholson scheme

1: use fully-implicit scheme

**MAX. MATRIX REFORMATIONS (REFMAX)**

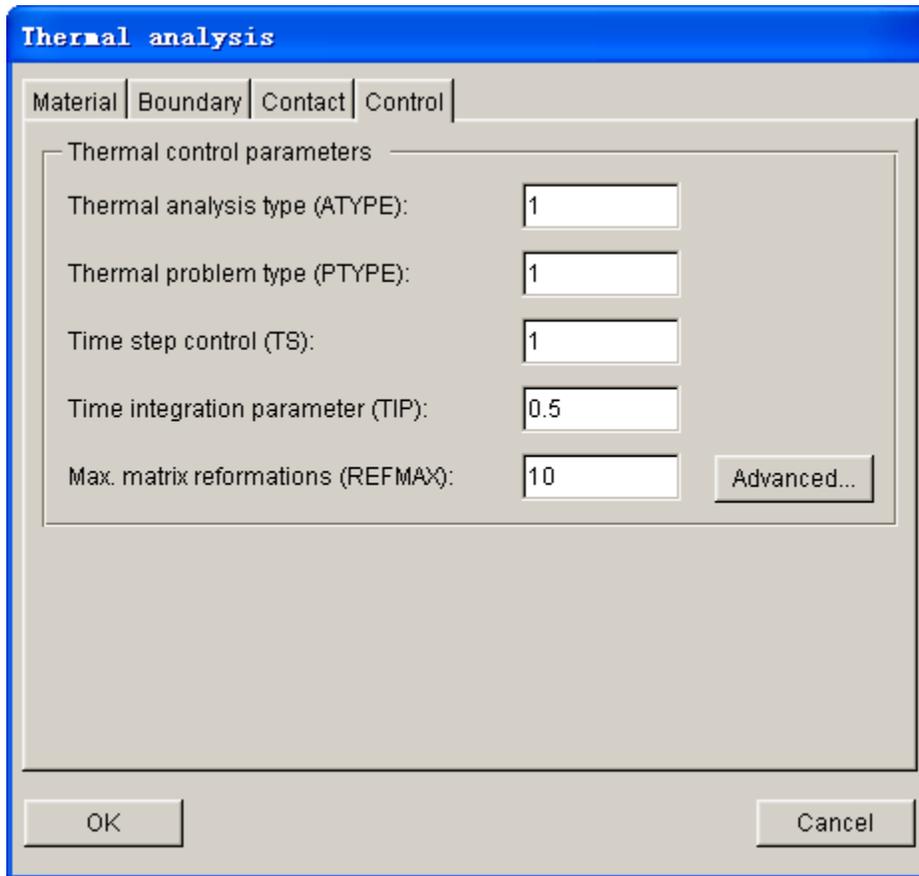


Figure 8.2.56 Control page

**8.2.4.4 TOOLS**

In the Tools definition page, the user may add and remove tools, adjust position between blank and tools, define tool properties, working direction and contact between blank and tools. The interface is illustrated in Figure 8.2.57.

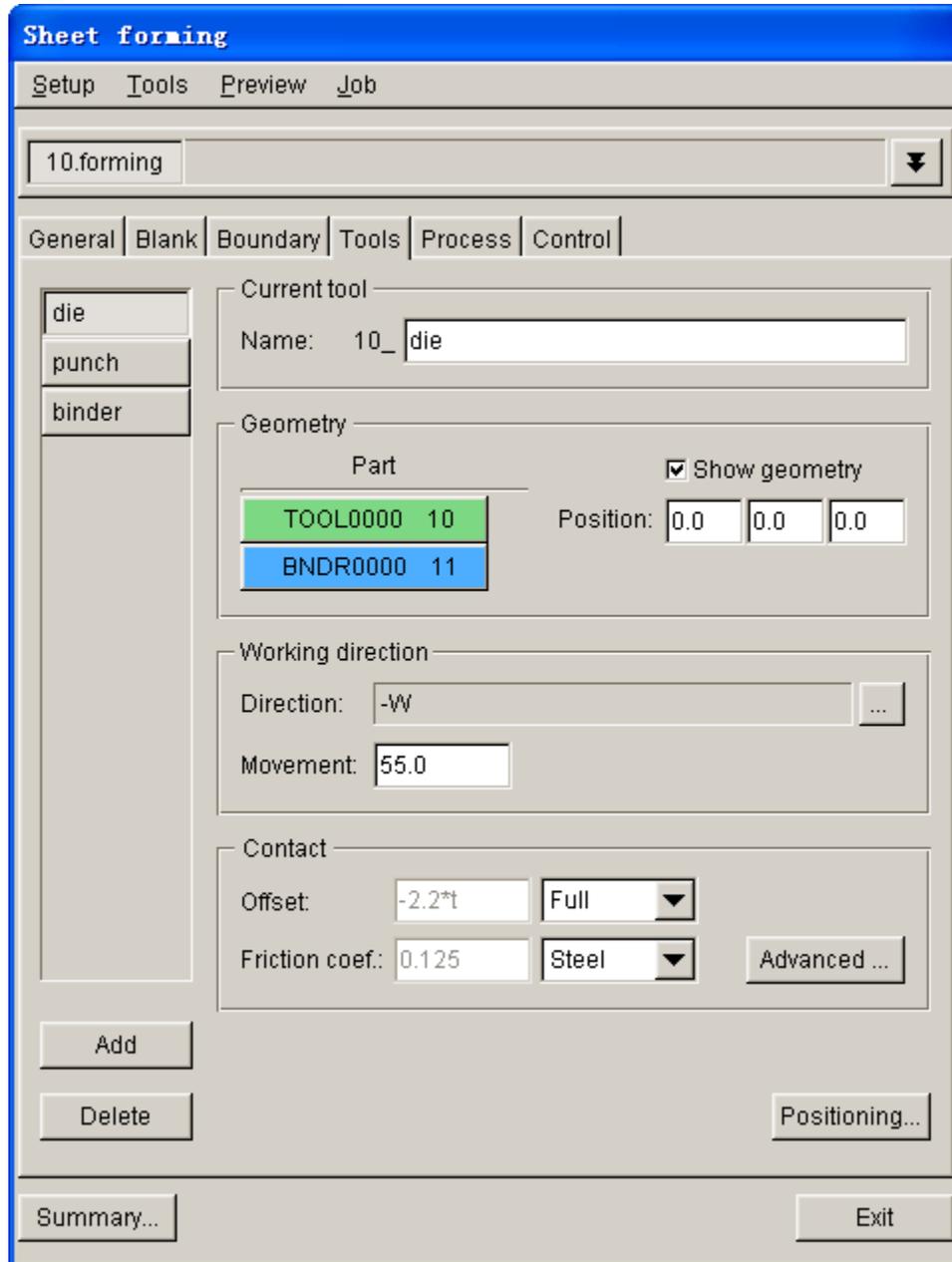


Figure 8.2.57 Tools definition page

## TOOL LIST

The list of tool names is displayed in the box on left hand of Tools definition page. The indented tool name button indicates the tool is working tool. In Auto Setup, default tools are set according to the selected process type. For example, if SINGLE ACTION process is chosen, default tools consist of die, punch and binder. Each basic parameter is set with default value, except for tool part, working direction, contact offset, etc. The user may directly define each tool from the menu page to saves time for defining tools.

**NOTE:** Undefined tool is labeled in red. In each setup, undefined parameters are labeled in red to remind the user that the setup is incomplete.

## ADD TOOL

The ADD TOOL function allows the user to create a new tool in current setup. The user may click ADD button to switch the interface to the new tool interface illustrated in Figure 8.2.58.

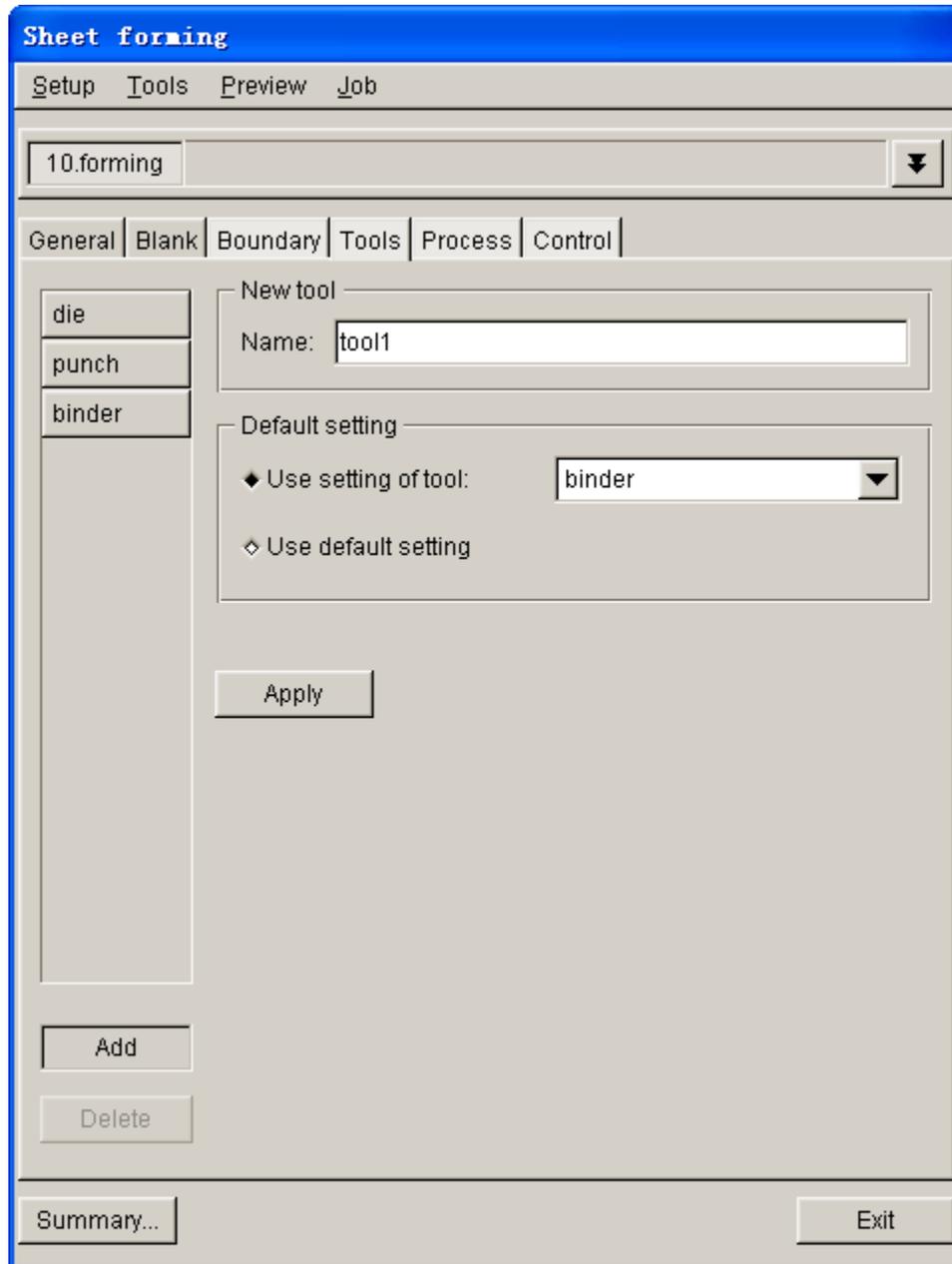


Figure 8.2.58 New Tool interface

## TOOL NAME

Allow the user to define a name for current tool. Default tool name is tool1, tool2, tool3, etc. The user may define a meaningful tool name as current tool for the ease of identifying each tool in the stamping simulation.

## DEFAULT SETTING

Allow the user to set default setting for current tool such as working direction, offset, etc. There are two options in default setting. One is using existing tool as reference, which newly generated tool adopted the basic parameters of the reference tool. The other one is to set the default setting of basic parameters of the newly generated tool.

**USER SETTING OF TOOL:** New tool adopts basic parameters of an existing tool. The user may select an existing tool from the pull-down list.

**USER DEFAULT SETTING:** Use default parameters 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.

## APPLY

Enable the user to generate new tool and list it in the tool list.

## DELETE TOOL

This function allows the user to remove current tool from the setup. The user may select any tool from tool list as current tool, following by clicking the DELETE button. The DELETE TOOL dialog box illustrated in Figure 8.2.59 is displayed.

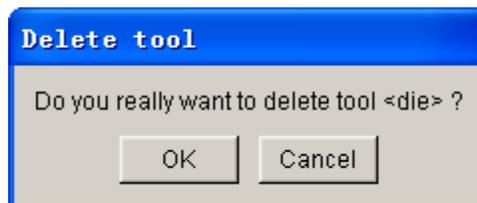


Figure 8.2.59 Delete Tool dialog box

Click OK to delete the current tool, or Cancel to quit the operation.

## TOOL NAME

This function allows the user to modify current tool name. When creating a tool, the user may arbitrary assign a name for the tool. After the tool is generated, the user may edit the tool name listed in Tools definition page. In the input field of NAME, the user may key in new tool name and press "Enter [CR]" to complete the operation.

*Note: Each tool must have a unique name. If the newly created tool has the same name with an existing tool in the setup, numerical number will be added at the end of new tool name, e.g. die 1, die 2, punch 1, punch 2, etc.*

## TOOL GEOMETRY

This function allows the user to assign parts in current tool. The user may add one or more parts in current tool. Click Define geometry button to display the DEFINE GEOMETRY dialog box illustrated in Figure 8.2.60.

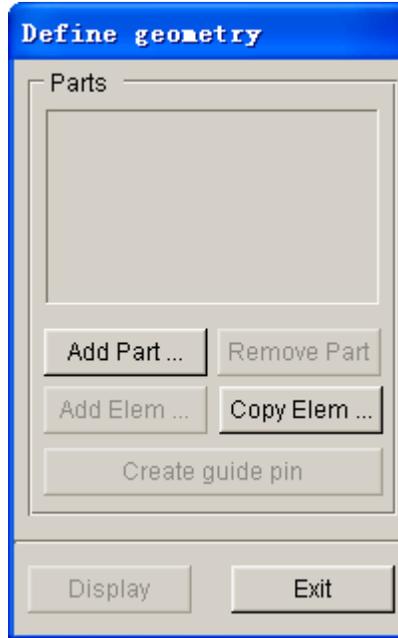


Figure 8.2.60 Tool Define Geometry dialog box

**ADD PART...:** This function allows the user to select one or more parts from part list as tool. The selected parts are displayed in Parts List window. Refer to 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 element(s) from existing elements. If only one die face is defined, the user may COPY or OFFSET tools (punch and binder) using this function. The COPY ELEMENTS dialog box illustrated in Figure 8.2.61 is displayed.

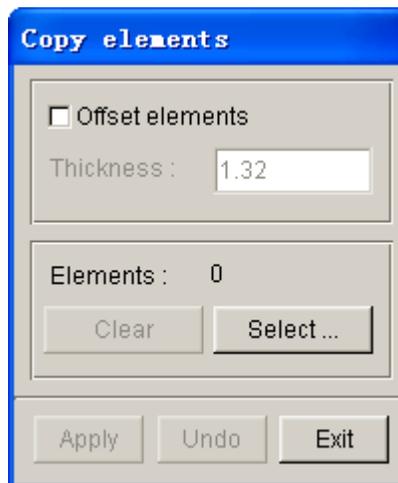


Figure 8.2.61 Copy Elements dialog box

**OFFSET ELEMENTS:** This function allows the user to offset new elements from existing elements and include these elements into the new part in current tool. The default name of new part is OFFSET01, OFFSET02, etc.. Usually, this option is toggled off, indicating only copy element operation is chosen.

**THICKNESS:** Element offset thickness. Default thickness is 1.0 (mm). The user may key in desired offset distance. The user may reverse offset direction by setting a negative offset thickness, such as -1.0.

**SELECT...:** This function allows the user to select element(s) for copy or offset operation. If the button is hit, the SELECT ELEMENTS dialog box is displayed.

**CLEAR:** This function allows the user to clear selected elements. The user may use SELECT button to re-select elements.

**APPLY:** After selecting elements, the user may click APPLY to perform copy or offset operation.

**UNDO:** The user may click UNDO to cancel the last operation.

**EXIT:** Exit COPY ELEMENT dialog box.

**CREATE GUIDE/PIN:** This function allows the user to create GUIDE and/or PIN. A new part for the GUIDE is created and included as current tool. Default name of new GUIDE is ACC0000, ACC0001, etc.. For detailed description of GUIDE and PIN, refer to Section 9.6.

**DISPLAY:** This function allows the user to display parts that are corresponding to the current tool on the display screen. After tool geometry is defined, parts corresponding to this tool are displayed in the part list.

## POSITIONING

This function enables the user to position tool by entering the travel distance between tool and HOME, as shown in Figure 8.2.62. Show geometry button is used to show/hide the display of the current tool.



Figure 8.2.62 Tool Positioning

## WORKING DIRECTION

In each setup, a unique working direction of tool has to be defined. Working direction is the direction of tool traveling towards the blank. Note: Working direction is a vector originated from the tool and pointing towards the blank. It is not the actual direction of tool travel. The vector can be pointing to the opposite direction of direction of tool travel. The interface of working direction is illustrated in Figure 8.2.63.

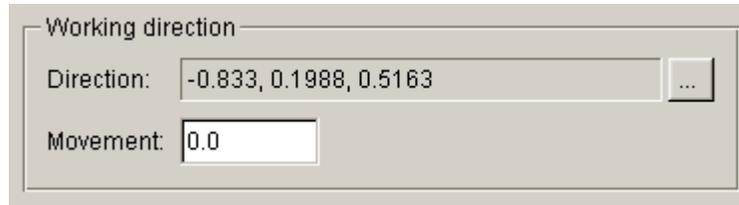


Figure 8.2.63 Define Working Direction

**DIRECTION:** This function displays the working direction of current tool 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 modify the working direction using the DEFINE DIRECTION dialog box illustrated in Figure 8.2.64.

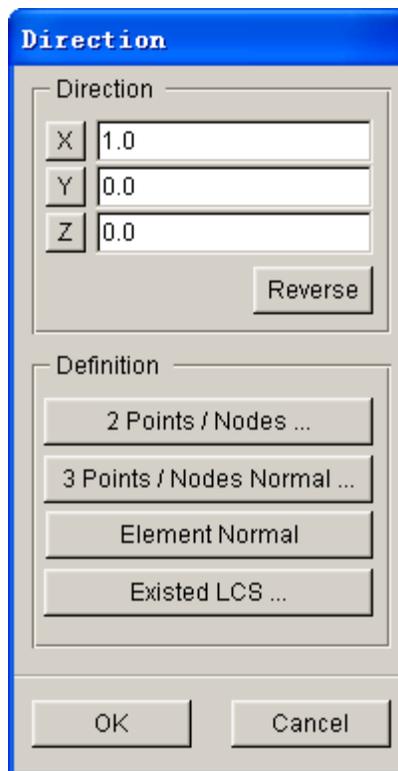


Figure 8.2.64 Define Tool Working Direction of Tool

**DIRECTION**

This function allows the user to determine direction using defined vector. The origin of the vector is similar to the origin of current coordinate system. The coordinate of end point is determined by the user.

 — Define X coordinate for the endpoint of vector.

 — Define Y coordinate for the endpoint of vector.

 — Define Z coordinate for the endpoint of vector.

### REVERSE

This function allows the user to reverse the defined direction.

### 2-POINTS/NODES

This function allows the user to select two points or nodes to define a direction.

### 3-POINTS/NODES

This function allows the user to select three points or nodes to define a direction. The defined direction is perpendicular to the plane defined by the three selected points/nodes. The direction follows the right hand rule.

### ELEMENT NORMAL

This function allows the user to select an element. The normal direction of selected element is defined as the desired direction.

### EXISTED LCS

This function allows the user to select a coordinate axis from the defined LCS as the desired direction. If no LCS is defined in the database, this button is disabled. After hitting this button, the dialog box illustrated in Figure 8.2.65 is displayed. The user proceeds to select a LCS and direction for CS.

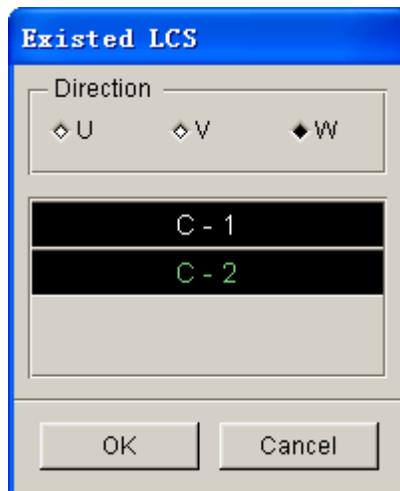


Figure 8.2.65 Existed LCS dialog box

**MOVE:** This function is used to displays the tool travel distance along the working direction from tool's original position after positioning operation. If negative value is observed, the tool moves in the opposite direction of working direction. The user can directly enter the travel distance to adjust tool.

### CONTACT

This function allows the user to define contact parameter between the blank and tools.

**OFFSET:** When defined tools are copied from the die face, contact offset is required for corresponding tools. For example, if the new punch is generated according to die face, an offset value is required for the corresponding punch. The blank thickness is used as the offset value. The user may change the value according to requirement.

**FRICITION COEF:** Allow the user to specify coefficient of friction.

**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 both auto and manual positioning illustrated in Figure 8.2.66. The user may select the checkbox of OnBlank option to activate auto positioning. Alternatively, the user may key in travel distance of the tool in the input box for Tools to perform manual positioning. After entering the travel distance of blank and/or tools, the user clicks ENTER key to update the position of blank and tools in display screen.

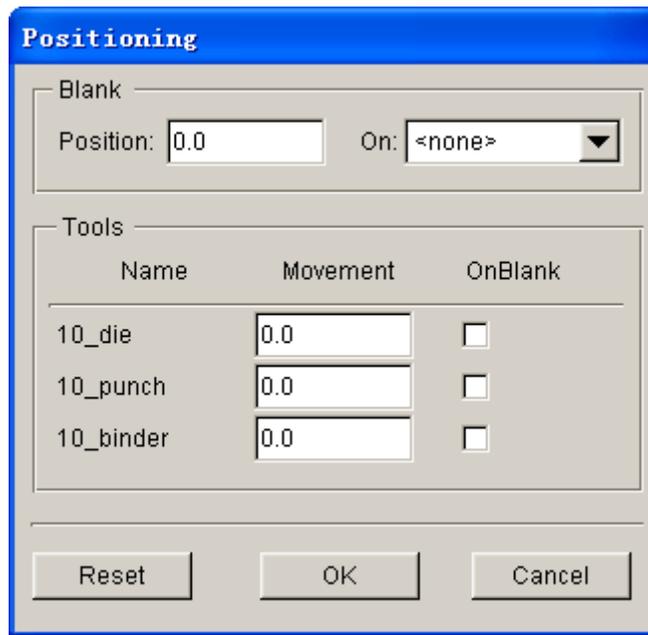


Figure 8.2.66 Tool Positioning

**BLANK POSITION:** The user may key in any value in the input data field to position the blank. The user may also select a tool from the drop-down list as reference for positioning of blank, and show the travel distance of the blank relatively to its original position. If the latter option is chosen, the checkbox of OnBlank option for the selected tool is disabled.

**TOOL MOVEMENT:** All the defined tools are listed in the Tools field. The user may define the initial position of tooling by entering a value for each tool in the input data field. If the checkbox of OnBlank option for the selected tool is toggled on, the tool position is automatically calculated according to the blank position. The movement of tool relatively to its original position is listed in input data field. Modification of this value is prohibited.

**NOTE:** Auto positioning of tools and blank are carried out along the working direction. The user must specify working direction for each tool prior to performing auto-positioning.

**RESET:** Clear all the travel distances between tools and blanks and reset to the initial position prior to positioning operation.

**OK:** Save result of positioning and exit the POSITIONING dialog box.

**EXIT:** Exit the POSITIONING dialog box without saving positioning result.

### 8.2.4.5 PROCESS

The PROCESS definition page includes the basic setup of process parameters. It includes stamping speed, blank holding force, forming time, etc. The user may change process parameters according to the actual stamping condition. Moreover, the user may add and delete any process. The basic process setup interface is shown in Figure 8.2.67.

The screenshot shows the 'Sheet forming' software interface. The window title is 'Sheet forming'. The menu bar includes 'Setup', 'Tools', 'Preview', and 'Job'. A dropdown menu shows '10.forming'. The 'Process' tab is active, showing a list of steps on the left with 'closing' selected. The main area is divided into sections: 'Current step' (Name: 10\_closing, Hydro checkbox), 'Tool control' (Tools: die, punch, binder; Action & Value: Velocity 2000.0, Stationary, Stationary; Show all checkbox), 'Duration' (Type: Closure, Fully match checkbox, Tools: die, binder, Gap: 0.0), and 'D3plot' (Frames: 5, Advanced... button). At the bottom are 'Add', 'Delete', 'Summary...', and 'Exit' buttons.

Figure 8.2.67 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. Default process is specified according to defined process type. For example, if SINGLE ACTION process is chosen, closing and drawing are created as default process. There are also default settings for the basic parameters such as stamping speed, blank holding force, stamping time, etc. Usually, few parameters are modified.

### ADD PROCESS

This function allows the user to add a new process in current setup. The user may click ADD button to display the new process interface illustrated in Figure 8.2.68.

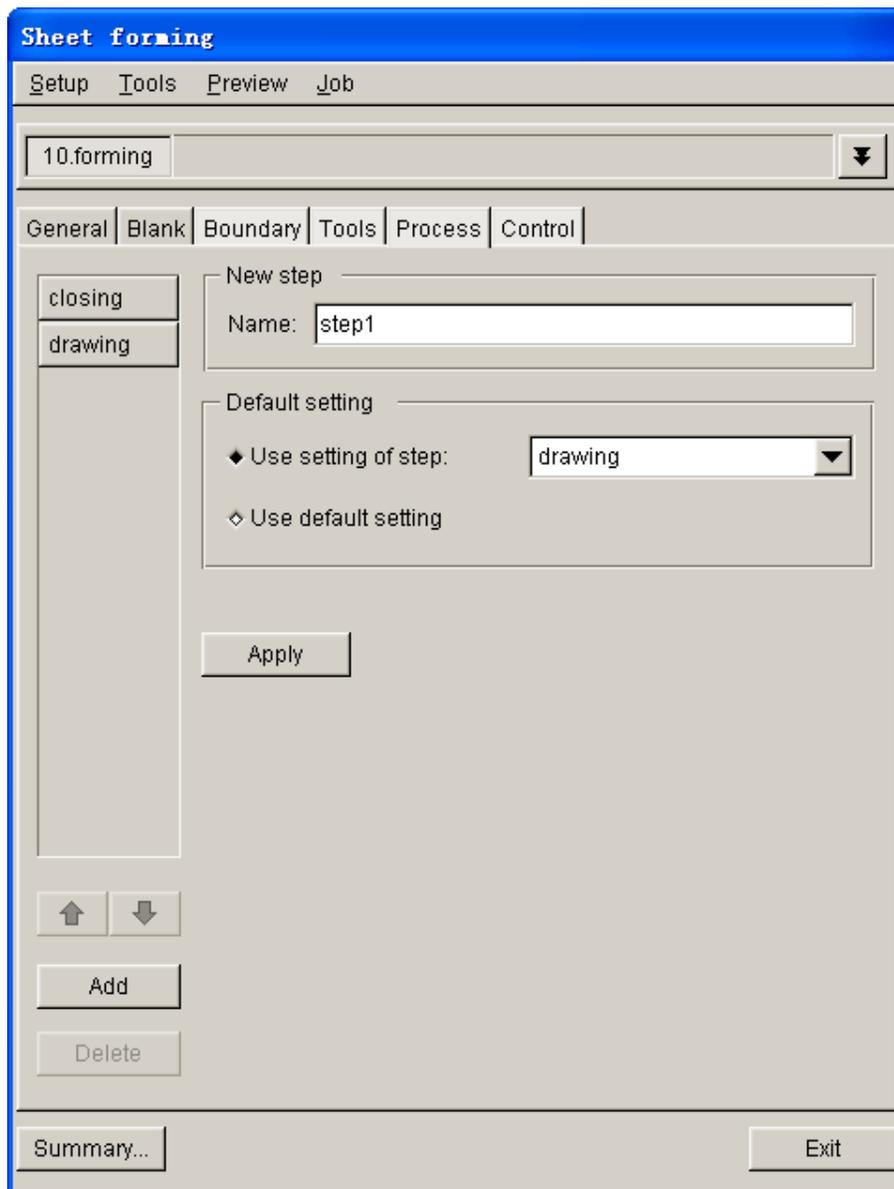


Figure 8.2.68 New process interface

## PROCESS NAME

Allow the user to name the current process. Default process names are step1, step2, step3, etc.. The user may define a unique name for each process for the ease of identifying the process.

## DEFAULT SETTING

Allow the user to select default setting for the new process. In default setting, the user may select the program default parameters for the new process. Otherwise, the user may choose to use the parameters of any defined process.

**USE SETTING OF STEP:** New process adopts parameters of existing process. The user may select an existing process from pull-down list.

**USE DEFAULT SETTING:** Use system default parameters to define a new process.

## DELETE PROCESS

This function allows the user to remove current process from setup. The user may select a process from the process list as current process, following by clicking DELETE button to display the dialog box illustrated in Figure 8.2.69. Click on OK button to delete the current process or Cancel button to abort the delete operation.

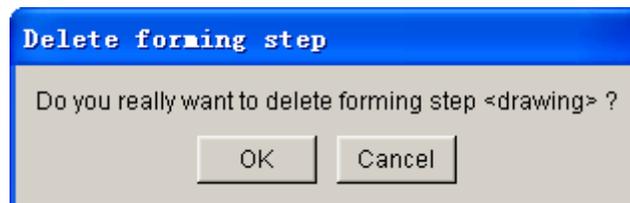


Figure 8.2.69 Delete process dialog box



MOVE STEP

This function allows the user to organize sequence of processes.

## PROCESS NAME & TYPE

### NAME

Display name of current process. The user may type in a new name in the input data field of Name.

### TYPE

Allow the user to select type of current process: Drawing or Hydroforming.

## TOOL CONTROL

This function is used to control action of tools define travel speed, blank holding force, travel curve, etc. in current process. A typical tool control option is illustrated in Figure 8.2.70.

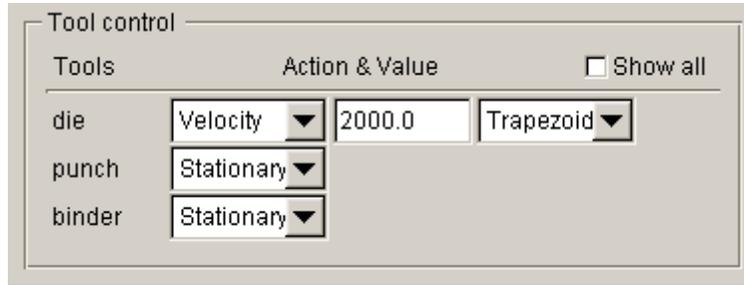


Figure 8.2.70 Tool Control Option

**TOOLS**

List all active tools of current process. By default, the non-active tools in the current process are not listed. The user may toggle on checkbox of SHOW ALL option to display all tools.

**SHOW ALL**

Display all defined tools in tool list.

**ACTION & VALUE**

This group lists all tool control options. Each tool can be controlled by one of the following option: NON-ACTIVE, STATIONARY, VELOCITY, DISPLACEMENT and FORCE.

**NON-ACTIVE**

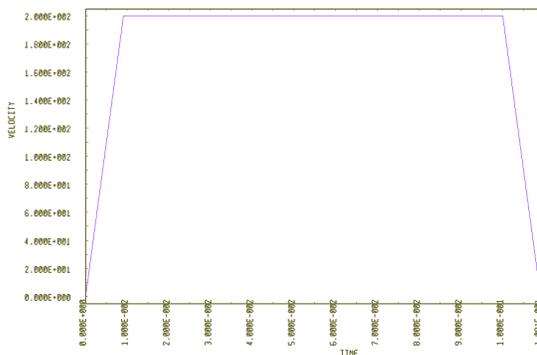
Defined tool is not used in this stage.

**STATIONARY**

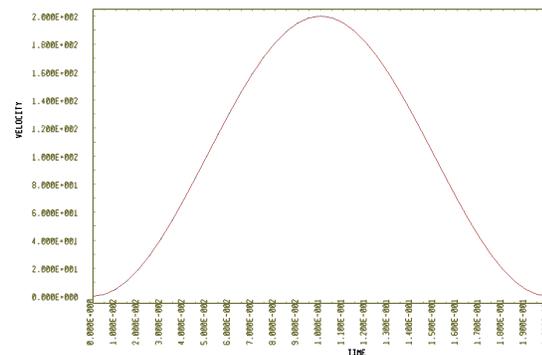
Defined tool is stationary in this stage.

**VELOCITY**

Motion of defined tool is controlled by velocity curve in this stage. The user may select one of the standard velocity curve option provided in Auto Setup: TRAPEZOIDAL, SINUSOIDAL, SIN.W-H, TRIANGULAR and VARIABLE. The standard velocity curves are illustrated in Figure 8.2.71. The user may type in peak velocity in the data input field of selected tool. Default velocity of die for drawing process is set as 5000 mm/s. For closing process, the default velocity of binder is set as 2000 mm/s.



a) TRAPEZOIDAL



b) SINUSOIDAL

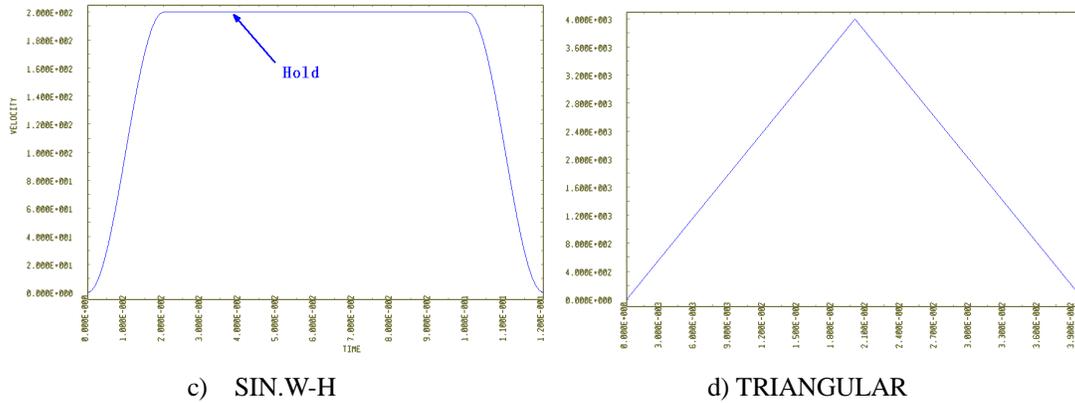


Figure 8.2.71 Types of standard velocity curves

If user-defined curve is chosen, the  button is activated to enable the user to define travel curve using the EDIT LOAD CURVE dialog box.

### DISPLACEMENT

Tool travel of this stage is controlled by displacement-time curve. The user clicks DEFINE button to create displacement-time curve. The procedure is similar to those described in the velocity curve section.

### FORCE

Force is applied to selected tool of current stage in closure and/or drawing process. There are two types of force control: CONSTANT and VARIABLE. If constant force option is chosen, a constant force value is required. For variable force option, the user clicks DEFINE button to create force-time curve.



This button is activated when force control is selected. The user may use this button to define RIGID BODY STOPPER. Rigid body stopper is applied on the binder ring to control upper and lower displacement of binder ring. The user may specify the parameters for rigid body stopper illustrated in Figure 8.2.72.

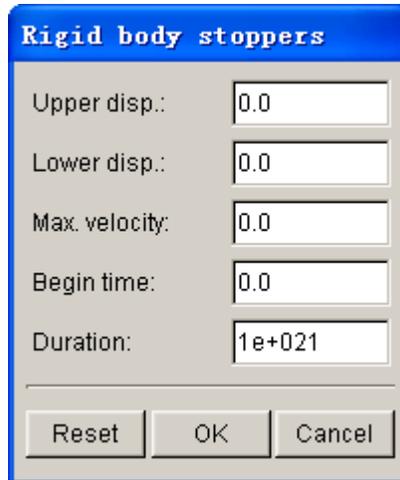


Figure 8.2.72 RIGID BODY STOPPERS dialog box

**UPPER DISP.:** The limit of upper maximum displacement of the tool.

**LOWER DISP.:** The limit of lower maximum displacement of the tool.

**MAX. VELOCITY:** Control the maximum velocity of rigid body stopper during the process.

**BEGIN TIME:** Activate RIGID BODY STOPPER.

**END TIME:** De-activate RIGID BODY STOPPER.

**DURATION:** Duration of RIGID BODY STOPPER.

After setting up the RIGID BODY STOPPER, the label of  button becomes blue. Detailed description of RIGID BODY STOPPER is available in LS-DYNA User's Manual.

#### PRESSURE

Apply pressure on selected tool in the current stage. The defined pressure curve is converted into force curve during output of input decks.

#### ASSOCIATION

Allow the user to define contact between two tools.

#### USER DEFINE CURVE

In each setup, all the defined curves are managed by the curve editor. The user may create, import, modify existing curve using the function provided in the EDIT LOAD CURVE dialog box illustrated in Figure 8.2.73.

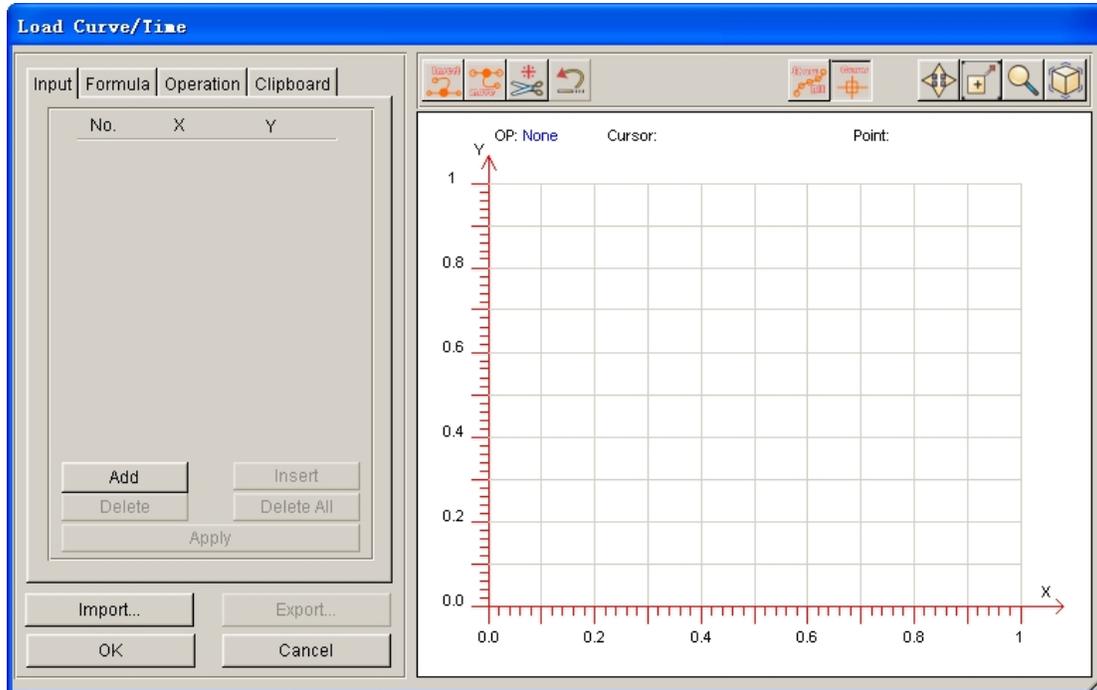


Figure 8.2.73 EDIT LOAD CURVE dialog box

**IMPORT CURVE**

Allow the user to import file consisting curve data. The extension of curve files is “.cur”. The curve file is defined according to the standard LS-DYNA keyword format.

.A typical curve format is shown in Figure 8.2.74.

```

$-----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
    
```

Figure 8.2.74 Typical curve format

**EXPORT CURVE**

Allow the user to export current curve to a text file with “.cur” extension. Click the Export button to display EXPORT CURVE dialog box, following by typing in the name of file. Then, click Save button to store the file in selected folder.

**INPUT**

This page displays data the points of current curve. If no curve is defined, user may create a new curve by entering point coordinates. The user may modify data points of existing curve using function such as add, delete, insert and clear.

**ADD:** Allow the user to add a data point at the end of curve. Click Add button to create two empty input data fields which are highlighted in yellow. Next, the user may key in data points in the input data field, following by hitting Apply button to complete the operation.

**INSERT:** Allow the user to insert a data point after current data point. Prior to clicking this button, the user may use mouse cursor to choose the current data point. The selected data point is highlighted in yellow. Next, click INSERT button to create two empty input data fields, following by typing in the data point and hit Apply button to complete the operation.

**DELETE:** Allow the user to remove current data point.

**CLEAR ALL:** Allow the user to clear all data points in current curve.

**APPLY:** The user may click APPLY button after performing add, insert or delete point operation. The curve is refreshed and displayed in the curve display window.

**FORMULA**

This page allows the user to create curve using mathematic formula such as linearity, quadratic, cubic, and sin (cos). When defining FLC curve, the Keeler’s empirical equation is used.

Figure 8.2.75 Formula Page

**TYPE:** Allow the user to select type of formula used to create a curve. Type of formula includes SIMPLE, QUADRATIC, CUBIC, SIN and COS.

**INTERVAL:** Allow the user to specify interval of abscissa. Default value varies according to type of selected formula.

**ABSCI-START:** Allow the user to specify start point of abscissa. Default value varies according to type of selected formula.

**ABSCI-END:** Allow the user to specify end point of abscissa. Default value varies according to type of selected formula.

**PARAMETER:** Allow the user to input required parameters for each formula. After a formula is chosen, the expression of the formula is displayed in the interface. For example, expression of QUADRATIC formula is:  $y = Ax^3 + Bx^2 + Cx + D$ . To create quadratic curve, the user must define parameters A, B, C and D. By default, these parameters are set to 1.

**APPLY:** After defining parameters, the user may click APPLY button to create curve. The created curves are displayed in curve display window, while data points of the curve are listed in the INPUT page.

### OPERATION

This page allows the user to perform additional operation to the curve, such as SCALE, TRANSLATE, etc.

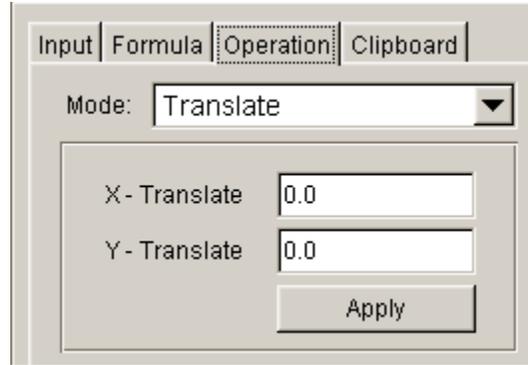


Figure 8.2.76 Curve Operation

**MODE:** Define the curve operation mode. Including SCALE and TRANSLATE.

**PARAMETER:** The user may input curve operation parameters in input data field. For example, the user needs to individually key in the scale factors for abscissa and ordinate to scale or translate a curve.

**APPLY:** After specifying operation parameters, the user may click APPLY button to complete the operation and refreshed the curve displayed 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. The user may also compare the curve in clipboard and current curve. The clipboard interface is illustrated Figure 8.2.77.

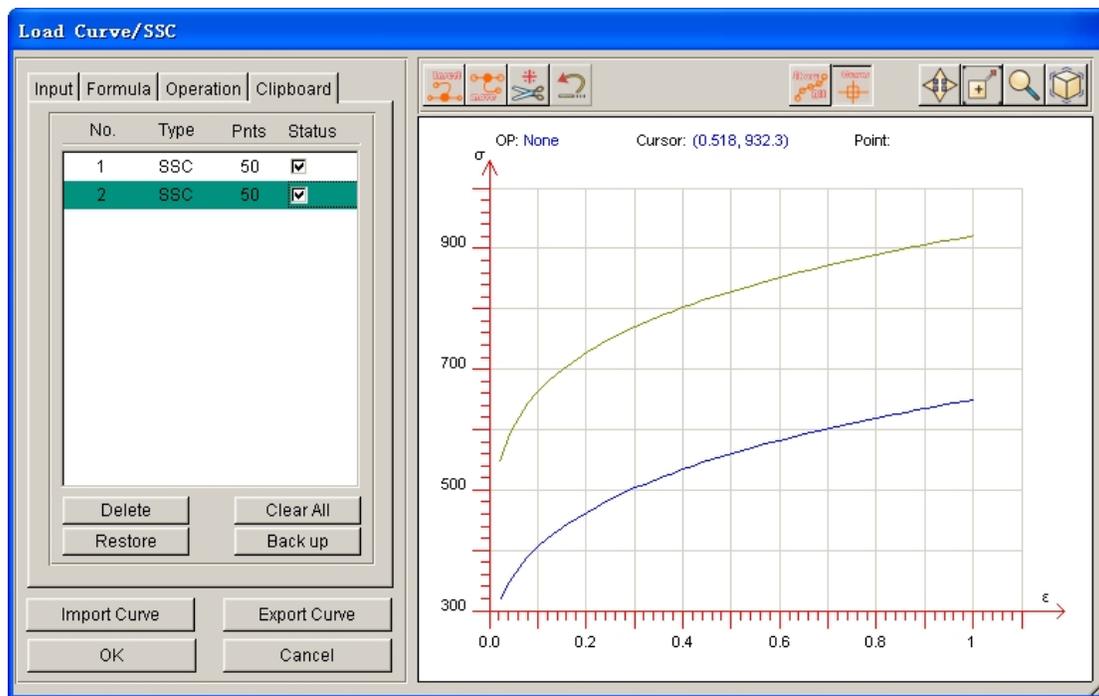


Figure 8.2.77 Clipboard Interface

**DELETE:** Remove selected curves from clipboard.





Figure 8.2.80 Move Points on the Curve

**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 curve display window. Release the mouse button to zoom 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 curve display window. Holds the mouse button and moves the mouse downward to zoom out. Move the mouse upward to zoom in the curve.

**FIT VIEW**  This function allows the user to fit the entire curve in the curve display window.

**DURATION**

This function is used to control the duration of current stage. The duration interface is illustrated in Figure 8.2.81. The user may select one of the four methods to define duration: Time, Travel, Closure and Until Home. Each of these four methods has its own advantages and requirement for different forming stage. The user may select an appropriate control method according to the requirement of the current setup.



Figure 8.2.81 Define duration time

### DURATION TYPE

The user may select a control type from the TYPE pull-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 data field.

**TRAVEL** Allow the user to specify tool travel distance along the working direction. The user needs to select a reference tool from tool list illustrated in Figure 8.2.82. Then, define the displacement of the tool along its working direction. The time of travel is automatically computed by Auto Setup.

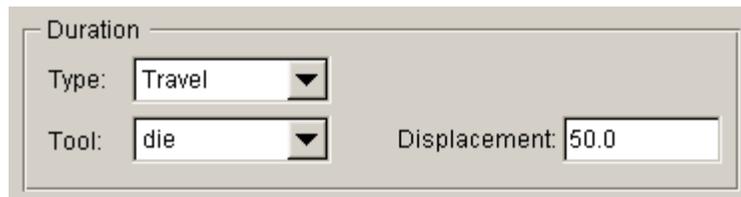


Figure 8.2.82 Travel control stage time

**NOTE:** The working direction can be acting in either negative or positive direction of the axis. 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, working direction should be assigned for all tools.

**CLOSURE** Allow the user to choose closure time required to close two tools as duration time. The user needs to select two reference tools from tool list illustrated in Figure 8.2.83.



Figure 8.2.83 Closure control stage time

**FULLY MATCH:** This option should be selected when the mesh of two selected tools are not created using physical offset. It is not required for the tools created using physical offset.

**GAP:** Allow the user to specify the distance from complete closure of the two selected tools to stop simulation of current stage. The calculation time is based on the mesh of the two tools, hence blank thickness has to be taken into account. If physical offset algorithm is chosen, the blank thickness is defined as the gap to calculate the closure time.

**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 illustrated in Figure 8.2.84.



Figure 8.2.84 Until home

## OUTPUT D3PLOT FILE CONTROL

This function allows the user to define the number of output files in D3PLOT format. By default, the D3PLOT files are output in equal time interval. In advanced option, the user may enter additional time 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 simulation. The user may change this value in the input data field. 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 illustrated in Figure 8.2.85.

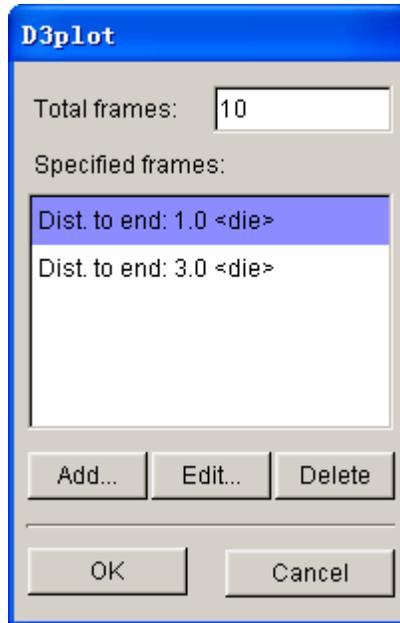


Figure 8.2.85 Add Special Output Frame

**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 specific output frames. The program displays the specified frame dialog box illustrated in Figure 8.2.86.

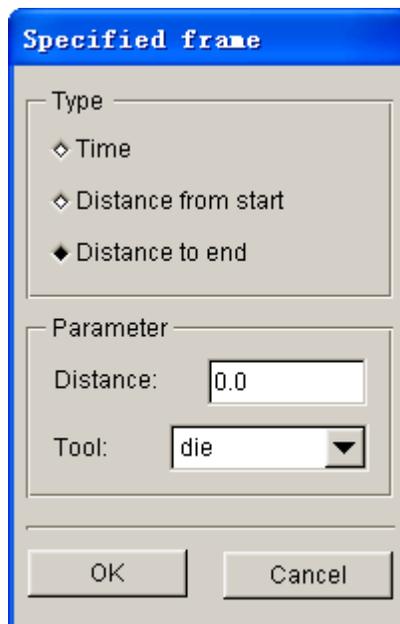


Figure 8.2.86 Specified Frame Define

**TIME:** Add an output frame by specified time. The interface is illustrated in Figure 8.2.87. The user may enter specified time in the TIME input data field 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 specified time. If time is longer than current stage calculation time, the program will ignore the input value. Click

OK button to complete the operation.

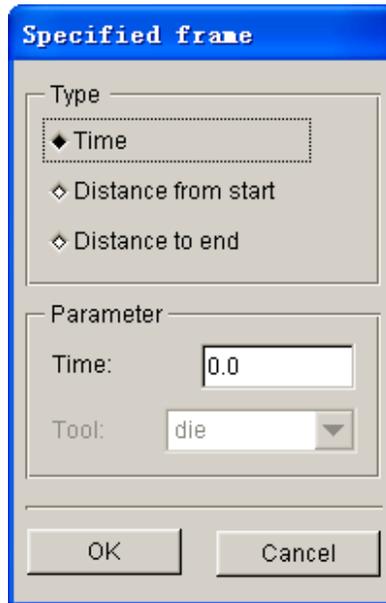


Figure 8.2.87 Input Specified Frame for Specified Time

**DISTANCE FROM START:** Add an output frame by specifying travel distance of the tool. The interface is illustrated in Figure 8.2.88. The user may specify a reference tool in tool pull-down list, following by typing in a value in the DISTANCE input data field. This 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 not valid for the operation.*

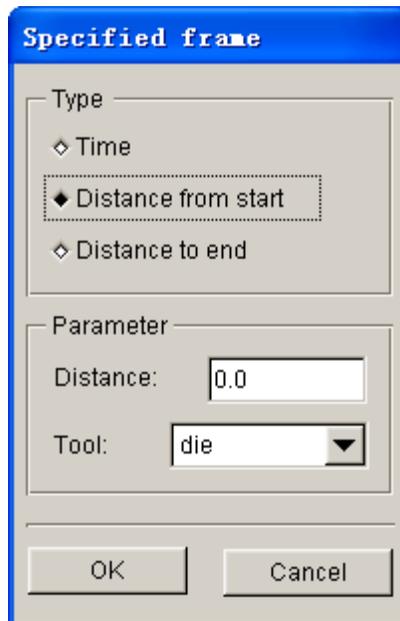


Figure 8.2.88 Define Output Frame by Tool Travel Distance

**DISTANCE TO END:** Add an output frame by defining the distance to the stage termination time. The interface is illustrated in Figure 8.2.89. The user may select a reference tool in TOOL pull-down list, following by typing a value in the DISTANCE input data field. This value is the distance of the tool from termination time.

**NOTE:** *The selected reference tool must be controlled by velocity in the current stage. Otherwise, the specified distance is not valid for the operation.*

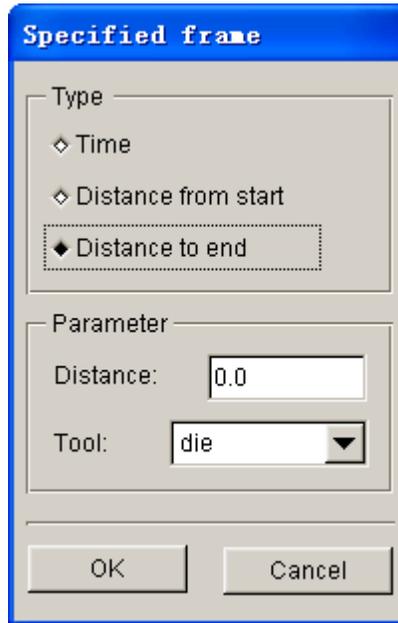


Figure 8.2.89 Define Specified Frame by Distance to Termination

After clicking OK button, the added frame is listed in specified frame list illustrated in Figure 8.2.90. Each item listed in the list displays the control type of frame, tool name and travel distance.

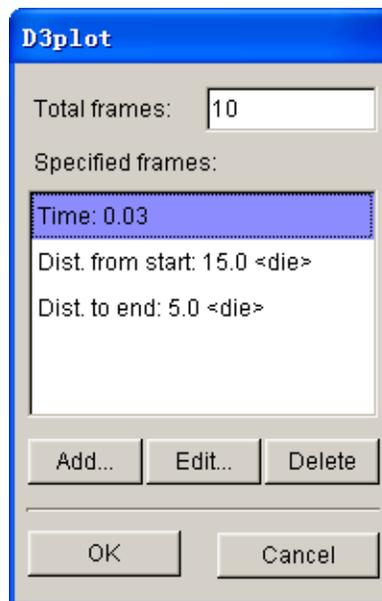


Figure 8.2.90 Specified Frame List

**EDIT...:** The user may modify a frame from the specified frame list.

**DELETE:** The user may remove a frame from the specified frame list.

**OK:** Save current setup 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 8.2.91.

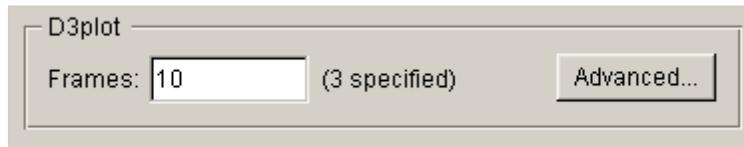


Figure 8.2.91 D3PLOT File of Special Output Time Point

### HYDRO MECHANICAL

This function allows the user to set parameters for hydro mechanical forming. The user may select Hydro Option illustrated in Figure 8.2.92.



Figure 8.2.92 Hydro Forming

### HYDRO MECH. PARAMETERS

In Hydro mech. field illustrated in Figure 8.2.93, the user may specify internal fluid pressure vs. time curve, boundary and direction of applied internal fluid pressure.



Figure 8.2.93 Hydro Mech. parameters setup

**ABOVE:** In press coordinate system, fluid pressure is applied on the top of blank illustrated in Figure 8.2.94.

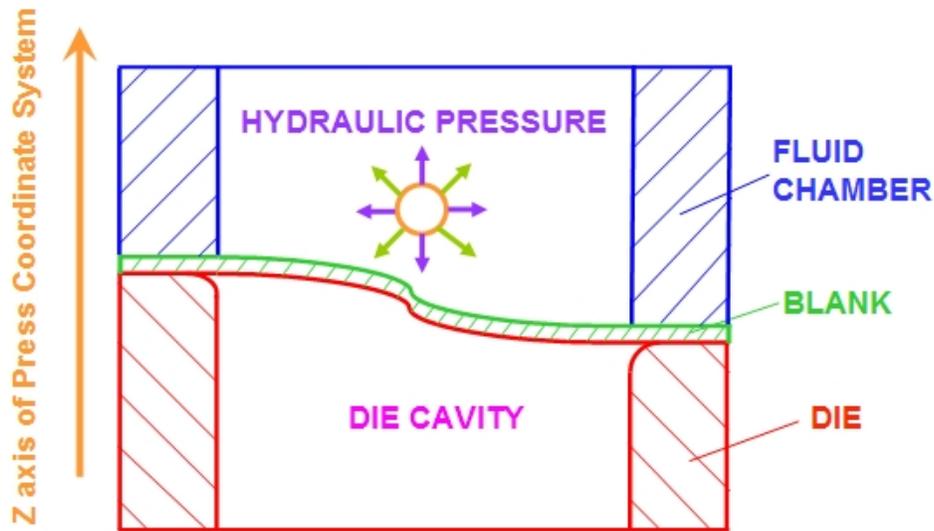


Figure 8.2.94 Fluid pressure is applied on top of blank

**BELOW:** In press coordinate system, fluid pressure is applied on the bottom of blank illustrated in Figure 8.2.95.

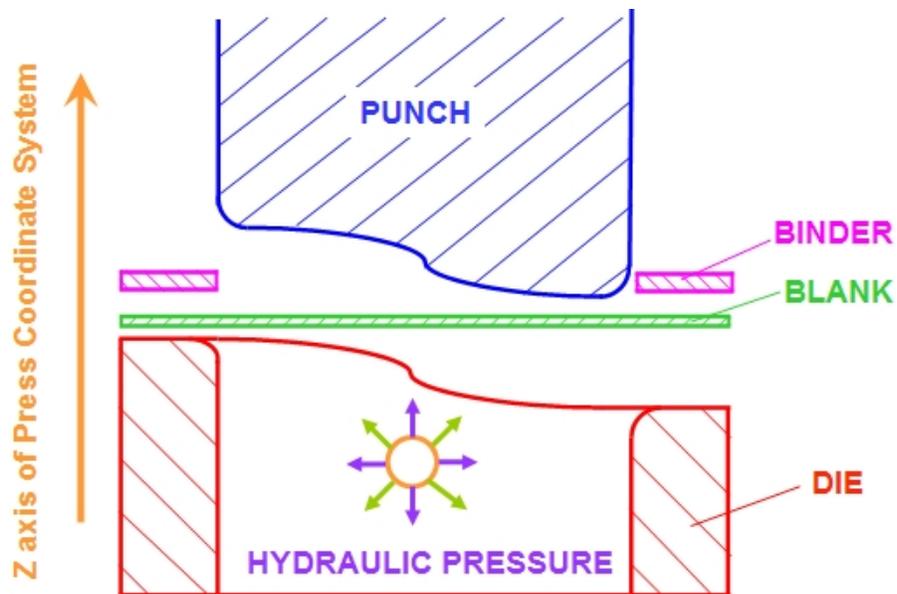


Figure 8.2.95 Fluid pressure is applied on bottom of blank

**BY NORMAL:** Direction of fluid pressure is defined according to normal vector of blank element. The user needs to check and ensure the element normals are consistent. Otherwise, the setup is incorrect.

**PRESSURE:** Set up internal fluid pressure vs. time curve. This parameter is essential in sheet hydro forming. The user may click  button to key in values of internal fluid pressure vs. time curve in the define dialog box illustrated in Figure 8.2.96.

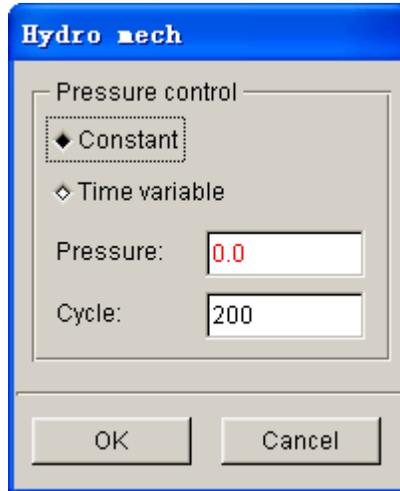


Figure 8.2.96 Hydro Pressure-Time Curve Defines

**CONSTANT:** Define fixed internal fluid pressure. The user may key in a constant fluid pressure in Pressure input data field.

**CYCLE:** Define analysis cycle time of fluid pressure. By default, cycle is set as 200.

**TIME VARIABLE:** Define variable internal fluid pressure. The user may click EDIT button to display curve define interface. For detailed description of curve definition, refer to Chapter 12.

**MASK:** Assign boundary of applied fluid pressure using closed loop line. Applied fluid pressure can be either inside or outside of the boundary. By default, fluid pressure is applied on all blank elements. Click the Entire button to display the MASK dialog box illustrated in Figure 8.2.97.

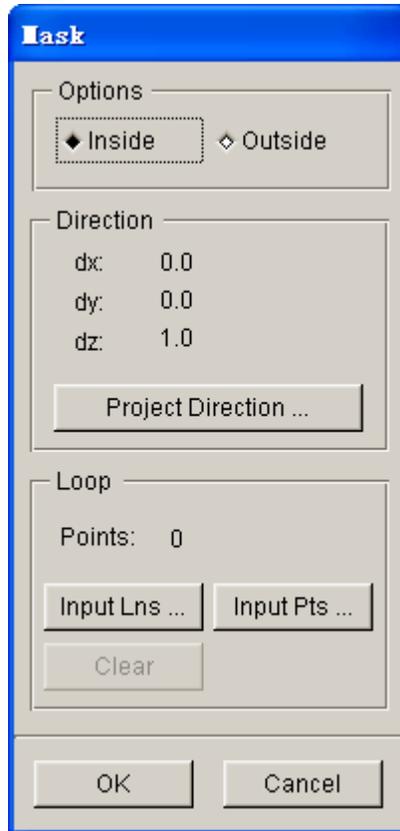


Figure 8.2.97 Define fluid pressure boundary

**INSIDE** Apply fluid pressure inside of the defined boundary line.

**OUTSIDE** Apply fluid pressure outside of the defined boundary line.

**PROJECT DIRECTION...:** Allow the user to select project direction. eta/DYNAFORM adopts one closed loop curve for the definition of fluid pressure, hence it is not necessary to position the curve on top of blank. 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 vector. Default vector starting point is the origin of WCS.

**POINTS:** Display the number of points along curve.

**INPUT LNS...:** The user may define region subjected to applied fluid pressure by defining a closed loop curve. 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 line using points.

**Notice:** When defining boundary of applied fluid pressure, it is recommended to select closed loop curve or a closed end point. If the curve/point is not closed, the program automatically closes curve and point.

**CLEAR:** Remove current defined line. The program displays a confirmation dialog box illustrated in Figure 8.2.98.



Figure 8.2.98 Confirmation dialog box

**OK:** Accept the current setup and exits the dialog box.

**CANCEL:** Abort the current setup and exits the dialog box.

If the region subjected to applied fluid pressure is defined, 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.

### 8.2.4.6 CONTROL

In Control parameter page, the user is allowed to set up basic parameters to control calculation. These parameters include time step size, number of CPUs, adaptive setup, implicit analysis parameters, restart file output control, etc. Control parameter setup interface is illustrated in Figure 8.2.99.

#### GENERAL

**TIME STEP SIZE:** Allow the user to set up time step size. Default time step size is  $-1.2e-6$ .

**ADVANCED....:** Advanced setup for basic parameters. The interface is illustrated in Figure 8.2.99.

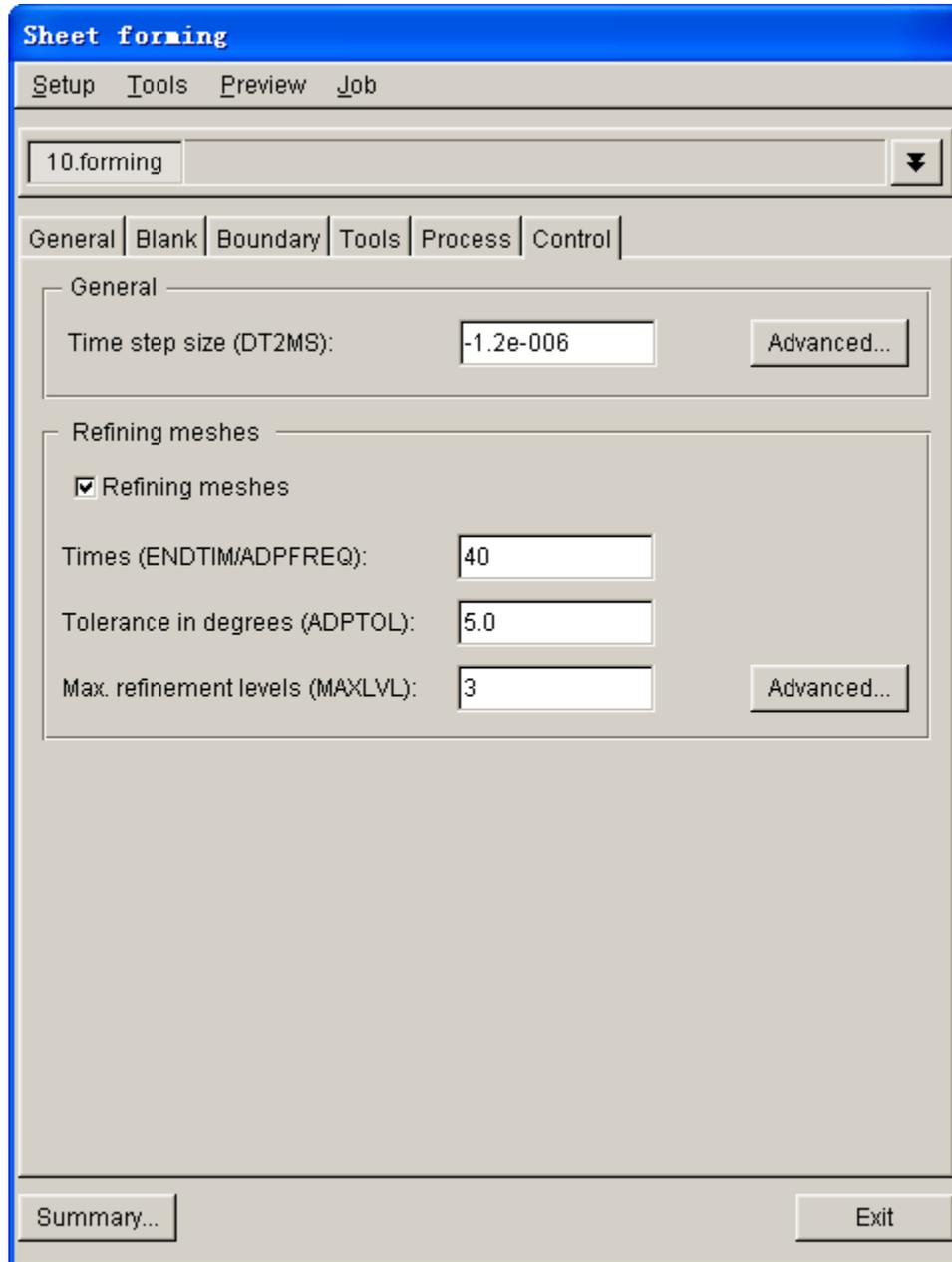


Figure 8.2.99 Basic control parameter setup interface

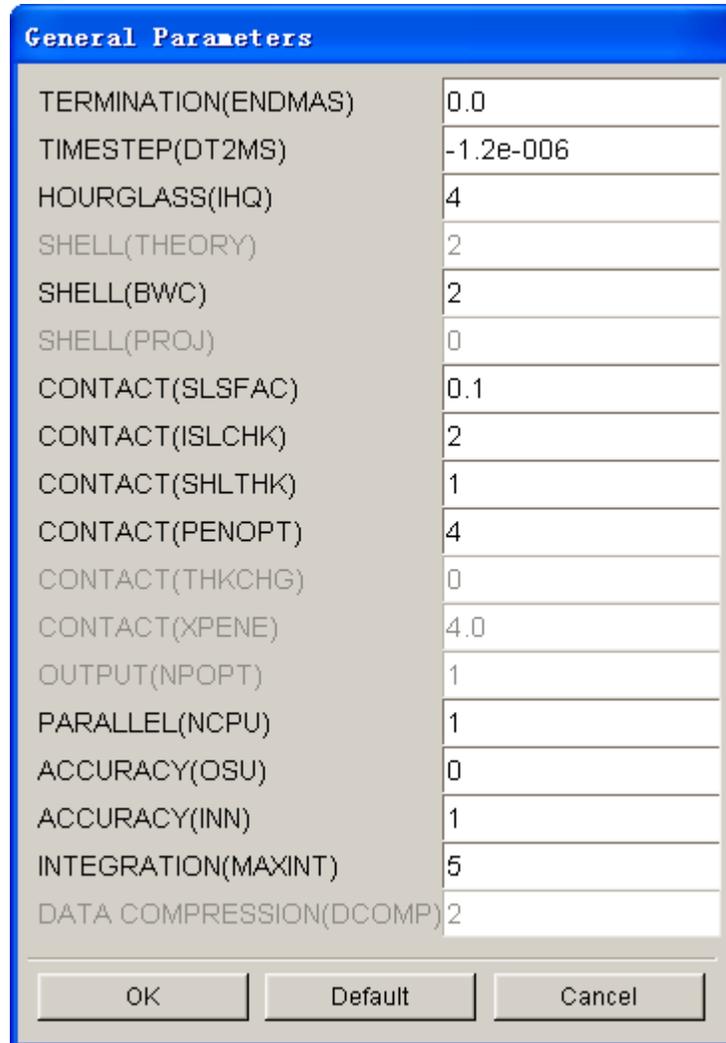


Figure 8.2.100 Basic parameter advanced setup interface

**ADAPTIVE MESHING**

**TIMES (ENDTIME/ADPFREQ):** Allow the user to set the adaptive frequency (ADPFREQ). Default value is 40.

**TOLERANCE IN DEGREE (ADPTOL):** Set angle change between elements during adaptive meshing. If angle change is greater than 5°, the coarse elements are divided into fine elements.

**MAX. REFINEMENT LEVELS (MAXLVL):** Set maximum allowable refinement level. Default level is 3, which mean a square/triangular element is split twice. For example, 1 square element is split into 16 square elements using MAXLVL of 3.

**ADVANCED...:** Advanced setting for adaptive meshing. The interface is illustrated in Figure 8.2.101.

Adaptive Meshing	
TIMES(ENDTIM/ADPFREQ)	40
DEGREES(ADPTOL)	5.0
OPTIONS(ADPOPT)	2
LEVEL(MAXLVL)	3
BIRTH TIME(TBIRTH)	0.0
DEATH TIME(TDEATH)	1e+020
MINIMUM SIZE(ADPSIZE)	1.0
PASS FLAG(ADPASS)	1
UNIFORM LEVEL(IREFLG)	0
ADAPT MESH(ADPENE)	1.0
REMESH THICK(ADPTH)	-0.5
ORIENTATION(ORIENT)	0
ELEMENTS(MAXEL)	0
ELEMENTS(IADPE90)	0
FUSION FREQUENCY(NCFREQ)	2
FISSION LEVEL(IADPCL)	1
FUSION DEGREES(ADPCTL)	0.0
INITIAL FACTOR(FACTIN)	1.0
INCREASE(DFACTR)	0.01

OK      Default      Cancel

Figure 8.2.101 Adaptive advanced parameters

## 8.2.5 GRAVITY

GRAVITY stage is used to simulate the deformation of blank under gravity load. The setup is similar to normal sheet forming process simulation. Refer to Section 8.2.4 Forming for description about basic parameters, blank define, tool define, etc. included in Gravity function.

### 8.2.5.1 BOUNDARY CONDITION

This function includes Constraints and Initial conditions. Refer to Section 8.2.4 Forming for definition of Boundary Condition. The interface of Boundary Condition is illustrated in Figure 8.2.102.

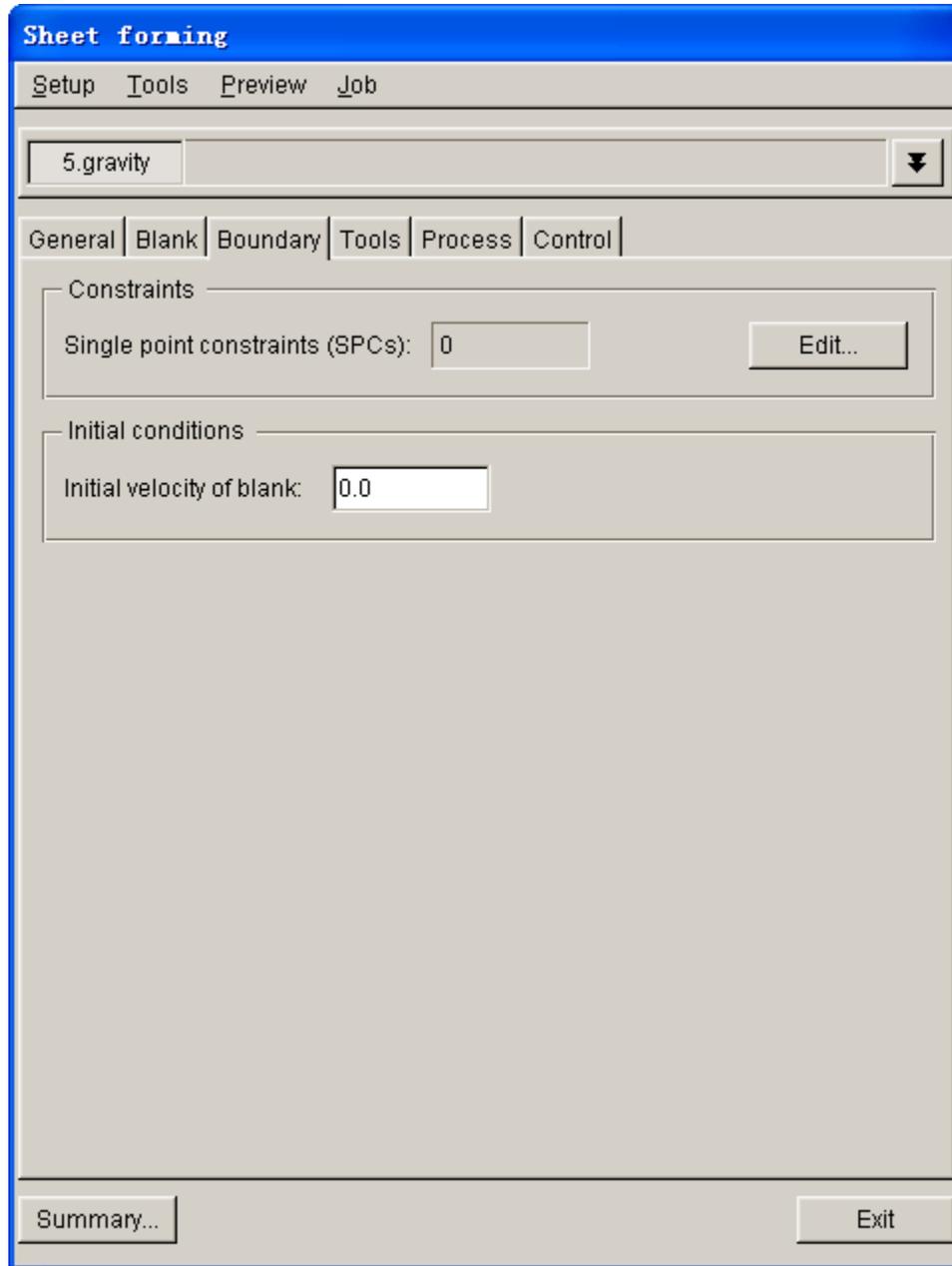


Figure 8.2.102 Boundary condition interface

**INITIAL VELOCITY OF BLANK:** Define the initial velocity of blank.

### 8.2.5.2 PROCESS

The interface of Process definition page is illustrated in Figure 8.2.103. In Process definition page, at least one tool is set as STATIONARY.

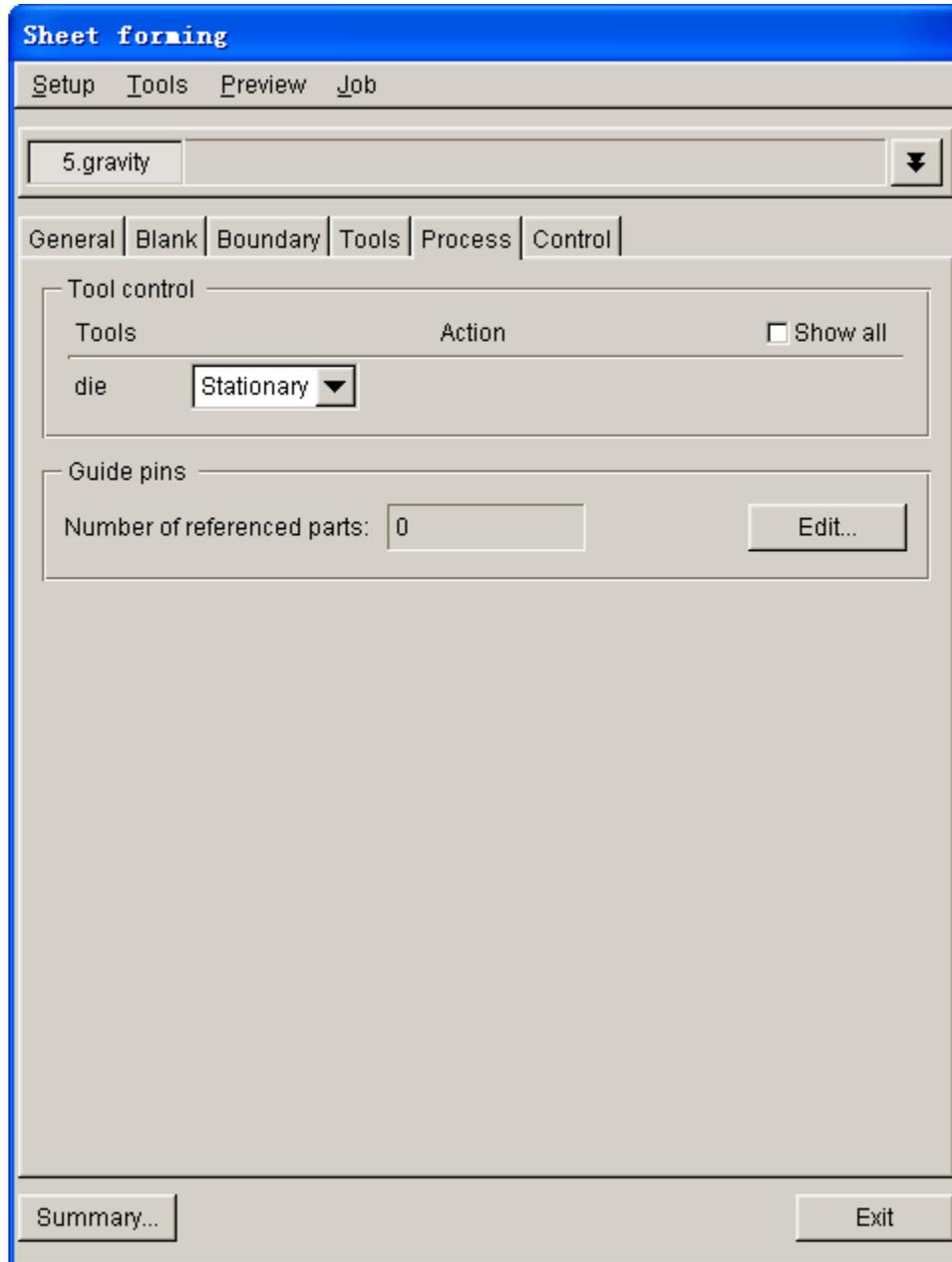


Figure 8.2.103 Process definition page

**TOOL CONTROL:** Set tool to be Stationary or Non-active.

**SHOW ALL:** Show all the defined tools.

**GUIDE PINS:** Add or create GUIDE and/or PIN, and create a new part for the GUIDE as current tool. Default name of new GUIDE is ACC0000, ACC0001, etc.. For detailed description of GUIDE and PIN, refer to Section 10.6.

### 8.2.5.3 CONTROL

In Control definition page, the user is allowed to set up basic parameters to control gravity simulation such as General and Explicit/Implicit analysis parameters. Control definition page is illustrated in Figure 8.2.104.

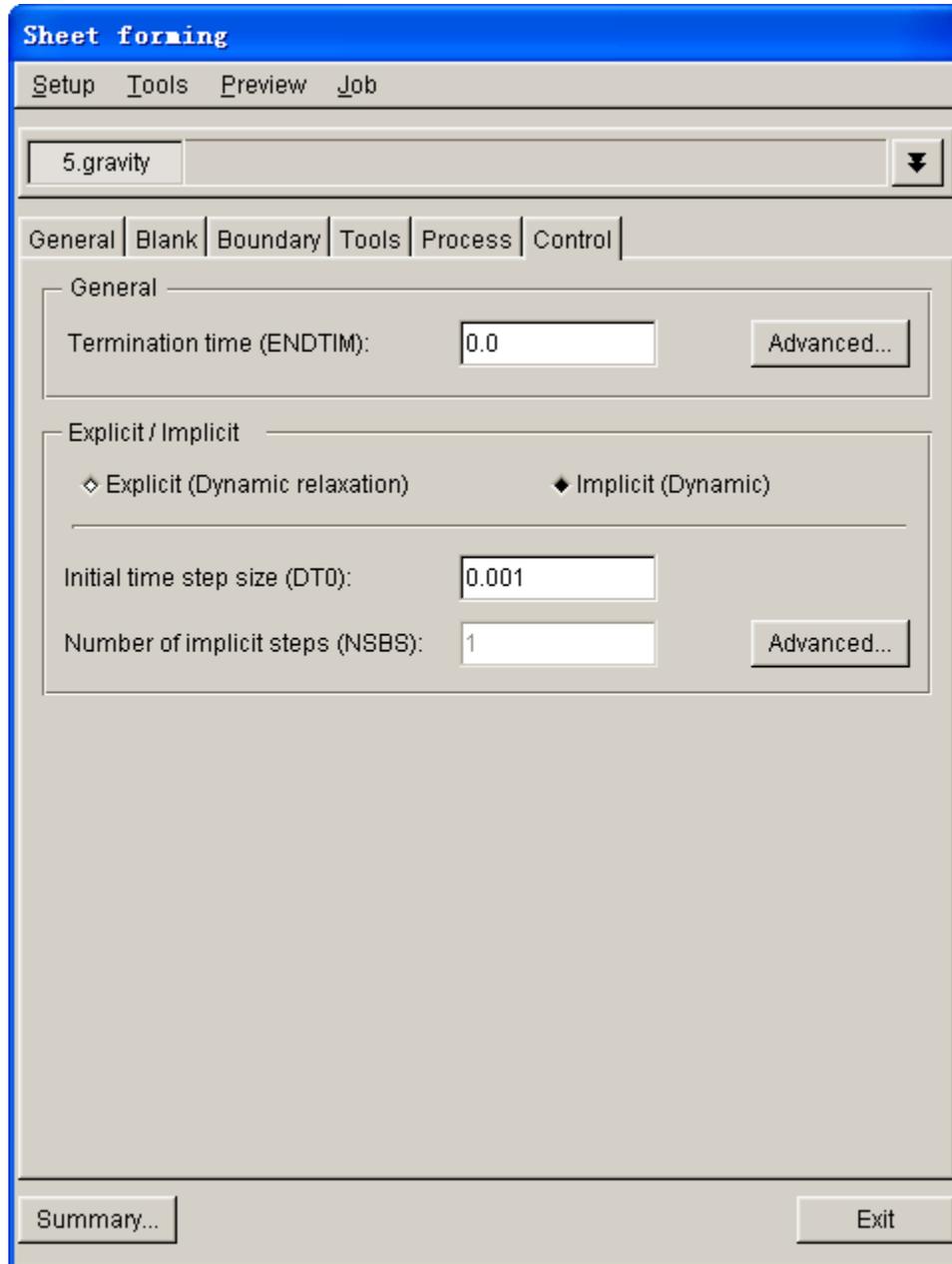


Figure 8.2.104 Control definition page

**TERMINATION TIME (ENDTIM):** Termination time.

**ADVANCED...:** Advanced setup of basic parameters. The dialog box is illustrated in Figure 8.2.105.

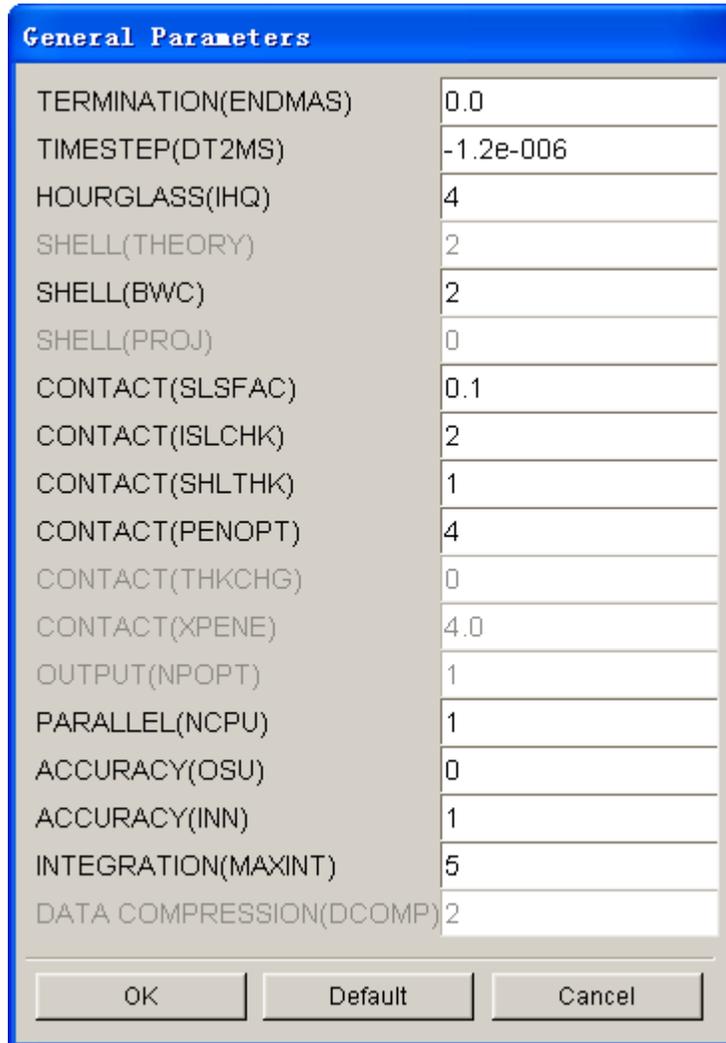
**EXPLICIT/ (DYNAMIC RELAXATION):** Set parameters used for explicit approach.

**ITERATIONS NUMBER (NRCYCK)**

**CONVERGENCE TOLERANCE (DRTOL):** Set convergence tolerance during dynamic relaxation calculation. Default convergence tolerance is 0.01.

**ADVANCED:** Advanced parameter of dynamic relaxation. The interface is illustrated in Figure

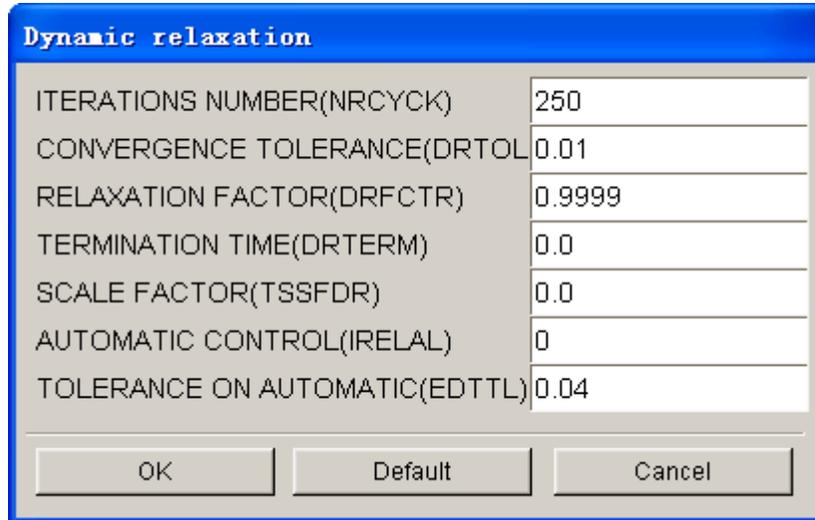
8.2.106.



General Parameters	
TERMINATION(ENDMAS)	0.0
TIMESTEP(DT2MS)	-1.2e-006
HOURGLASS(IHQ)	4
SHELL(THEORY)	2
SHELL(BWC)	2
SHELL(PROJ)	0
CONTACT(SLSFAC)	0.1
CONTACT(ISLCHK)	2
CONTACT(SHLTHK)	1
CONTACT(PENOPT)	4
CONTACT(THKCHG)	0
CONTACT(XPENE)	4.0
OUTPUT(NPOPT)	1
PARALLEL(NCPU)	1
ACCURACY(OSU)	0
ACCURACY(INN)	1
INTEGRATION(MAXINT)	5
DATA COMPRESSION(DCOMP)	2

OK Default Cancel

Figure 8.2.105 Basic parameter advanced setup interface



The image shows a dialog box titled "Dynamic relaxation" with a blue header. It contains a table of parameters and their values, and three buttons at the bottom: "OK", "Default", and "Cancel".

Parameter	Value
ITERATIONS NUMBER(NRCYCK)	250
CONVERGENCE TOLERANCE(DRTOL)	0.01
RELAXATION FACTOR(DRFCTR)	0.9999
TERMINATION TIME(DRTERM)	0.0
SCALE FACTOR(TSSFDR)	0.0
AUTOMATIC CONTROL(IRELAL)	0
TOLERANCE ON AUTOMATIC(EDTTL)	0.04

Figure 8.2.106 Dynamic relaxation parameter

**IMPLICIT (DYNAMIC):** Set parameters used for implicit approach.

**INITIAL TIME STEP SIZE (DT0):** Initial time step size. Default value is 0.001.

**NUMBER OF IMPLICIT STEP (NSBS):** Number of implicit step. Default value is 1.

**ADVANCED...:** Set advanced parameter of implicit analysis. The list of parameters is illustrated in Figure 8.2.107.

Implicit analysis	
GENERAL / INITIAL_TIMESTEP(DT0)	0.001
SOLUTION / METHOD(NSOLVR)	2
SOLUTION / ITERATION_LIMIT(ILIMIT)	1
SOLUTION / STIFFNESS_LIMIT(MAXREF)	1000
SOLUTION / DISPLACEMENT(DCTOL)	0.001
SOLUTION / ENERGY(ECTOL)	1.0
SOLUTION / RESIDUAL_CONVERGENCE(RCTOL)	0.0
SOLUTION / LINE_SEARCH(LSTOL)	0.0
SOLUTION / DISP_NORM(DNORM)	0
SOLUTION / DIVERGENCE(DIVERG)	2
SOLUTION / INITIAL_STIFFNESS(ISTIF)	0
SOLUTION / NONLEAR_PRINT(NLPRINT)	1
SOLVER / METHOD(LSOLVR)	5
SOLVER / PRINT(LPRINT)	0
SOLVER / EIGENVALUE(NEGEV)	0

Buttons: OK, Default, Cancel

Figure 8.2.107 Implicit parameters

## 8.2.6 TRIMMING

Trimming is the subsequent process carried out after successful drawing process and prior to performing springback analysis. It allows the user to perform hole piercing and/or trimming operation on the drawn parts. Trimming process consists of four basic pages such as General and Blank pages described in Section 8.2.4, Forming. In Multi Stage setup, the user can also set boundary condition for trimming.

### 8.2.6.1 PROCESS

This function allows the user to define single or multiple groups of lines. These lines are chosen to perform piercing and/or trimming operation illustrated in Figure 8.2.108.

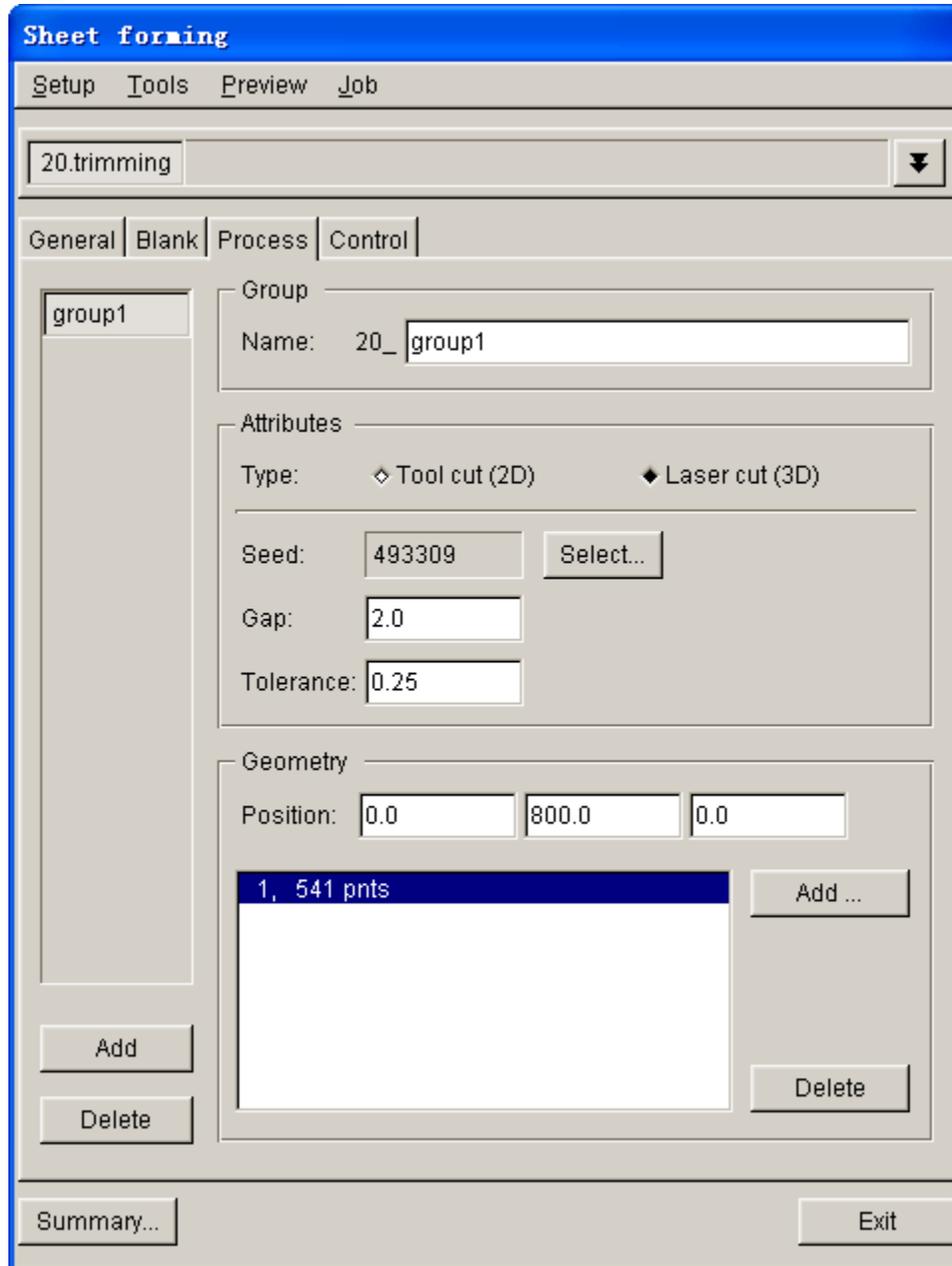


Figure 8.2.108 Trimming process

**GROUP LIST**

Group List lists all defined group names of the current line. The user classifies these lines for ease of line management. If multiple line groups are defined, the latest line group is always highlighted in the interface as well as on the display screen.

**ADD**

Create new line group.

**DELETE**

Remove selected or current line group. The dialog box illustrated in Figure 8.2.109 is displayed prompting the user to confirm Delete operation.



Figure 8.2.109 Delete question dialog box

### GROUP NAME

This function allows the user to modify the name of the current group. When creating groups, the user can specify a unique name for the new group. The user may click on Group button to modify the group name by typing the new name in the input data field of NAME, following by press ENTER key to complete the operation.

### ATTRIBUTES

This function allows the user to define line type, trimming mode and direction, seed, gap, tolerance, etc. The user can select different line type for definition of different attributes.

#### TYPE

There are two types of line: Tool cut (2D) and Laser cut (3D), which correspond to 2D and 3D trimming respectively.

*Note: The trimming direction of 2D line is determined according to the defined vector direction; and the trimming direction of 3D line is determined according to the normal direction of element.*

#### MODE

There are two modes of 2D trimming: TRIMMING and HOLE. The user can classify these trim curves.

#### DIRECTION

Define trimming direction. Refer to the previous sections for description about defining direction.

#### TOLERANCE

Trimming tolerance is used to control the size of elements generated after trimming operation. If the value is zero, element size is not restricted; if the value is 0.5, the size of the newly generated element should at least be half of its parent element; if the value is 1, it is allowed to move the original node to the trim curve without generating new element after trimming. If the tolerance value is negative, the user only perform simple trimming, i.e. deleting the entire element while keeping the serrated boundary. The default tolerance is 0.25.

#### SEED

Any node on trimmed part after trimming operation can be defined as seed node.

#### GAP

Define the maximum gap between trim curve and mesh. If the value is greater than a specified gap, the corresponding segment of trim curve is disabled.

**GEOMETRY**

This function allows the user to translate, add and delete lines.

**POSITION**

Translate the selected trim curve. The user can input the translation value and click on the distance to be translated to complete the line translation. There are three input data fields corresponding to dx, dy and dz values under the current CS.

**ADD**

Allow the user to add one or more trim curves. Click ADD button to display the SELECT LINE dialog box. Refer to the previous sections for information on methods of selecting lines.

**DELETE**

Remove the selected lines. Select the lines and click DELETE button to display a dialog box illustrated in Figure 8.2.110.

- OK:** Confirm the selected line.  
**CANCEL:** Abort the Delete operation.

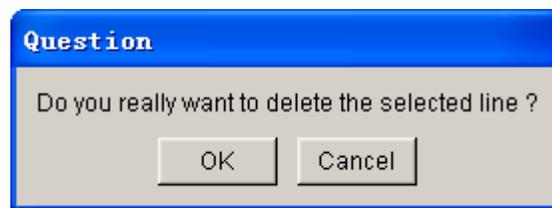


Figure 8.2.110 Delete line question dialog box

**8.2.6.2 CONTROL**

This function allows the user to set parameters of refining mesh along trimming curves. The interface is illustrated in Figure 8.2.111.

**REFINING MESHES****REFINING MESHES ALONG TRIMMING CURVES**

Once this option is selected, the user is allowed to set parameters of refining mesh along trimming curves. If this option is not selected, the trimming operation does not consider such operation.

**MAX. REFINEMENT LEVELS (IFADA):** The default level is 3, i.e. an element can be refined into 16 elements.

**MIN. REFINEMENT SIZE (SMIN):** The allowable minimum refinement size. The default is 0.

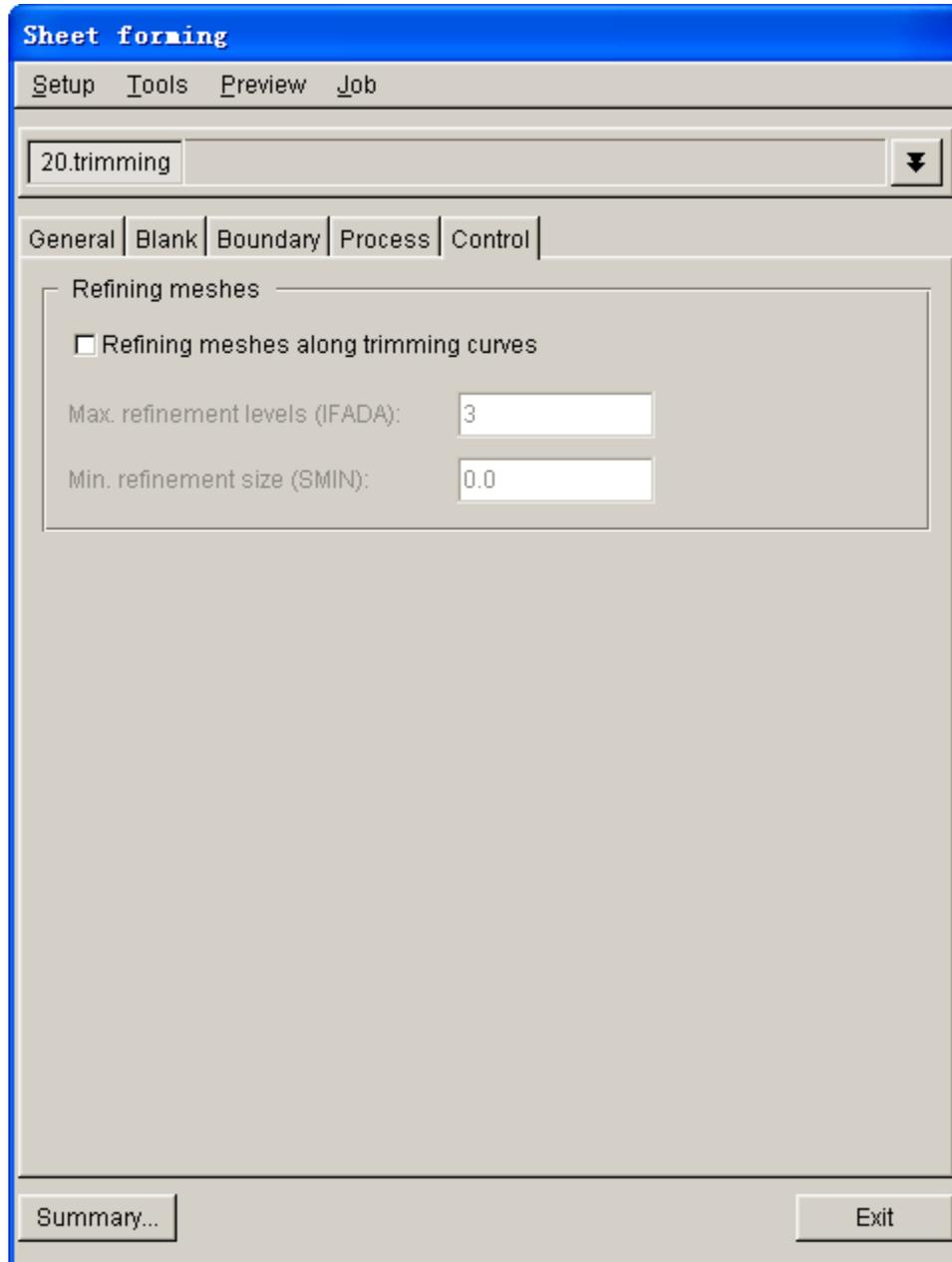


Figure 8.2.111 Trimming control parameters interface

## 8.2.7 SPRINGBACK

The SPRINGBACK under Process template is used to simulate the springback of the blank after drawing simulation, as shown in Figure 8.2.112. The user can also add springback stage in multi stage simulation, as illustrated in Figure 8.2.113. Springback process consists of four pages. For information about General and Blank, refer to Section 8.2.4 Forming.

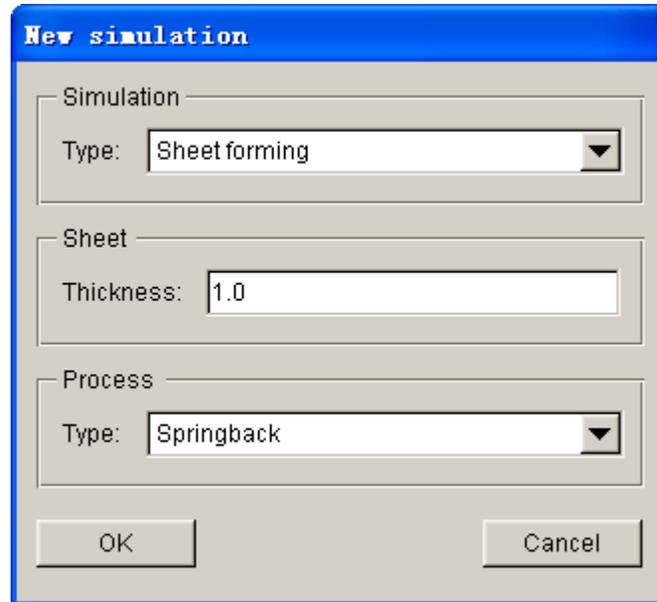


Figure 8.2.112 Springback analysis

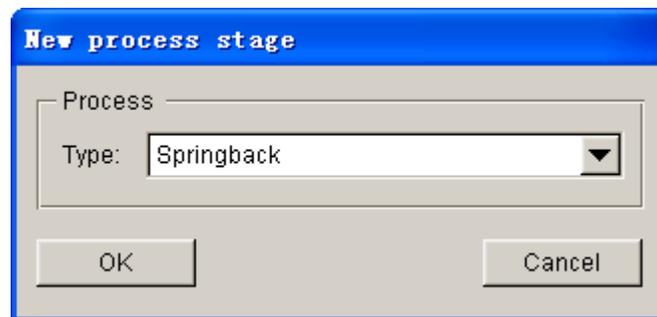


Figure 8.2.113 New springback stage

### 8.2.7.1 BOUNDARY CONDITION

The Boundary Condition page is illustrated in Figure 8.2.114. This page enables the user to define springback constraints. Refer to Section 8.2.4.3 for detailed description on definition of springback constraints.

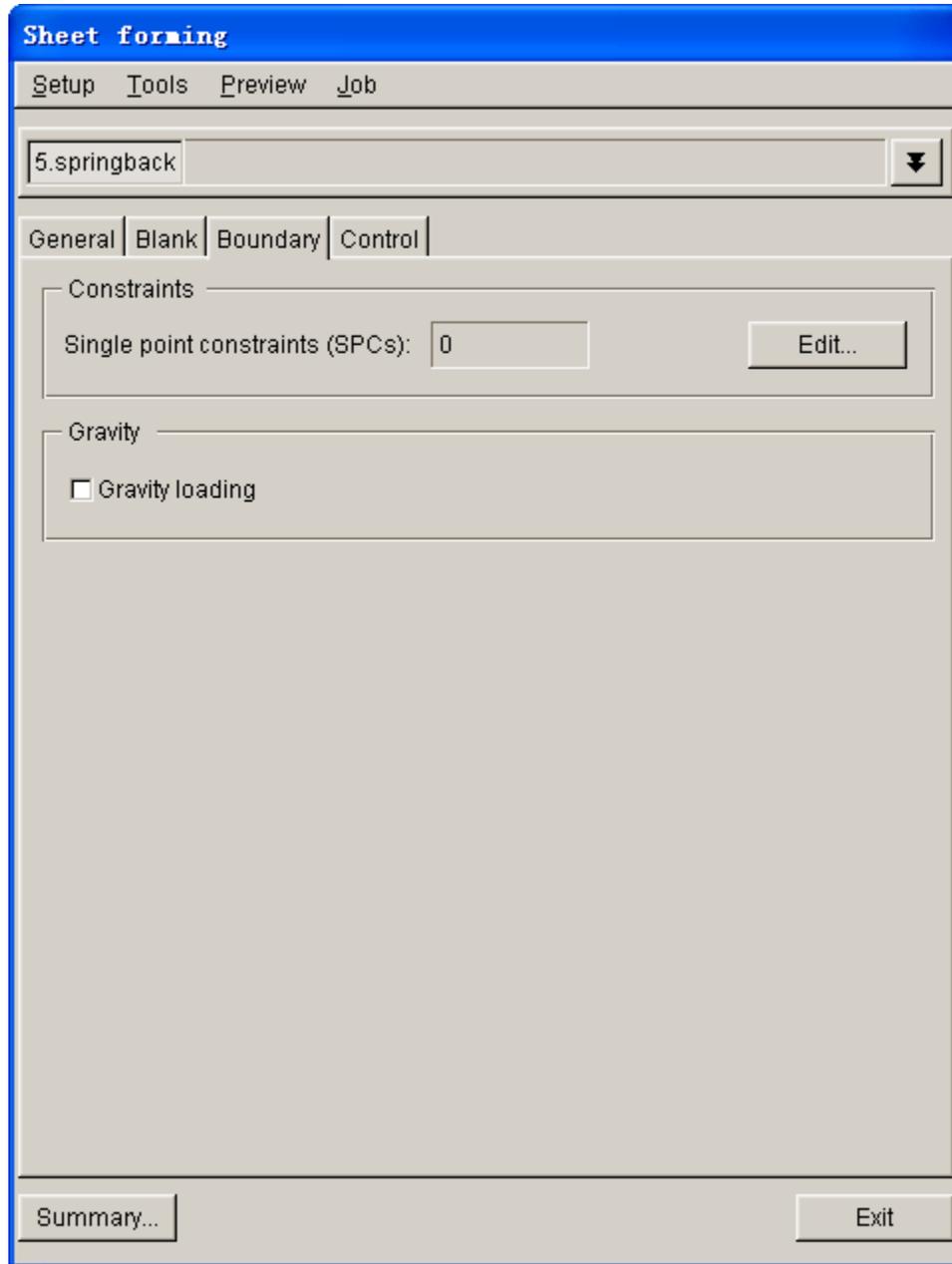


Figure 8.2.114 Springback boundary condition

## CONSTRAINTS

Allow the user to select three nodes to constrain rigid displacement of blank during springback analysis. 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. The relationship of location for these three constraints is illustrated in Figure 8.2.115.

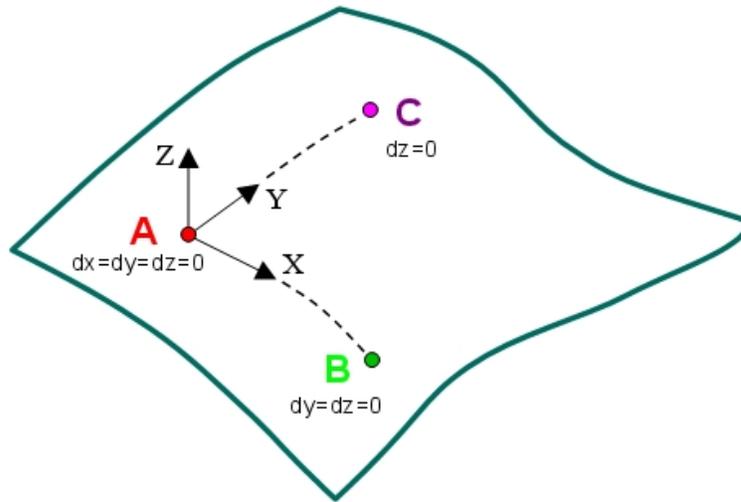


Figure 8.2.115 Springback constraints definition

### 8.2.7.2 CONTROL

Set control parameters for springback analysis, as illustrated in Figure 8.2.116.

#### GENERAL

**SHELL WARPING STIFFNESS (BWC):** Default is set as 2, i.e. Belytschko-Tsay

**ADVANCED...:** Advanced parameter setting.

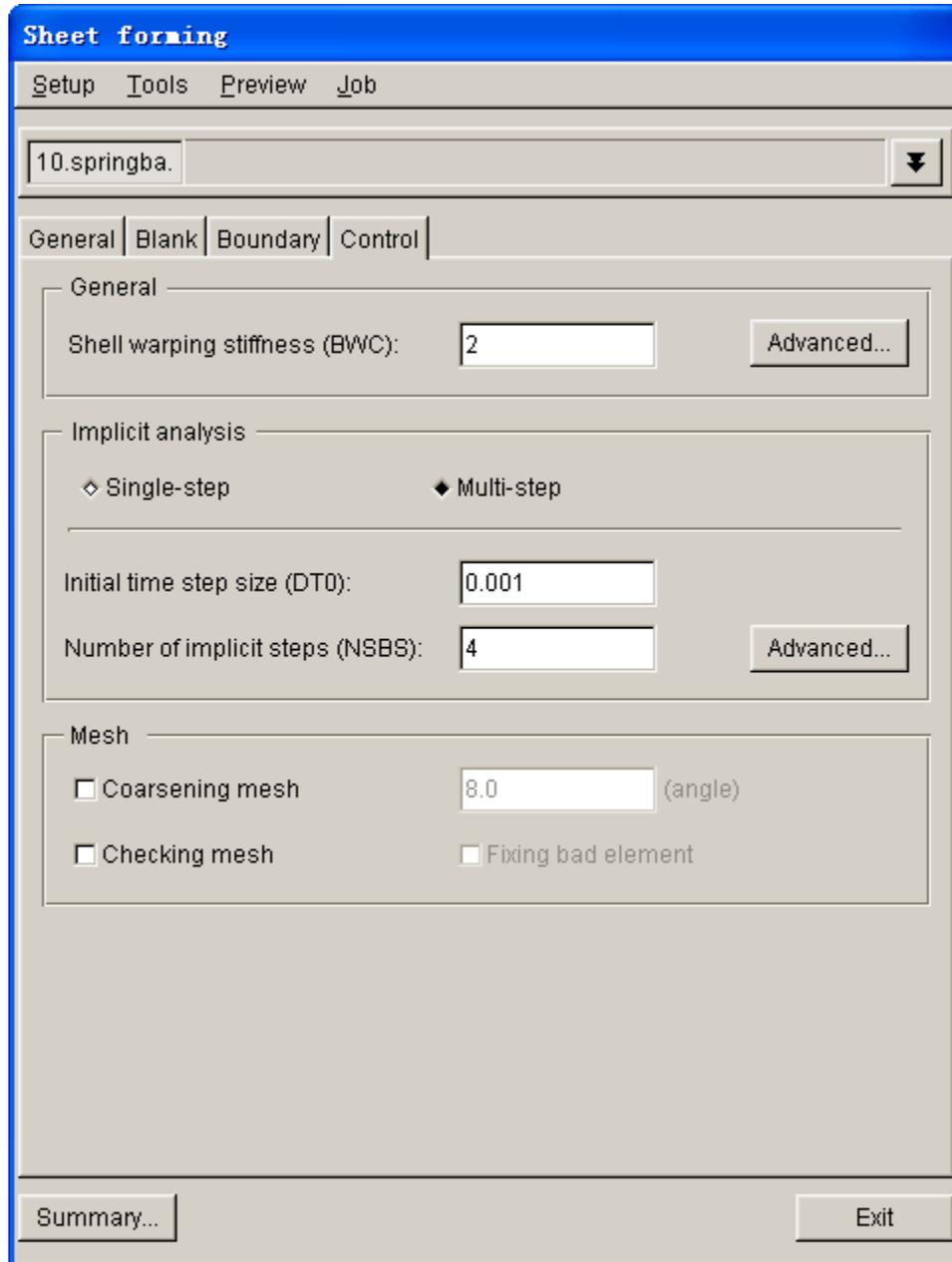


Figure 8.2.116 Springback control parameters interface

## IMPLICIT ANALYSIS

Allow to user to select approach of springback analysis: single step springback or multiple step springback analysis. Multiple step springback is set as default option. Single step springback analysis usually leads to good result. However, converged result may not be obtained using single step springback analysis for relatively flexible part. The user is recommended to conduct springback analysis using multiple step approach. Detailed descriptions for setting up single or multiple step springback analysis is described in LS-DYNA User's Manual.

**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. Set to 1 if single step implicit analysis is performed. If the value is greater than 1, multiple implicit analysis is used.

**ADVANCED...:** Set advanced parameter of implicit analysis. See Figure 8.2.117 for the list of parameters.

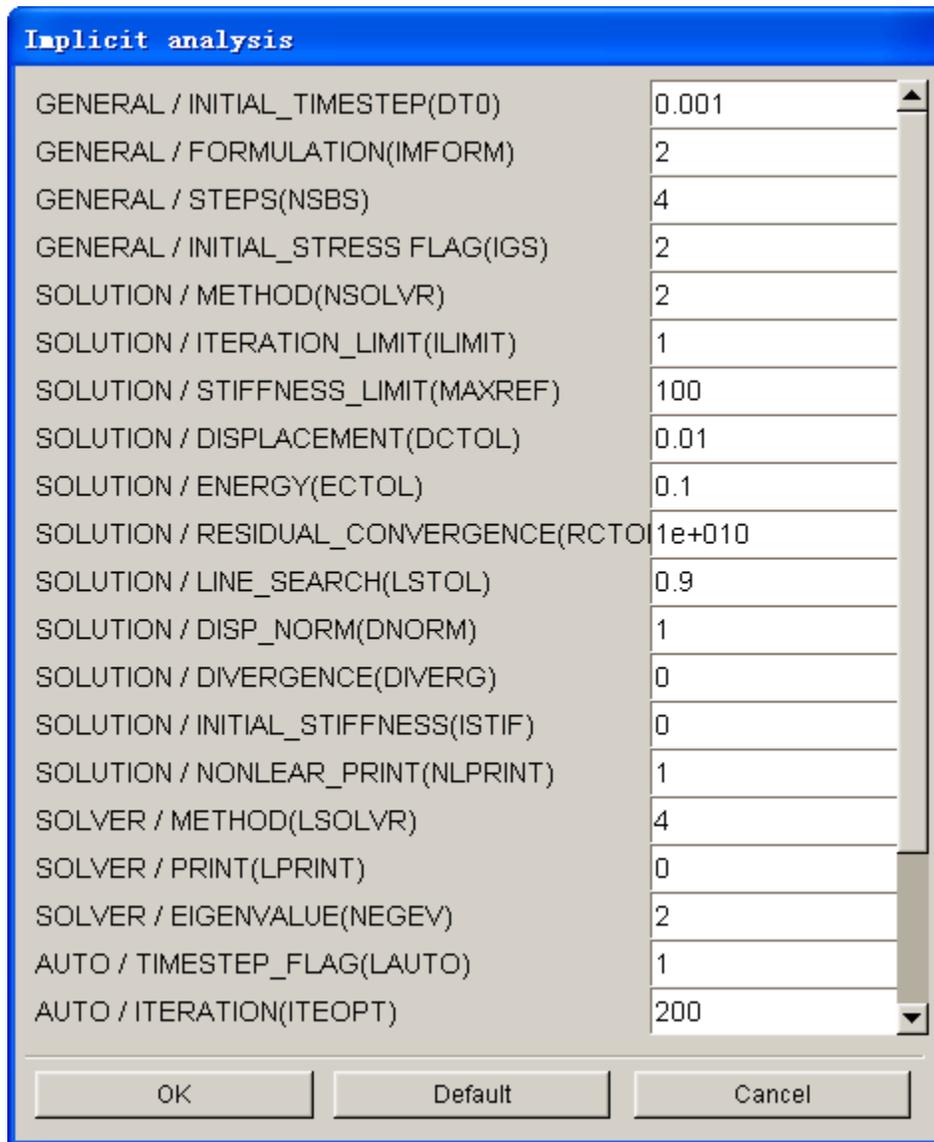


Figure 8.2.117 Advanced parameter for Implicit Analysis

### COARSENING MESH

Allow the user to use coarsening mesh technique during springback analysis. During drawing simulation, adaptive mesh is adopted to refine blank mesh to capture details on tooling. Adaptive meshing increases model size and poses challenge to the convergence in springback analysis. This option merges the refine

mesh on the flat areas to reduce model size. It reduces convergence problem and shorten the computing time.

**ANGLE:** Set the angle criteria for merge neighboring elements. If difference of element normal angle between adjacent elements is less than the given criteria, these elements are merged. Default angle is 8°.

### CHECKING MESH

This function allows the user to check mesh quality during Springback analysis. The user can check adaptive constraints, minimum aspect ratio, minimum and maximum allowable interior angle and minimum element size.

### FIXING BAD ELEMENT

This function enables solver to automatically repair distorted elements and output the relevant information into text file. It helps to increase precision of springback analysis, as well as improving convergence.

## 8.2.8 SUPERPLASTIC FORMING

Superplastic forming module is used to analyze the formability of sheet metal under superplastic temperature and subjected to applied pressure. The setup of forming process is similar to the forming process using common sheet metal, with an exception for the Process setup described in the following section.

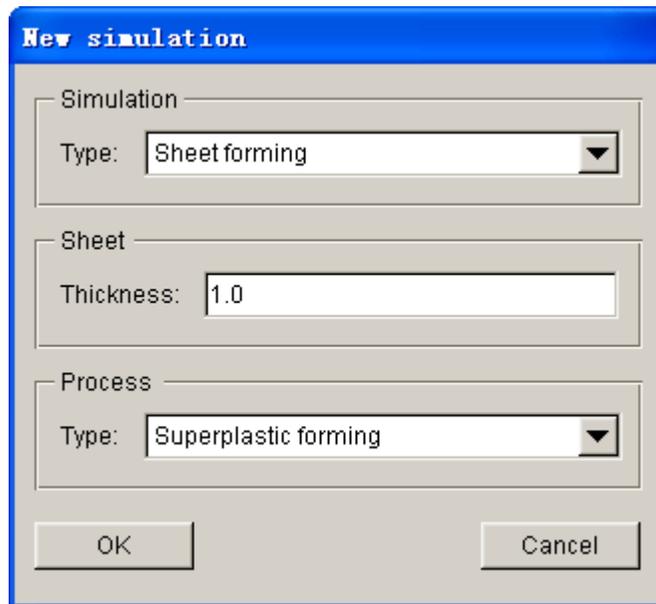


Figure 8.2.118 Superplastic forming

### 8.2.8.1 PROCESS

Process page enables the setup of basic process parameter for superplastic forming such as pressure curve, percent to terminate, tool contact, desired strain rate, termination time, etc. The user can modify process parameters according to practical situation. Process page for superplastic forming is illustrated in Figure 8.2.119.

**PHASE**

The user can select one or two phase analysis. If one phase analysis is selected, the sheet is directly formed into the final part shape by applied pressure. If two phase analysis are selected, preforming is performed in the first phase to partially or fully make the sheet to confront to preformed die. In second phase, reverse pressure is applied to confront the sheet to die geometry to obtain the final part shape. Pressure and reverse pressure are usually used for superplastic forming process to obtain even part thickness.

**PHASE I**

Define parameters such as pressure curve, percent to terminate and contact tool.

**PRESSURE CURVE**

Click on <none> button to define pressure curve using EDIT LOAD CURVE dialog box described in previous section.

**CONTACT TOOL**

Select tools contacting with sheet.

**PERCENT TO TERMINATE**

Enter the contact node percent between sheet and selected contact tool. The user can use contact node percent to control the termination time of this phase. If the sheet contacts with 80% nodes of the contact tool, this forming phase is terminated.

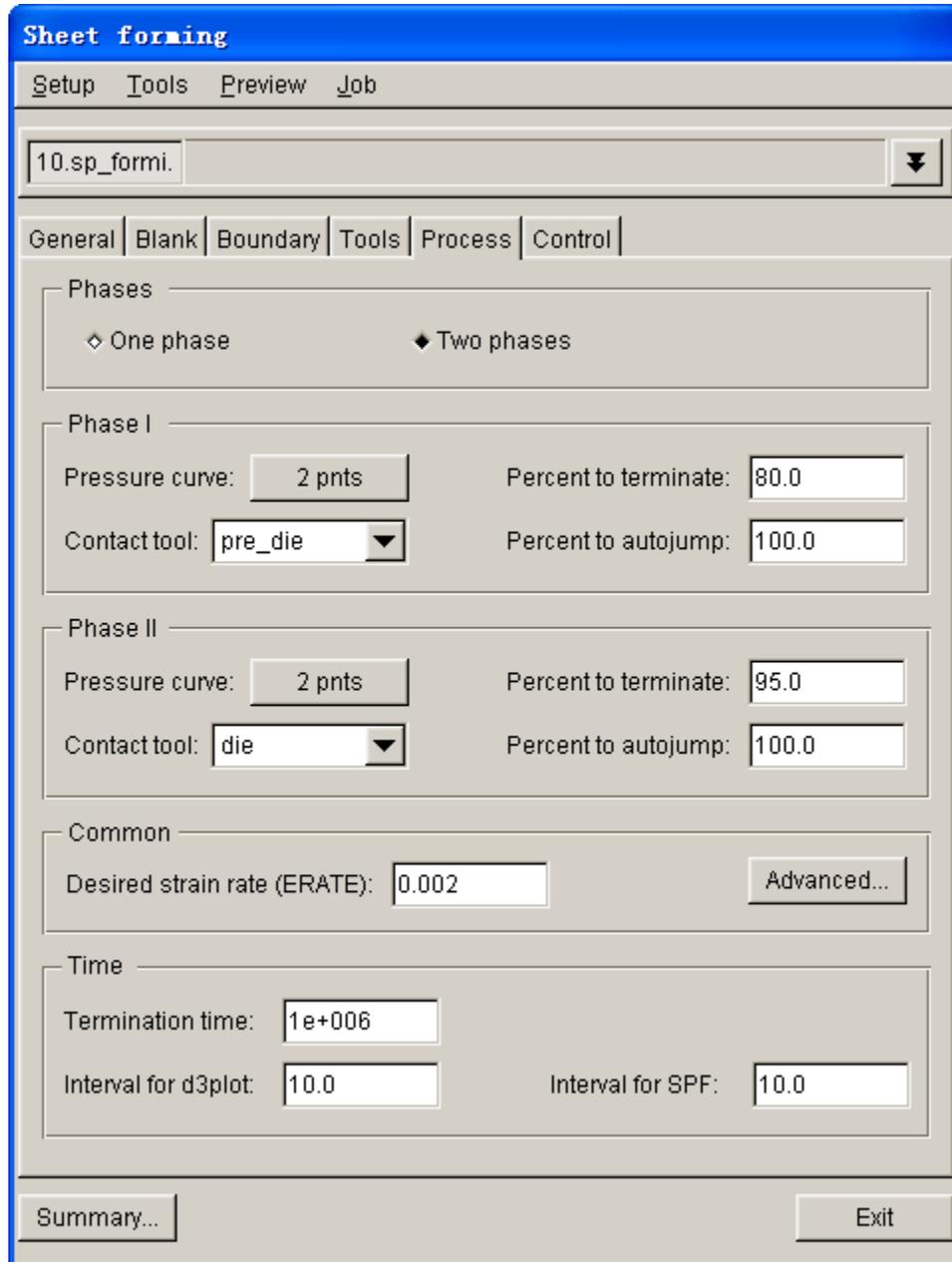


Figure 8.2.119 Process page

**PERCENT TO AUTOJUMP**

Key in the contact node percent in this field if autojump option is selected in the Advanced dialog box. The current phase is automatically terminated if the contact nodes reach the input percent.

**PHASE II**

If part is formed in two phases, the definition of parameters for Phase II is added to the interface. The definition of parameter for Phase II is similar to Phase I. Refer to the above description.

**COMMON**

**DESIRED STRAIN RATE (ERATE)**

Set desired strain rate. Strain rate is the derivative of equivalent plastic strain with respect to time.

**ADVANCED...**

Set advanced forming parameters. Refer to LS-DYNA User's Manual for description about advanced setting.

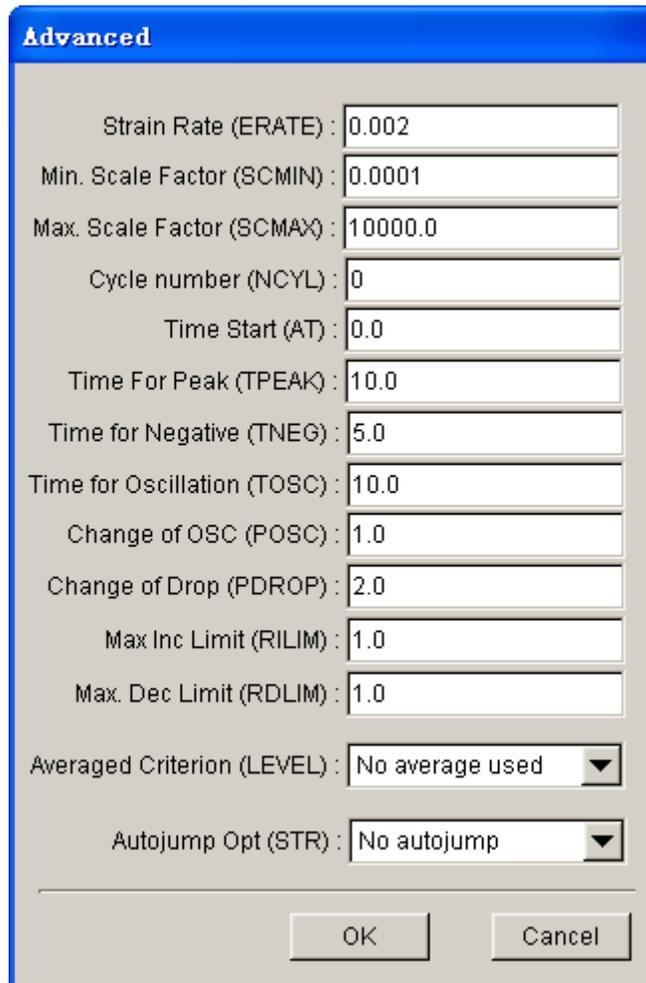


Figure 8.2.120 Advance setting for superplastic forming

**8.2.9 TUBE FORMING**

The setup process of tubular hydroforming is basically similar to setup of normal sheet forming. The user may refer to the setting of sheet forming described in previous sections. Specialized setup options related to tubular hydroforming are provided in this section.

**8.2.9.1 NEW SIMULATION**

Create a new simulation. Select TUBE FORMING as simulation type from the drop-down list illustrated in Figure 8.2.121. Tube forming consists of four pages: FORMING, SPRINGBACK, CUTTING and

SUPERPLASTIC FORMING, as shown in Figure 8.2.122.

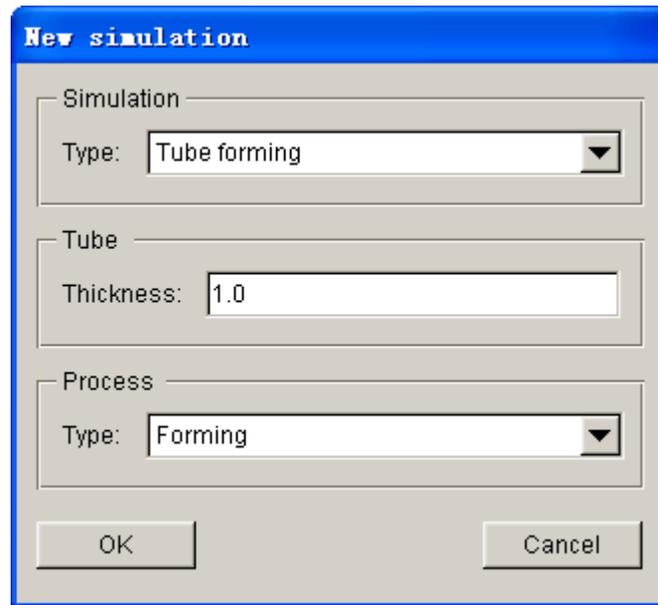


Figure 8.2.121 Tube forming

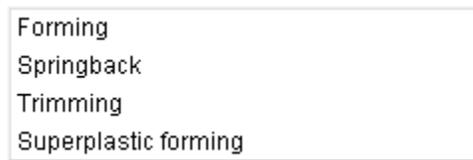


Figure 8.2.122 Tube forming page

The user keys in initial blank thickness, following by clicking OK button to display the setup interface of tube forming.

### 8.2.9.2 GENERAL

Similar to those described in setup of sheet forming. The user may specify title and working coordination system. The interface is illustrated in Figure 8.2.123. Detailed description of the GENERAL page is provided in Section 8.2.3.

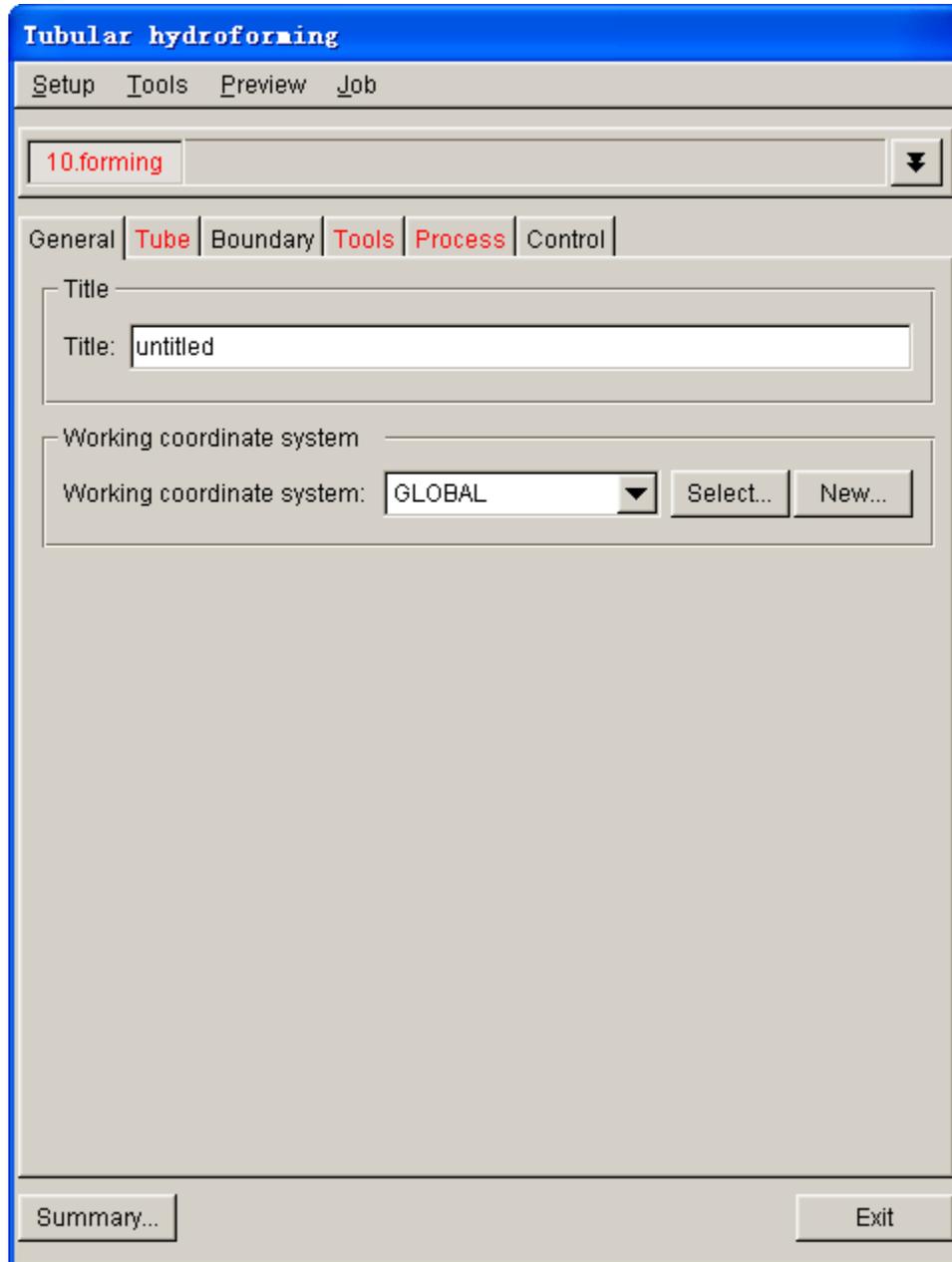


Figure 8.2.123 Interface of tubular hydroforming

### 8.2.9.3 TUBE

This function allows the user to assign as part as tube. The user may click PART button to select a part from the Part List as tube. The interface is illustrated in Figure 8.2.124. The process of defining tube is similar to blank definition process described in previous section.

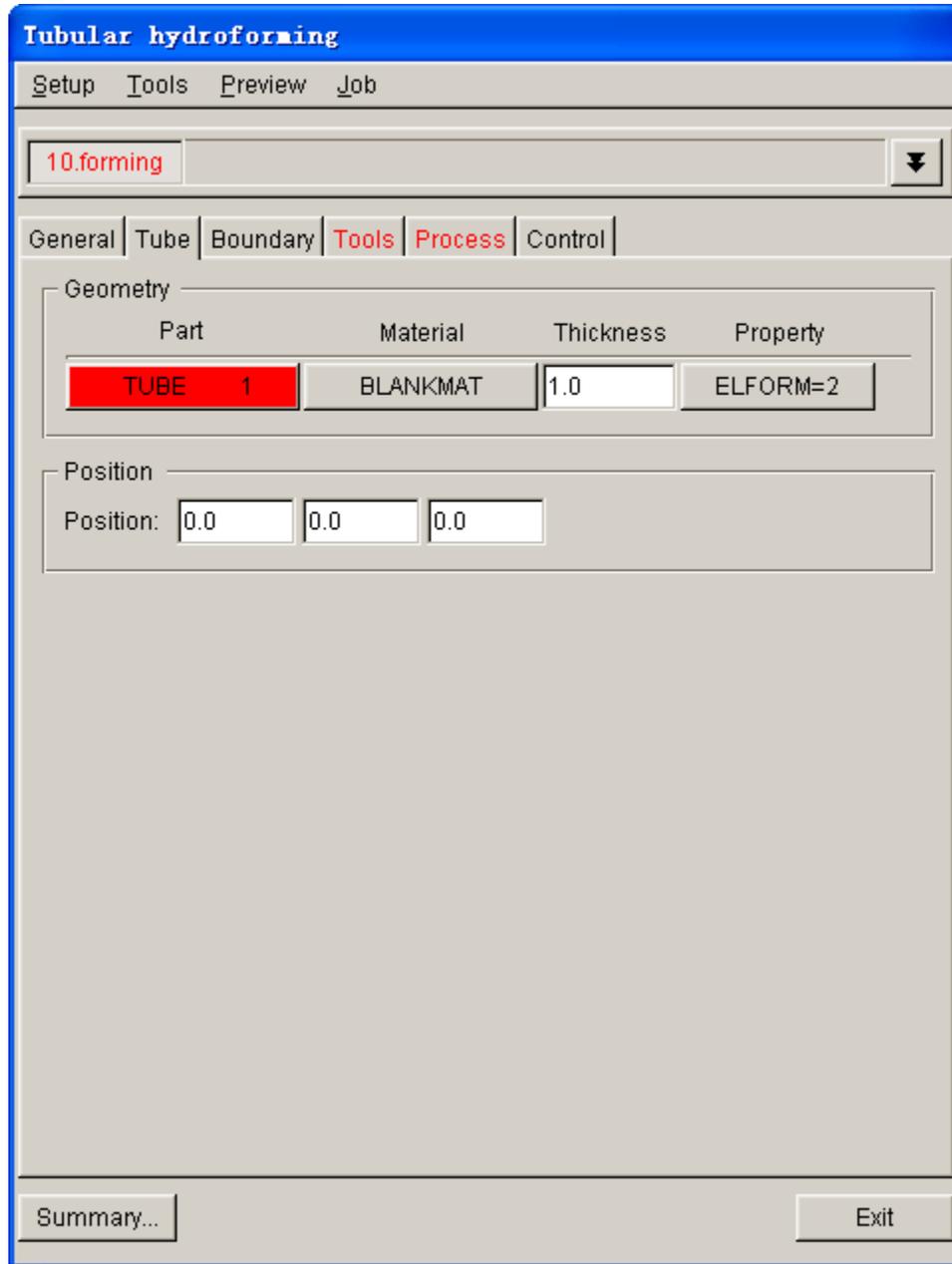


Figure 8.2.124 Tube definition

### 8.2.9.4 TOOLS

This function allows the user to assign parts as tools in hydroforming process. The interface of the tool definition is illustrated in Figure 8.2.125. For description of tool definition, refer to Section 8.2.4.4.

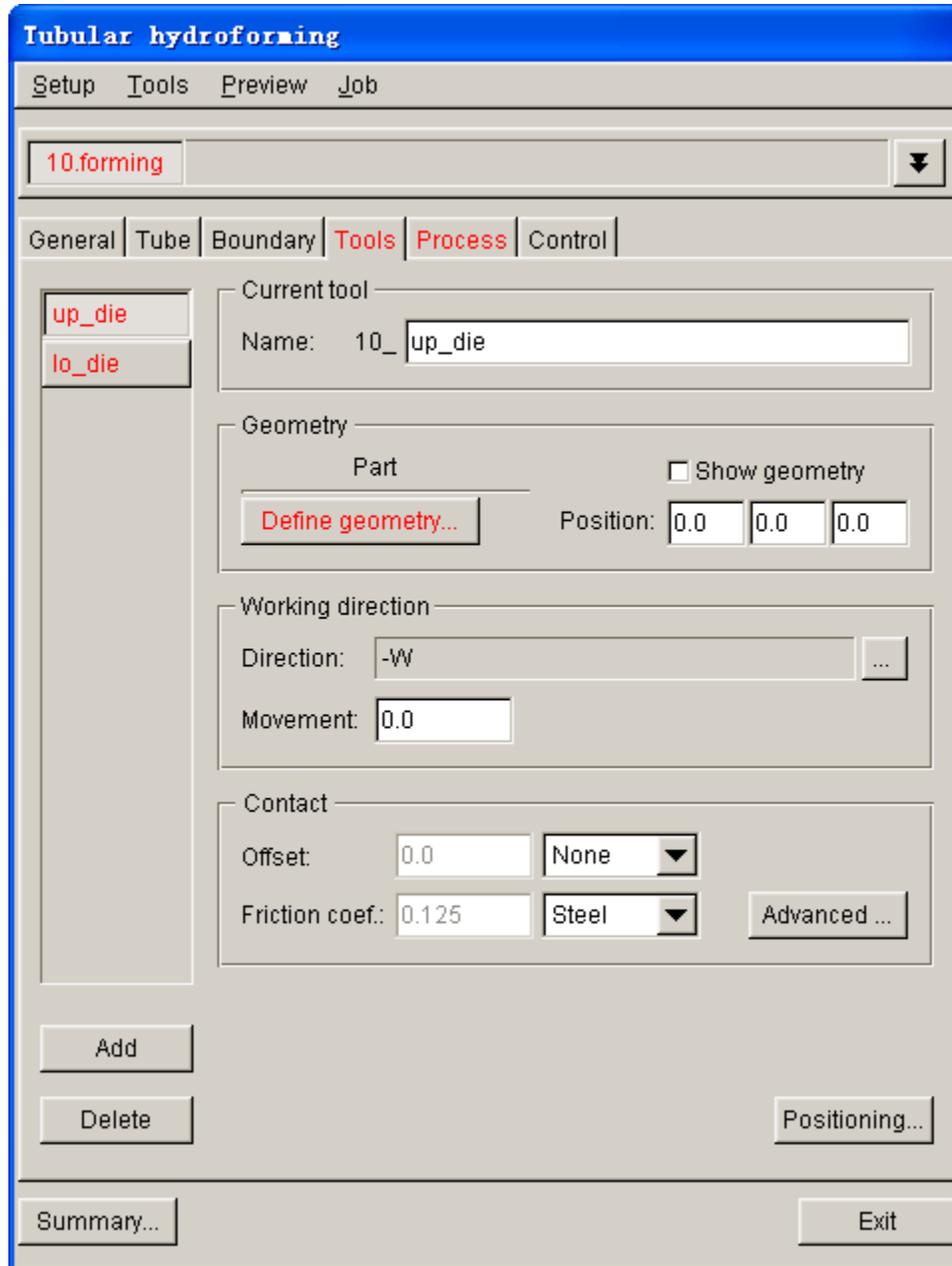


Figure 8.2.125 Tool Definition

### 8.2.9.5 PROCESS

This function allows the user to set process parameters for tubular hydroforming. The basic setting is similar to sheet hydroforming described in Section 8.2.6. For tubular hydroforming, the user may specify fluid flux to control fluid internal pressure. The setup interface is illustrated in Figure 8.2.126.

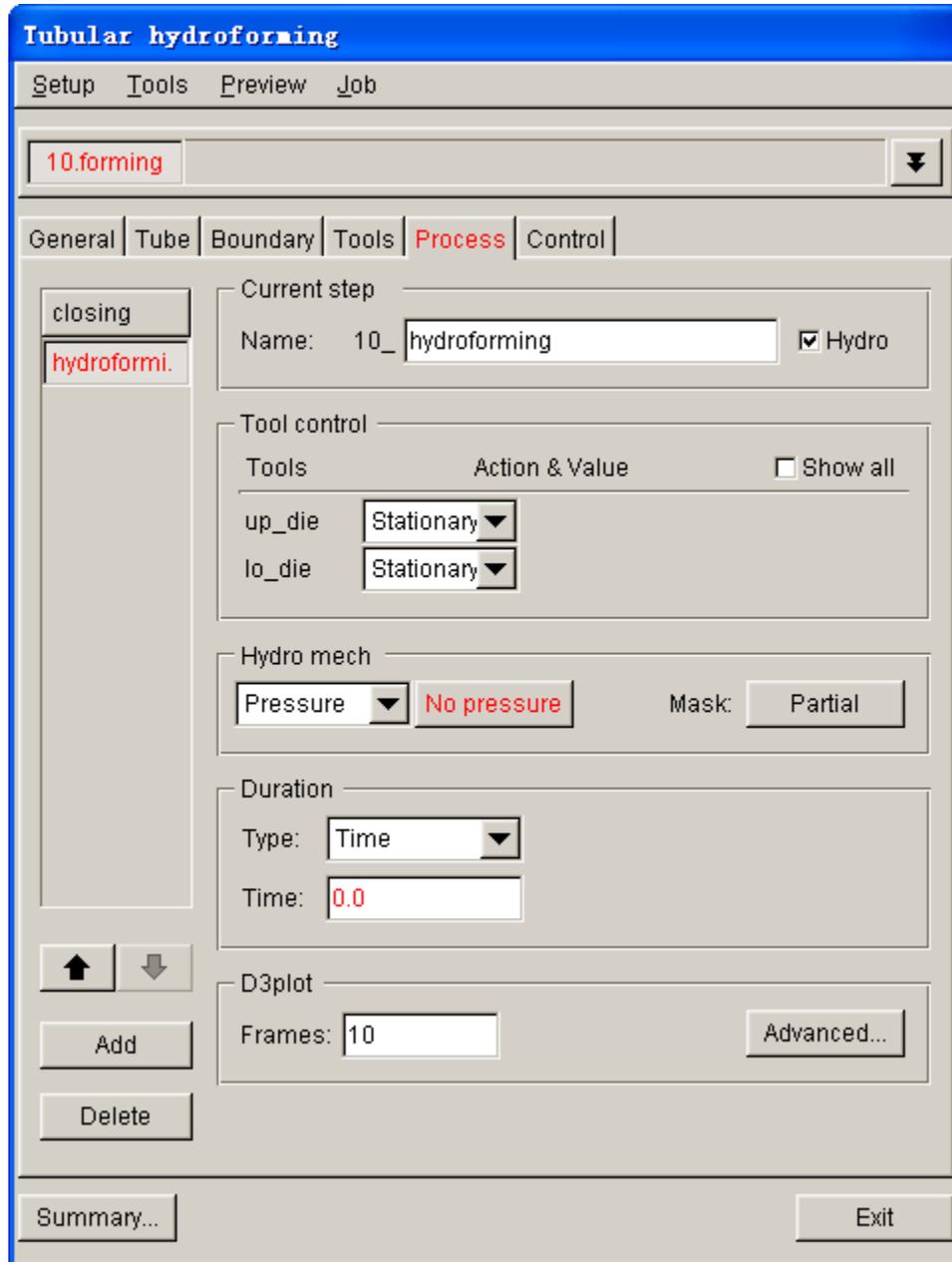


Figure 8.2.126 Hydroforming control parameters

The user needs to select VOLUME from the drop-down list in Hydro mech field.

**NO FLUX:** Allow the user to specify fluid flux vs. time curve to control internal fluid pressure. The dialog box is illustrated in Figure 8.2.127.

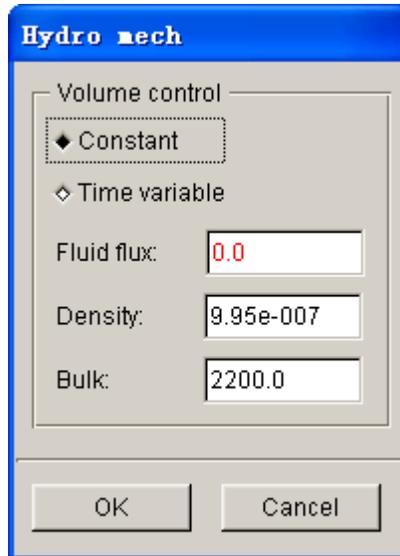


Figure 8.2.127 Fluid flux dialog box

**CONSTANT:** Define constant fluid flux. The user may key in a constant fluid flux in FLUID FLUX input data field.

**TIME VARIABLE:** Define time variable fluid flux. The user may click EDIT button to display the EDIT LOAD CURVE dialog box described in Chapter 12.

**FLUID FLUX:** Define flux of fluid.

**DENSITY:** Define density of fluid.

**BULK:** Defines bulk viscosity of fluid.

## Chapter 9

# SPRINGBACK COMPENSATION PROCESS

Springback Compensation Process (SCP) is a brand new module embedded in eta/DYNAFORM 5.6 for calculation of springback compensation of tools. Application of advanced sheet materials such as aluminum alloy, high strength and ultra high strength steel helps to reduce body weight of an automobile without sacrificing the safety performance. Increasing application of advanced sheet materials to manufacture body of automobile has also created several issues in metal stamping industry. One of the crucial issues is the springback phenomenon as a result of elastic recovery of sheet material and uneven stress distribution after forming.

Traditionally, springback may be resolved in the shop floor by overcrown, overbend, restrike, stiffening darts, etc. Today, stamping CAE software plays a significant role in the prediction of springback of sheet metal components and compensation for tooling to alleviate springback. Using stamping CAE software, the user can conduct springback analysis and morphing of part after a sequence of process such as draw, trimming and flanging. However, springback compensation techniques are needed for users to compensate the original tooling surface to resolve the springback phenomenon. For decades, the springback compensation is performed depending on the experience of engineers. With advanced development of computer hardware and software technology, springback compensation can be quickly performed using CAE software such as eta/DYNAFORM.

In eta/DYNAFORM, springback compensation can be performed to correct part shape after springback to archive the dimension tolerance of intended part design. As illustrated in Figure 9.1, springback compensation is carried out after series of forming simulation and springback analysis. Iterative compensation approach may be adopted to obtain reasonable compensated tool surface.

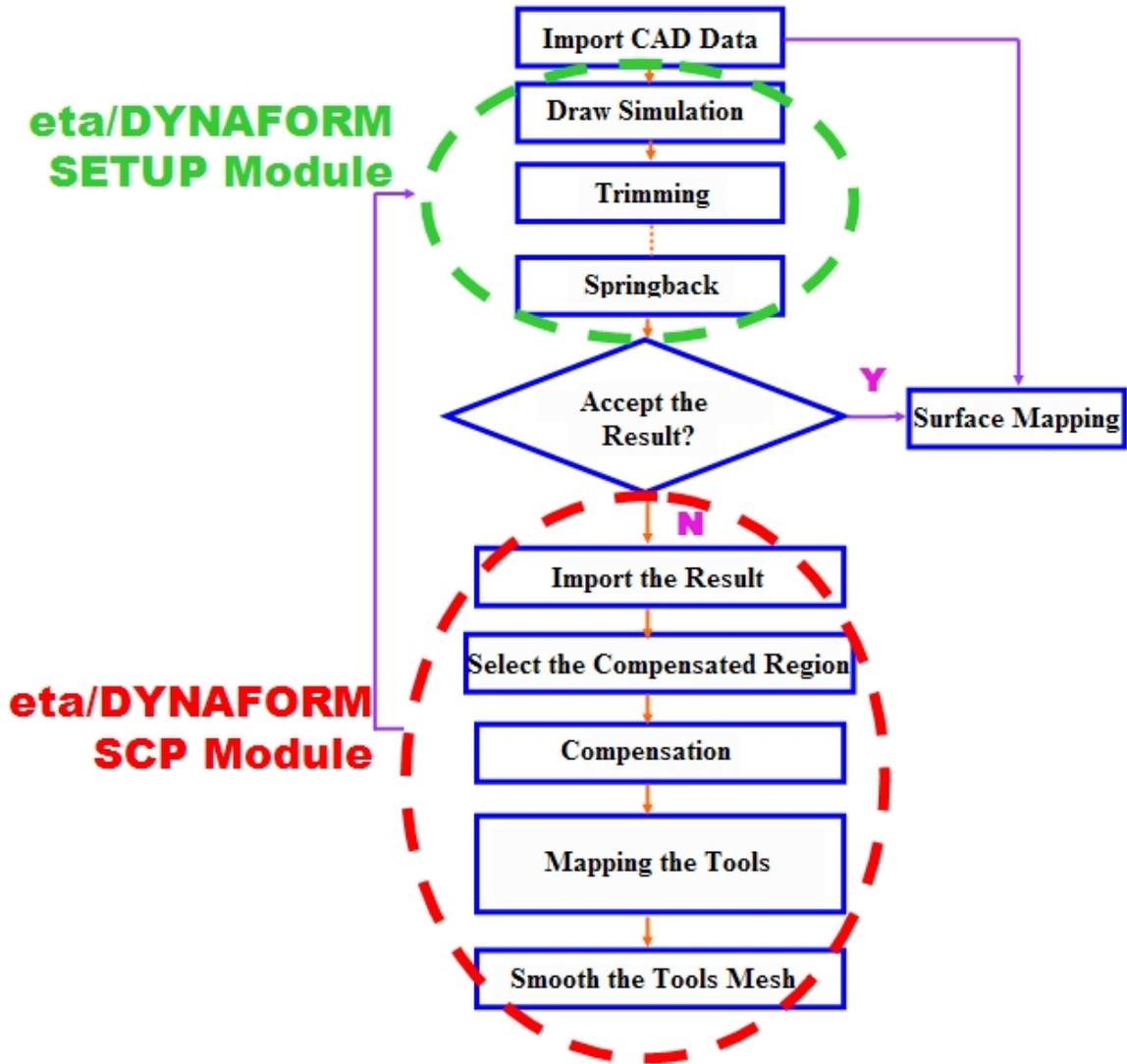


Figure 9.1 Springback Compensation Procedure

SCP menu illustrated in Figure 9.2 provides functions such as COMPENSATION, RESULT CHECK, TOPOLOGY REPAIR, MORPHING and SURFACE MAPPING. Detailed description of each function is given in the following sections.



Figure 9.2 SCP Menu

## 9.1 COMPENSATION

Prior to performing compensation, the user needs to import result files (DYNAIN format) of deformed sheet metal part before and after springback. The tool mesh of last forming step (if this is the first springback compensation, import the original tool mesh) is required for springback compensation. If the desired blank shape cannot be obtained after first compensation, several iterations may be performed to compensate the tooling. Springback compensation dialog box is illustrated in Figure 9.1.1. Click on each icon in the dialog box to import or assign the part. The color selected icon will be changed from red to green if the part is defined.

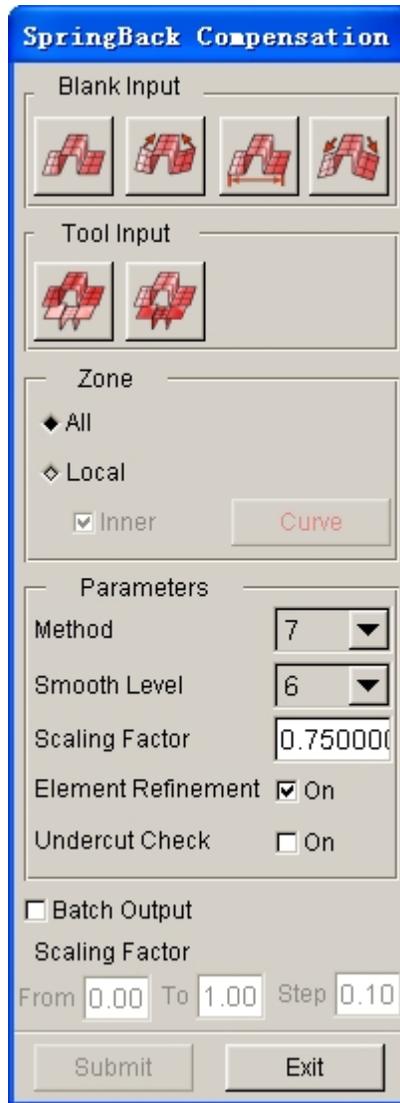


Figure 9.1.1 Springback Compensation dialog box

### BLANK INPUT

This field enables the user to import deformed parts (before and after springback) and assign parts for springback compensation.

**BEFORE SPRINGBACK**

Import or select the blank model before springback. The user can import file in dynain format.

**AFTER SPRINGBACK**

Import or select the blank model after springback. The user can import file in dynain format.

**DESIRED SHAPE**

Import or select the desired blank shape. The user can import file in dynain format. It is usually the shape before springback.

**COMPENSATED SHAPE**

Import or select the compensated blank shape. The user can import file in dynain or \*.tmp format. In first springback compensation iteration, it is the blank shape before springback. In second and subsequent iterations, the user should import disp.tmp file for the compensated part and define it as the compensated blank shape.

**TOOL INPUT****MASTER TOOLS**

Import or select master tools. The user can import file in \*.nas or \*.new format.

**SLAVE TOOLS**

Import or select slave tools. The user can import file in \*.nas or \*.new format.

**ZONE****ALL**

Select all regions of part for springback compensation.

**LOCAL**

Select local regions of part for springback compensation.

**INNER**

Use curve tool to define a region. If INNER option is selected, part area insider the curve will

be compensated. If INNER option is not selected, part area outside of curve will be compensated. The user can click on CURVE button to define curve.

## PARAMETERS

Compensation parameters consist of METHOD, SMOOTH LEVEL, SCALING FACTOR, ELEMENT REFINEMENT AND UNDERCUT AVOIDANCE. Refer to LS-DYNA User's Manual for detailed description of each parameter.

### METHOD

LS-DYNA provides 8 methods for springback compensation. The recommended methods are 7, 3 and 6, and the other methods are seldom used. Each method has its advantages and disadvantages. The three commonly used methods are described below:

#### Method 7

This is a non-linear method and the iterative method is used. If the punch is chosen as the master side, the binder will not be changed. The only change occurring involves inside punch opening. Under this option, the smoothing factor has little effect. The smoothness of the modified tool depends on the magnitude of the springback and the size of the addendum region.

Advantages: The binder will not be changed.

Disadvantages: The change will be limited inside the addendum region, and the modified surface may not be smooth if the springback magnitude is large and the transition is small.

#### Method 6

This is a non-linear method and the iterative method is used. The smoothness and the transition region of the modified surface will depend on the springback magnitude and the smoothing factor. If the springback magnitude is large, the transition region will be increased automatically. On the other hand, the transition region will be smaller if the springback magnitude is small. At the same time, a larger smoothing factor will result in a smaller transition region.

Advantages: The smoothness of the modified surfaces can be controlled.

Disadvantages: It is impossible to limit the transition region, and the binder surface could change if the springback is large.

#### Method 3

Similar to Method 6, however, it is a linear method.

### SMOOTH LEVEL

The smooth level parameter controls the smoothness of the modified surfaces. A large value makes the surface smoother. The commonly used value is between 5 and 10. If springback is large, the transition region is expected to be large. However, by using a smaller value of smooth level, the region of transition can be reduced.

### SCALING FACTOR

The scales how much of the shape deviation is compensated. If this factor is appropriate, the user can obtain the desired result with only a few repeated calculations. Otherwise, plenty times of modification are required. The product of this scaling factor and springback quantity is the compensation quantity for reverse direction. The default is 0.75.

### ELEMENT REFINEMENT

Refine the tool mesh.

**UNDERCUT**

Automatically check the element with undercut generated after springback compensation.

**BATCH OUTPUT**

This function allows the user to output a series of input decks with different scaling factors. The input decks are stored in different folders. If this option is activated, the user can key the range and increment of scaling factor.

**SUBMIT**

After all parameters are defined, the user can click on SUBMIT button to display the SUBMIT JOB dialog box illustrated in Figure 9.1.2. Refer to the relevant sections for description about methods for submitting jobs.



Figure 9.1.2 Submit Job dialog box

**EXIT**

Exit springback compensation dialog box.

## 9.2 RESULT CHECK

Upon completion of springback compensation, the user can import compensated tool for undercut check. The Undercut Check dialog box is illustrated in Figure 9.2.1.



Figure 9.2.1 Undercut Check

**ADD:** Add the parts which require undercut check to the list. The user can import the disp.tmp and rigid.new files after springback compensation and add them into the list.

**REMOVE:** Remove parts from the list.

**REVERSE:** Reverse normal of the selected part.

**MARGINAL:** Yellow indicates a marginal area with a wall angle of approximately 0~3°.

**SEVERE:** Red indicates severe undercut where the backdraft is less than 0°.

**PLOT:** Plot the undercut contour in display screen. The user can adjust model according to this plot.

**EXIT:** Exit Undercut Check dialog box.

## 9.3 TOPOLOGY REPAIR

This function allows the user to perform topology repair on the mesh which is imported and with adaptive mesh before and after springback. A comparison of mesh before and after topology repair is illustrated in Figure 9.3.1. Since this function is same with Mesh Topology Repair function in DFE menu, the user can refer to Section 6.1.1.9 for detailed description.

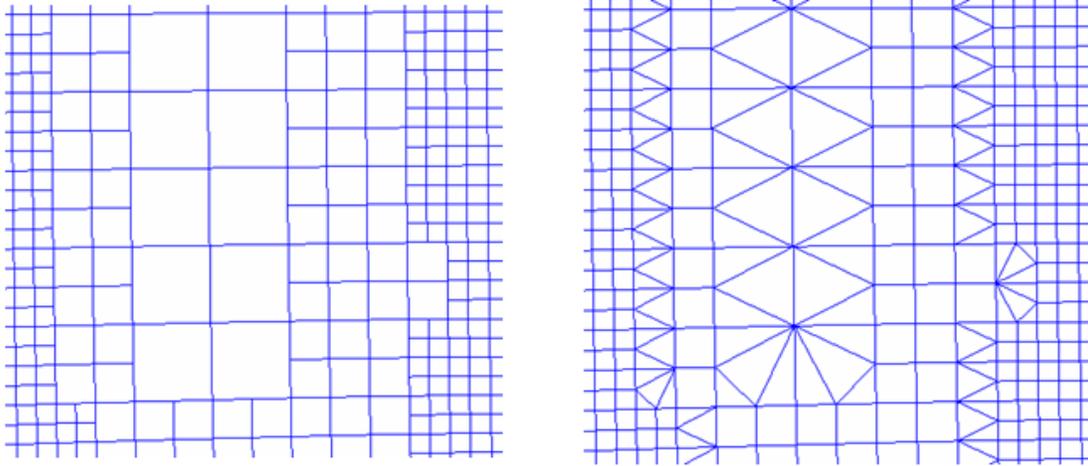


Figure 9.3.1 A comparison of mesh before and after topology repair

## 9.4 MORPHING

This function allows the user to morph selected elements. The user can morph the tool meshes to smooth the discontinue mesh after springback compensation. Refer to Section 6.5 for detailed description about element morphing.

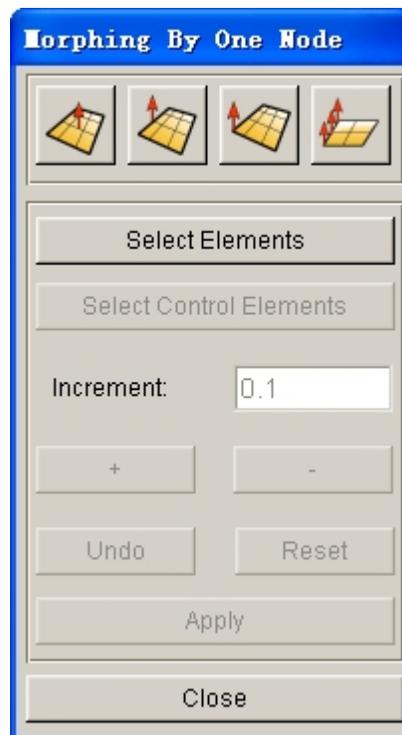


Figure 9.4.1 Element Morphing dialog box

## 9.5 SURFACE MAPPING

The Surface Mapping dialog box illustrated in Figure 9.5.1, enables the user to map original die face to the compensated tool mesh after the SCP of desired part is obtained.

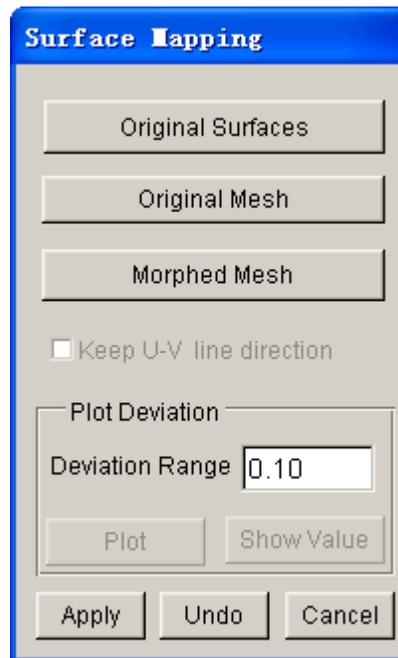


Figure 9.5.1 Surface Mapping dialog box

### ORIGINAL SURFACE

Allow the user to select the original die surfaces for mapping operation.

### ORIGINAL MESH

Allow the user to select the original tool mesh which is generated using original die surfaces.

### MORPHED MESH

Allow the user to select the tool mesh generated from springback compensation.

### KEEP U-V LINE DIRECTION

Allow the user to keep the U-V line direction of the mapped surface same as the original surface. This option is toggled on by default.

### PLOT DEVIATION

Allow the user to display the contour of distance deviation between the mesh node and mapped surfaces, as illustrated in Figure 9.5.2.

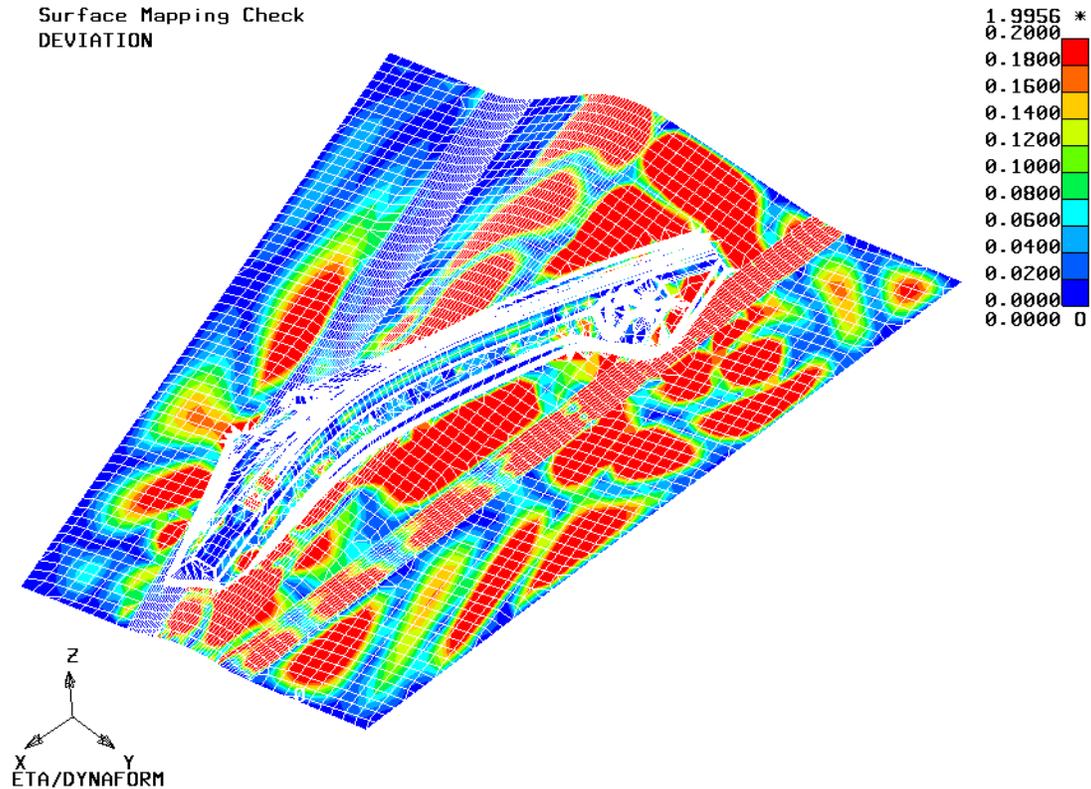


Figure 9.5.2 Contour Plot for Deviation between Mapped Surface and Mesh Node

#### DEVIATION RANGE

Set the deviation range. The default range is from 0 to 0.1. If the maximum value displayed in the plot is significant, the user can adjust the color display range.

#### PLOT

Select this option to display the deviation contour. The node with maximum deviation is marked with \*. The node with minimum deviation is marked with 0.

#### SHOW VALUE

This function allows the user to select a node on the screen by mouse cursor, following by displaying the deviation for this node on the display screen. This function enables the user to easily measure deviation for each node as illustrated in Figure 9.5.3.

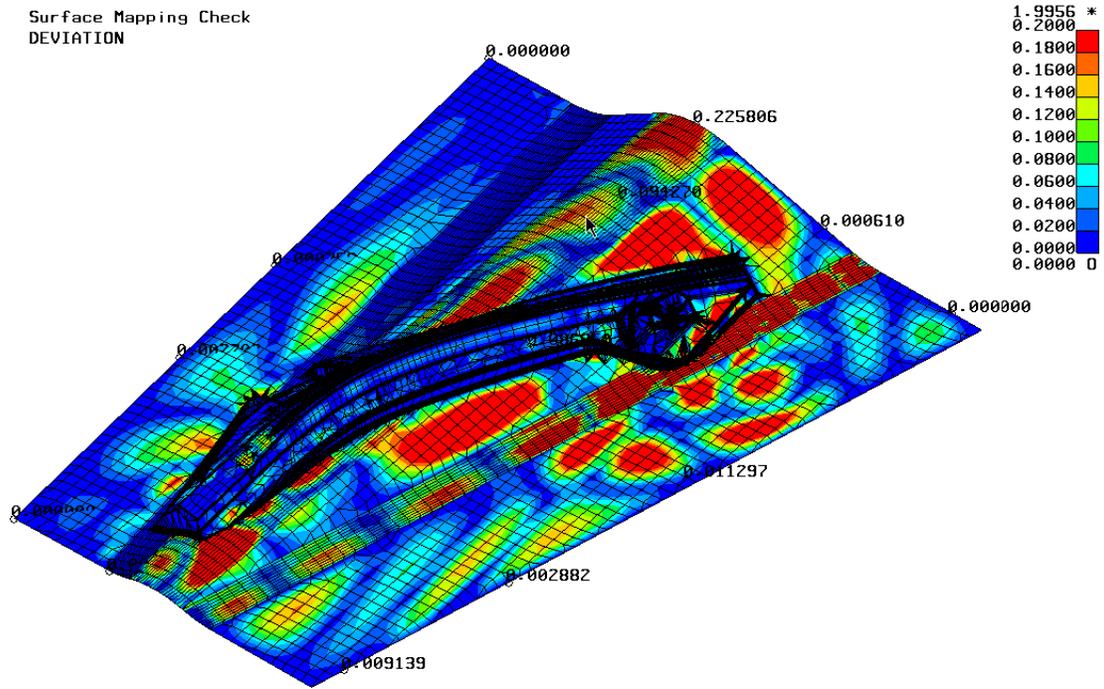


Figure 9.5.3 Measure deviation for node

## Chapter 10

# TOOL DEFINITION

The functions provided in this menu are shown in Figure 10.1. 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.

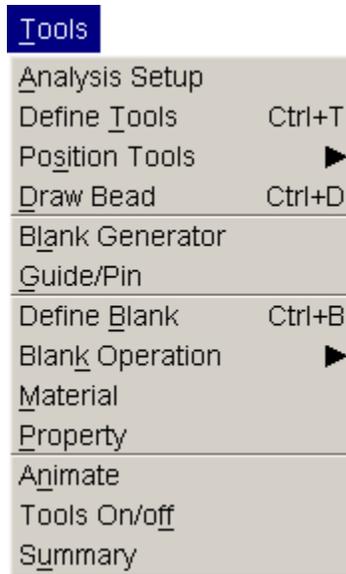


Figure 10.1 Tool Menu

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

## 10.1 ANALYSIS SETUP

This function enables the user to define forming parameters for the LS-DYNA ANALYSIS. The dialog box is shown in Figure 10.1.1.

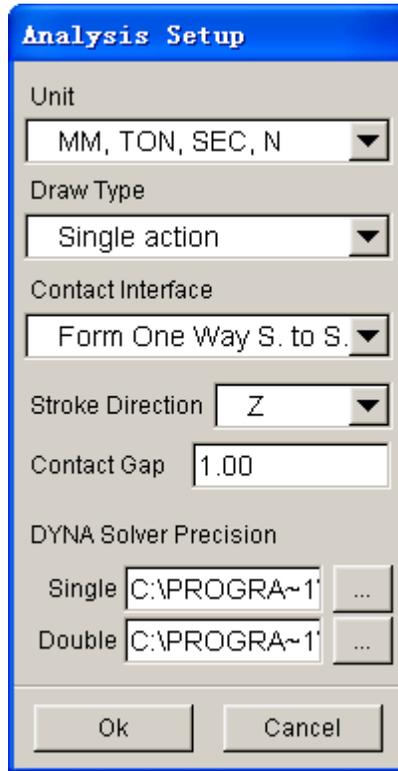


Figure 10.1.1 The Analysis Setup dialog box

### UNIT

The user can select a UNIT system by clicking the drop down button of UNIT. As shown in Figure 10.1.2, four types of UNIT systems are available. The default UNIT system is set as <MM, TON, SEC, N>.



Figure 10.1.2 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 illustrated in Figure 10.1.3.

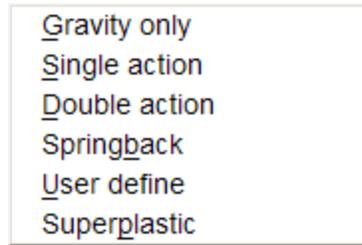


Figure 10.1.3 The Draw Types drop down menu

**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.

**SUPERPLASTIC**

Simulate superplastic forming process.

**NOTE:** *The defined DRAW TYPE works in conjunction with both the BLANK OPERATION/AUTO POSITION and TOOL/AUTO POSITION functions.*

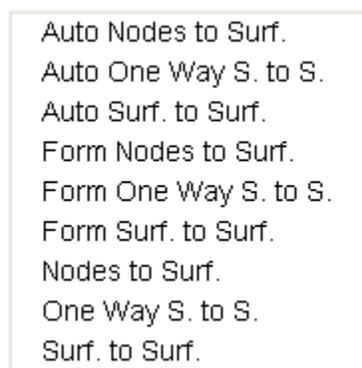


Figure 10.1.4 Type of contact interfaces

**CONTACT INTERFACE**

There are several types of contact interface available for an LS-DYNA stamping simulation, as listed in Figure 10.1.4. The user can set type of contact interface between all tooling and blank. The default contact interface between blank and 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 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 path of Single and Double Precision solver. User can click the  button to change the path.

## 10.2 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 10.2.1, provide options for using STANDARD TOOLS setup and/or USER DEFINED TOOLS setup.

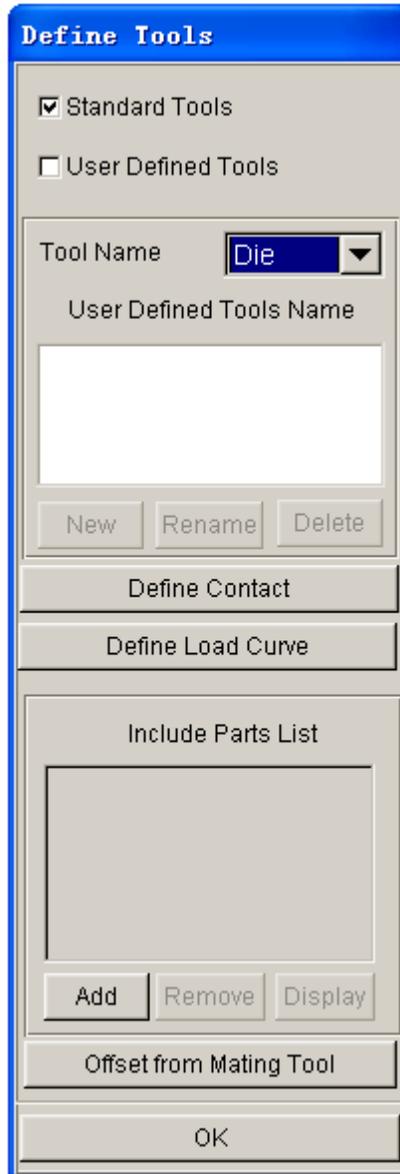


Figure 10.2.1 Define Tools dialog box

### 10.2.1 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 10.2.1.

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 10.2.2 User Defined Tool Name dialog box

### **RENAME**

Edit existing tool name.

### **DELETE**

Remove selected tools from the USER DEFINED TOOL list.

## **10.2.2 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 4.6 for more information. The selected parts are listed in the INCLUDE PARTS LIST window illustrated in Figure 10.2.1.

## **10.2.3 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.

## **10.2.4 DISPLAY (TOOL)**

This function enables the user to show all the parts included in the current tool definition in the display screen. To display the part's current tool, click on the DISPLAY button.

## **10.2.5 DEFINE CONTACT**

This function is used to define the contact interface between blank and a tool. In most cases, \*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 illustrated in Figure 10.2.1 to display the TOOLS CONTACT dialog box as shown in Figure 10.2.3.

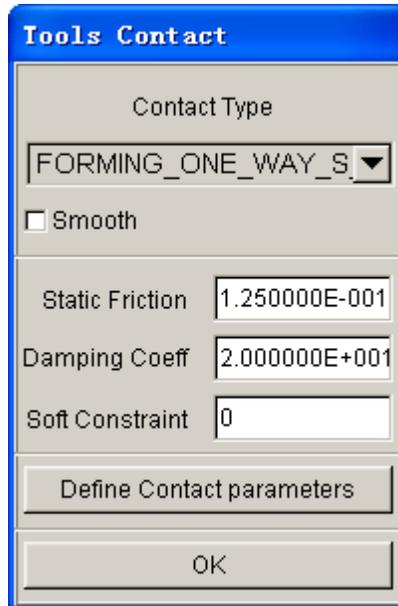


Figure 10.2.3 Define Contact dialog box

**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

If SMOOTH option is selected, the program will automatically add SMOOTH option behind the selected contact type and add the corresponding card.

**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.*

## 10.2.6 DEFINE LOAD CURVE

This function allows the user to define, read in, or modify MOTION and/or FORCE load curves. As shown in Figure 10.2.4, 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).

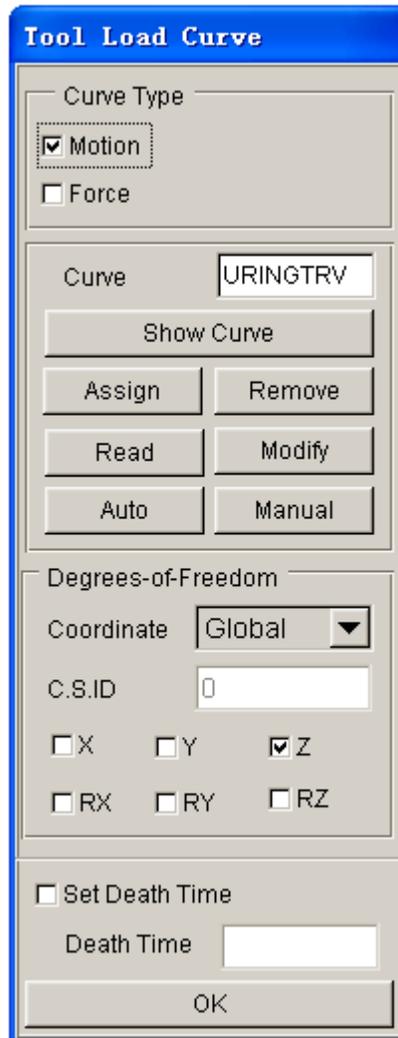


Figure 10.2.4 Tool Load Curve dialog box

### 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 10.2.5, the user can select a particular load curve from the list. Click on the OK to button to confirm the selection and to complete the ASSIGN CURVE operation.

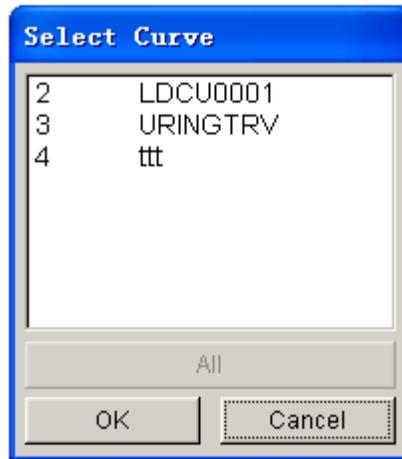


Figure 10.2.5 List Curve

## READ

Allow the user to read in an external load curve using the window in Figure 10.2.6.

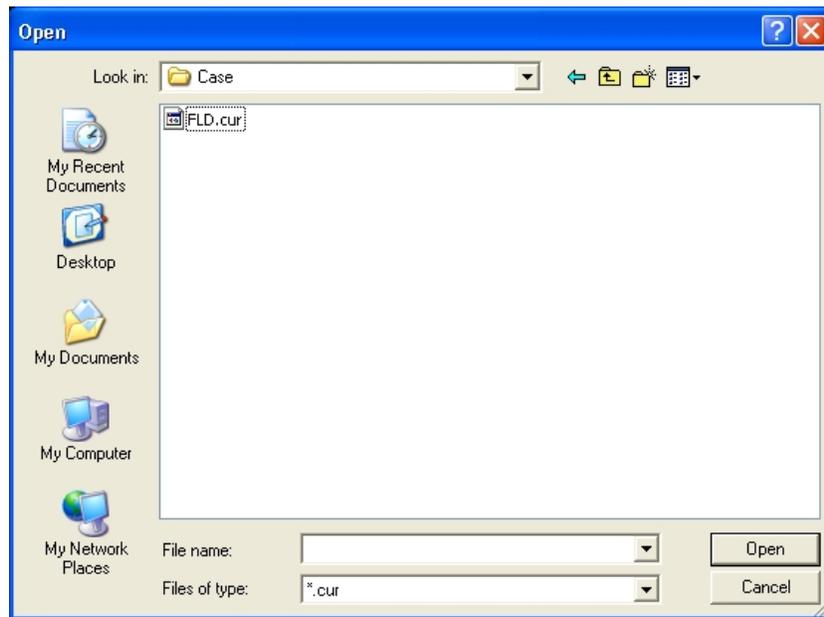


Figure 10.2.6 Read Curve File

## AUTO

This function is used to generate a velocity/displacement/force load curve. If the MOTION option is checked, the MOTION CURVE dialog box illustrated in Figure 10.2.7 is displayed.

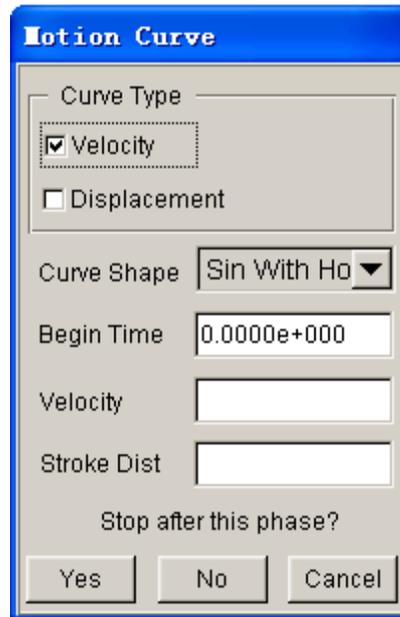


Figure 10.2.7 Auto Motion Curve dialog box

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 for a period of time.

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 10.2.8) is displayed.

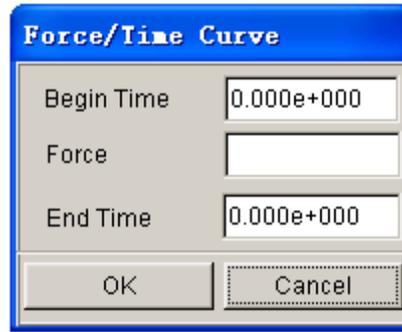


Figure 10.2.8 Auto Force Curve dialog box

- Input values in the data fields of BEGIN TIME, FORCE, and END TIME. The default end time is the end time of motion curve.
- Select OK to create the force curve. A dialog box showing the force curve is displayed. 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 10.2.9) is displayed. The user selects either YES/NO to accept/reject the operation.

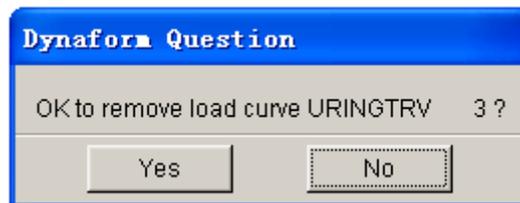


Figure 10.2.9 Remove Warning

### 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 10.2.10.

The steps listed below are 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.

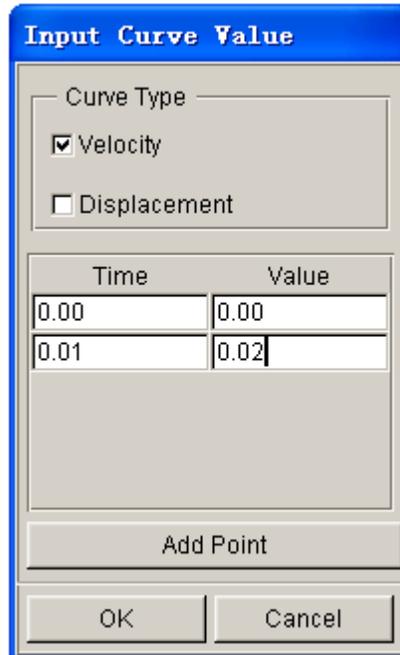


Figure 10.2.10 Input Curve dialog box

### 10.2.7 GENERATE TOOL FROM MATING TOOLS

This function is used to generate mating tool mesh from an existing tool mesh. It combines the operations of copying or offsetting elements, creating parts, and adding parts to the current tool definition.

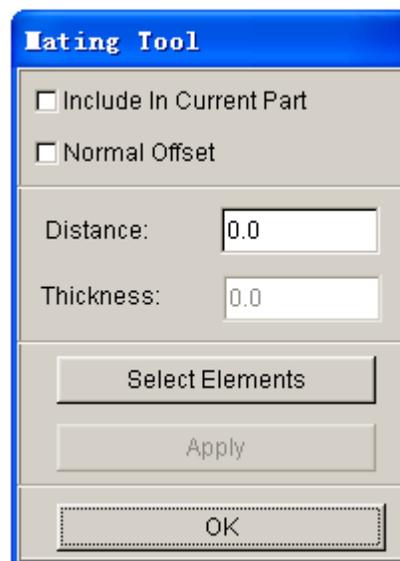


Figure 10.2.11 Mating Tool dialog box

- Select OFFSET FROM MATING TOOL to display the MATING TOOL dialog box (as shown in Figure 10.2.11).
- Toggle the INCLUDE IN CURRENT PART checkbox on/off. If the option is toggled on, the new copied or offset elements are 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, hit APPLY button.
- Select OK to exit the dialog box.

## 10.3 POSITION TOOLS

The POSITION TOOLS menu contains a submenu that offers the following operations:

- AUTO POSITION
- MOVE TOOL
- MIN. DISTANCE

### 10.3.1 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 10.1, ANALYSIS SETUP, to edit the default setting.

### 10.3.2 MOVE TOOL

This function allows the user to translate any tool in any direction of current C.S (Global or User Defined). The dialog boxes are shown in Figures 10.3.1 and 10.3.2 for Global and Local C.S, respectively. To move the tool, the user selects the target tool from the tool list, as well as direction. Key in the distance for translation following by hitting APPLY button to complete the translation. The user can hit REVERSE button to translate the tool in the opposite direction.

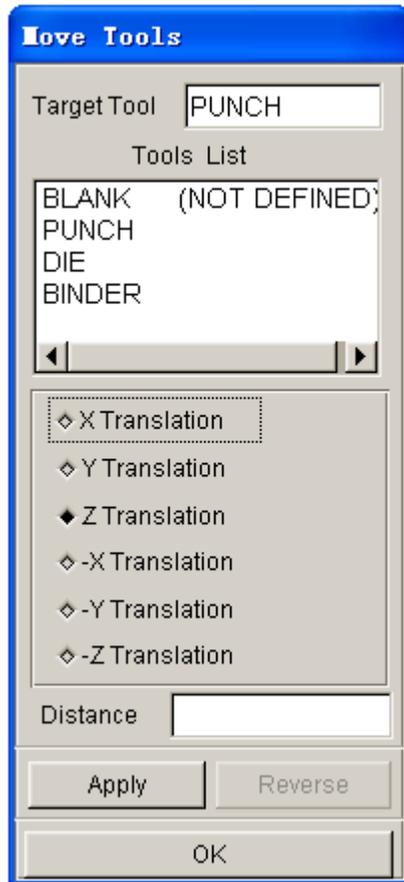


Figure 10.3.1 Global C.S. is Current C.S.

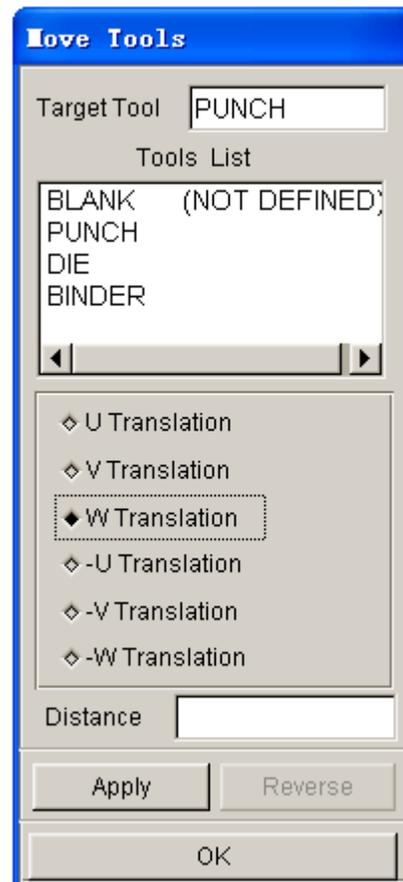


Figure 10.3.2 Local C.S. is Current C.S.

### 10.3.3 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 illustrated in Figure 10.3.3.

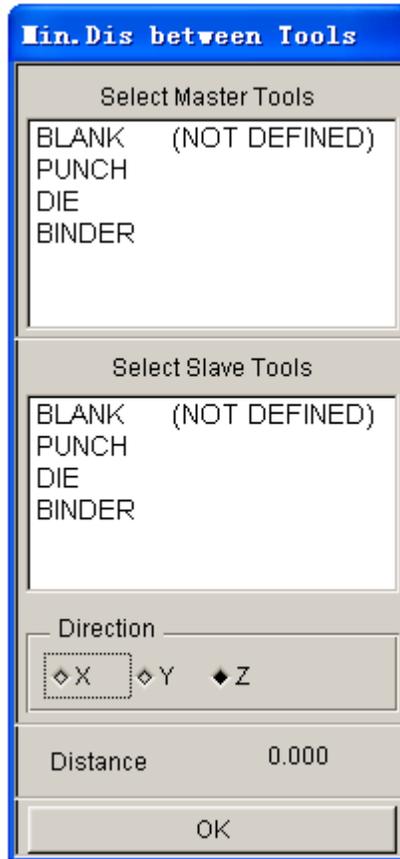


Figure 10.3.3 Min. Distance dialog box

## 10.4 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 10.4.1.

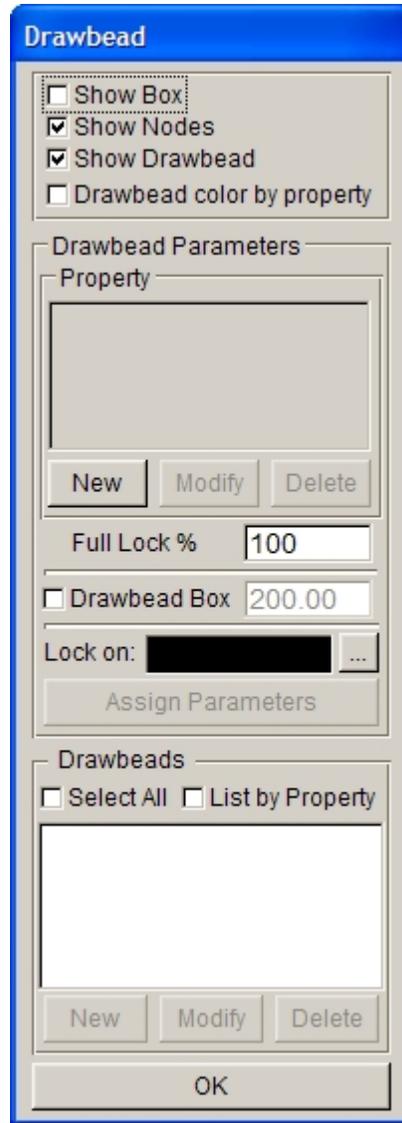


Figure 10.4.1 Define Drawbead dialog box

### 10.4.1 DRAWBEAD DISPLAYING OPTION

The user can select options illustrated in Figure 10.4.2 to display the drawbeads.

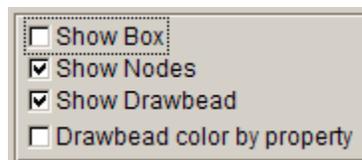


Figure 10.4.2 Drawbead Displaying Options

### 10.4.1.1 SHOW BOX

This option allows the user to show drawbead box on the display screen, as illustrated in Figure 10.4.3.

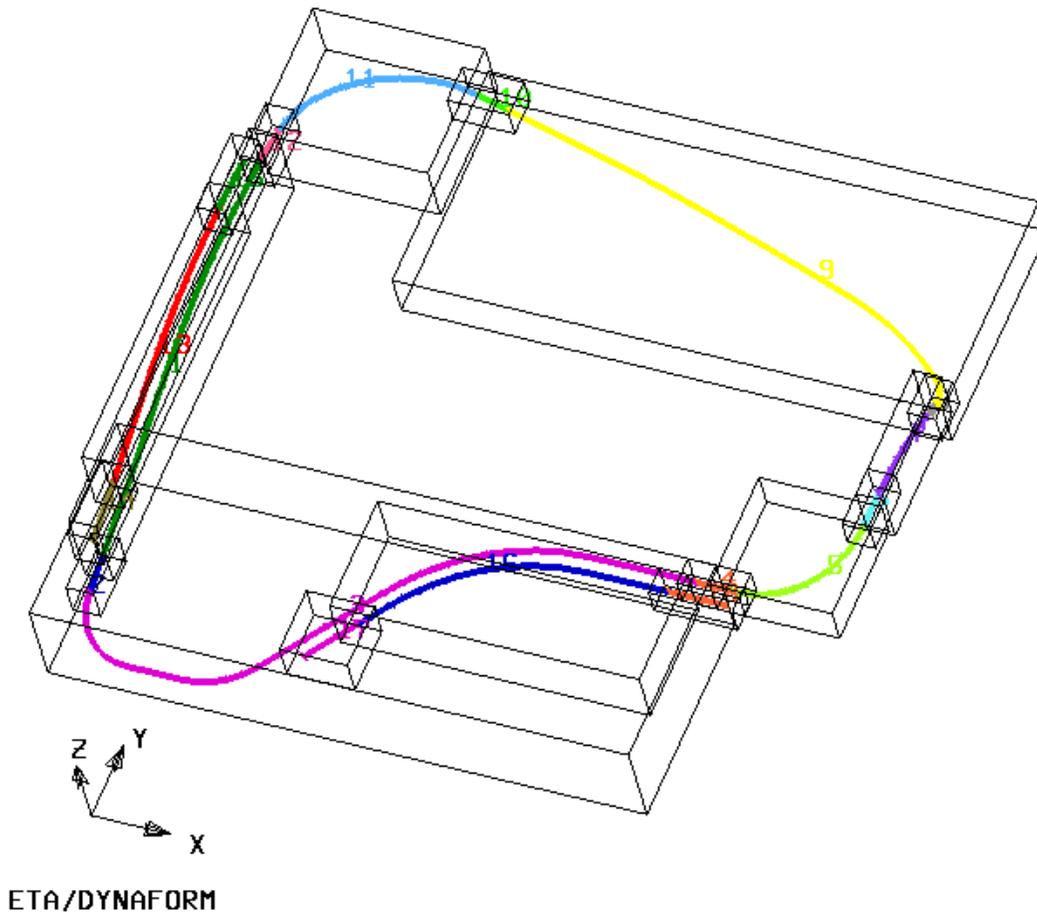


Figure 10.4.3 Show drawbead Box

### 10.4.1.2 SHOW NODES

This option allows the user to show all nodes of drawbeads on the display screen. This option is toggled ON by default.

### 10.4.1.3 SHOW DRAWBEAD

This function allows the user to display all drawbeads on the display screen. This option is toggled ON by default.

### 10.4.1.4 DRAWBEAD COLOR BY PROPERTY

This option allows the user to display drawbeads with either property color or color of drawbead lines.

## 10.4.2 DRAWBEAD PARAMETERS

The user can select options in this field to create drawbead property, set size of drawbead box, lock

drawbead to a specified part and assign drawbead parameters. The Drawbead Parameters field is illustrated in Figure 10.4.4.

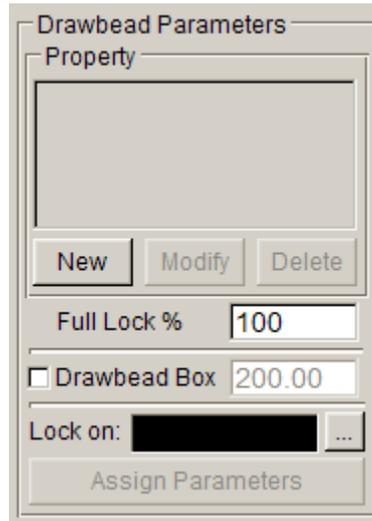


Figure 10.4.4 Drawbead Parameter field

### 10.4.2.1 CREATE PROPERTY

In LS-DYNA, each drawbead carries its own property. Prior to creating the drawbead, the user must define the property accordingly. In the create property field, NEW, EDIT, DELETE functions enable the user to create, modify or delete the property, respectively.

- NEW PROPERTY

Click on NEW button to display the DRAWBEAD PROPERTY dialog box illustrated in Figure 10.4.5.

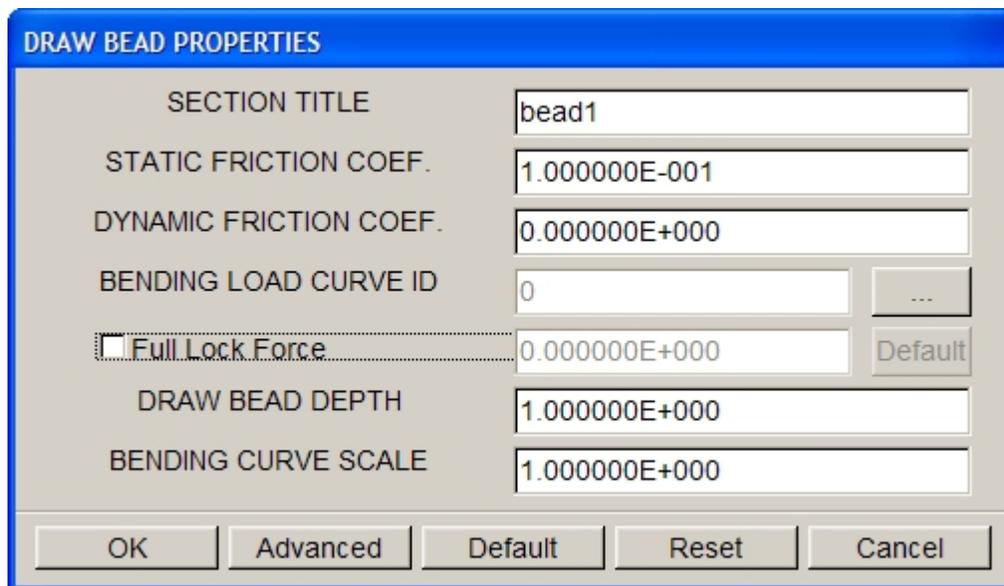


Figure 10.4.5 Drawbead properties dialog box

### SECTION TITLE

Allow the user to name title of 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 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 drawbead. The default dynamic coefficient of friction is 0. It means the dynamic friction between the blank and drawbead is ignored.

#### BENDING LOAD CURVE ID

Allow the user to define the  $F_{\text{bending}}$  component of the drawbead restraining force. The  $F_{\text{bending}}$  component is referred as force per unit drawbead length as a function of displacement as illustrated in Figure 10.4.6.

**NOTE:** The **NORMAL FORCE** of drawbeads is automatically set to  $1.0e-6$  by eta/DYNAFORM. Hence, only the **BENDING FORCE** needs to be defined by the user.

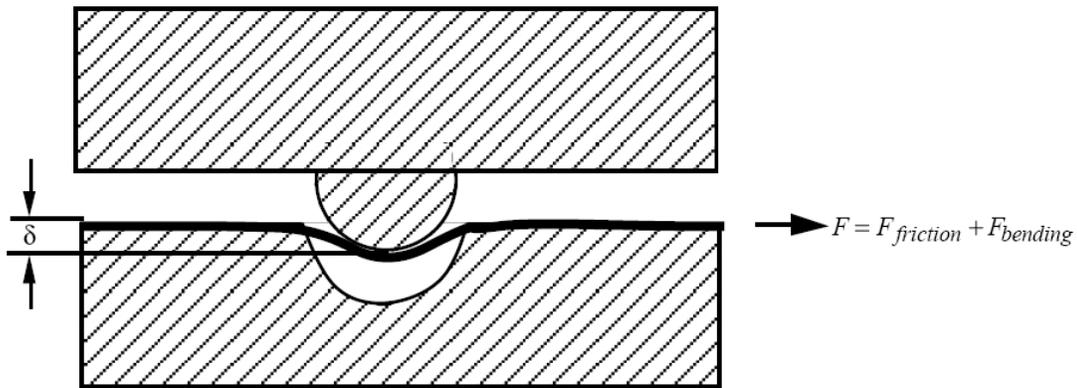


Figure 10.4.6 Drawbead resisting force as a function of the drawbead displacement

The user can click on the  button to define the bending curve of drawbead property using the DRAWBEAD LOAD CURVE dialog box illustrated in Figure 10.4.7. 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 10.4.8. 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 10.4.9. Please refer Section 12.13.1 for detailed information about load curve definition.

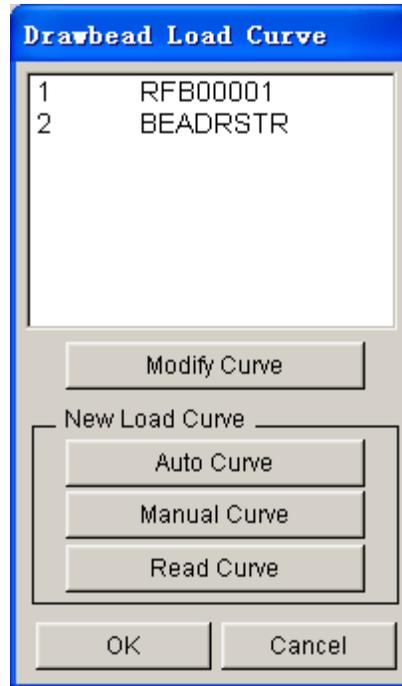


Figure 10.4.7 Drawbead Load Curve dialog box

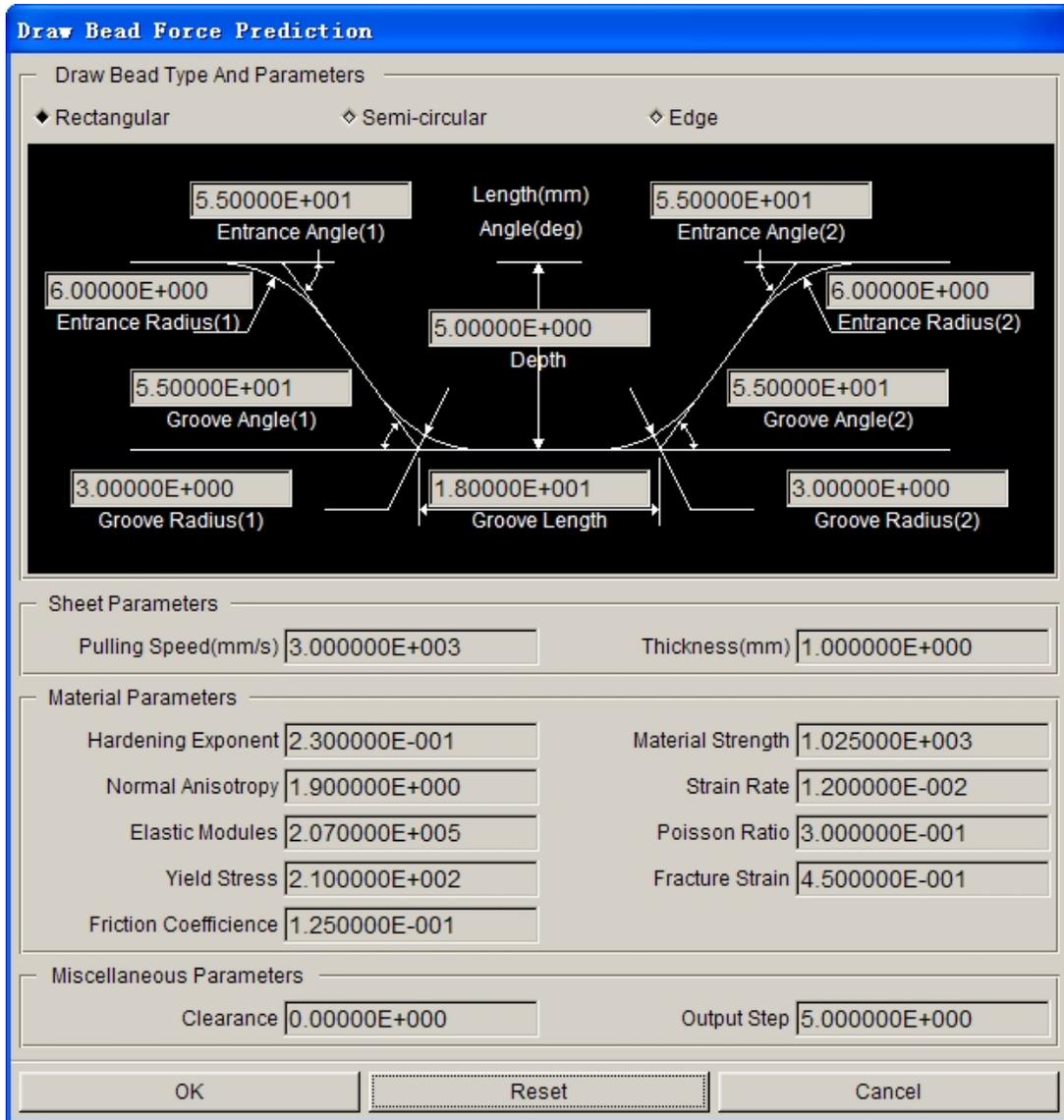


Figure 10.4.8 Drawbead Force Prediction dialog box

X	Y
0	0
0.01	1

Figure 10.4.9 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 depth of the drawbead. The default depth of drawbead is set as 1.0 (mm).

**BENDING CURVE SCALE**

Allow the user to set scale factor for the BENDING CURVE. The default scale factor is set as 1.0 (or 100%).

After each new property is created, it is added to the property list and set as the current property.

- **MODIFY PROPERTY**

Modify current property.

- **DELETE PROPERTY**

Remove current property.

**Note:** The current property can be deleted only if no drawbead is assigned. Otherwise, the program prompts:

*The property assigned to drawbead. Cannot be deleted.*

### 10.4.2.2 FULL LOCK%

This function enables the use to change the scale factor of bending curve of a selected drawbead. Prior to changing the percentage value, the user must select one or multiple drawbead.

### 10.4.2.3 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 bead and 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 APPLY button 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.*

### 10.4.2.4 LOCK ON

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 illustrated in Figure 10.4.10. The user selects a target part to lock the drawbead(s). Normally, DIE or BINDER is selected as target part.

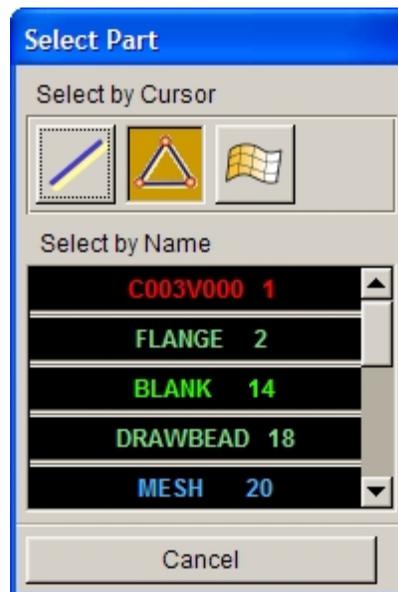


Figure 10.4.10 Select Part dialog box

After target part is selected, a eta/DYNAFORM QUESTION dialog box illustrated in Figure 10.4.11 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.

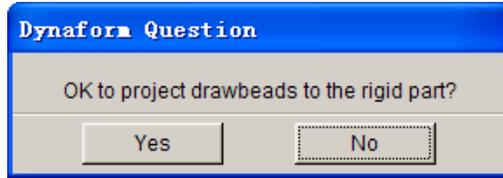


Figure 10.4.11 eta/DYNAFORM Question dialog box

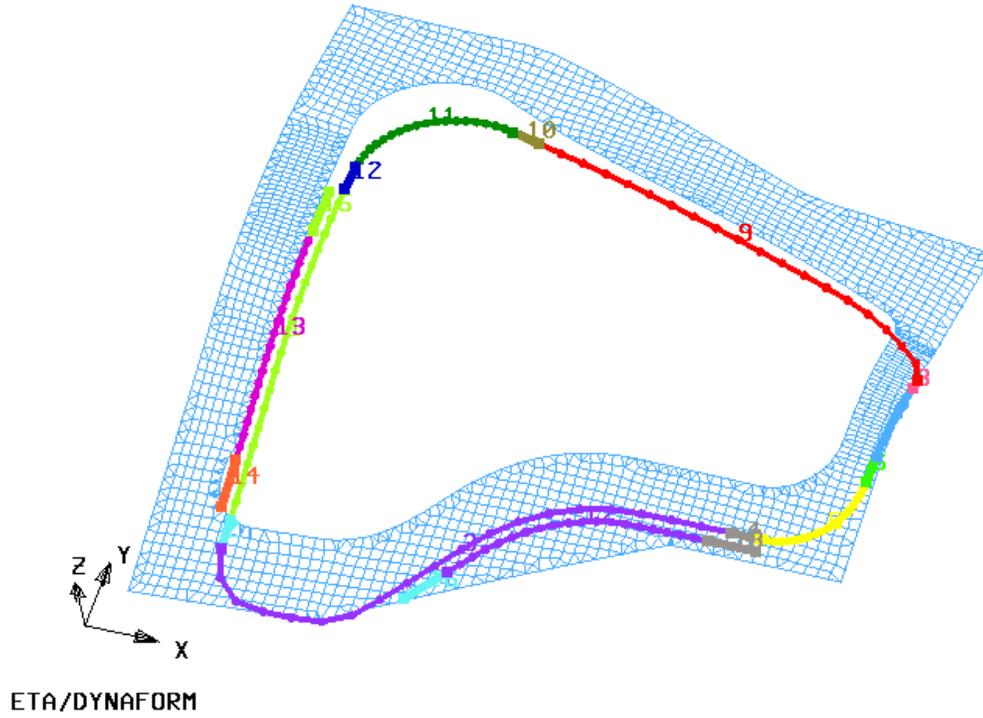
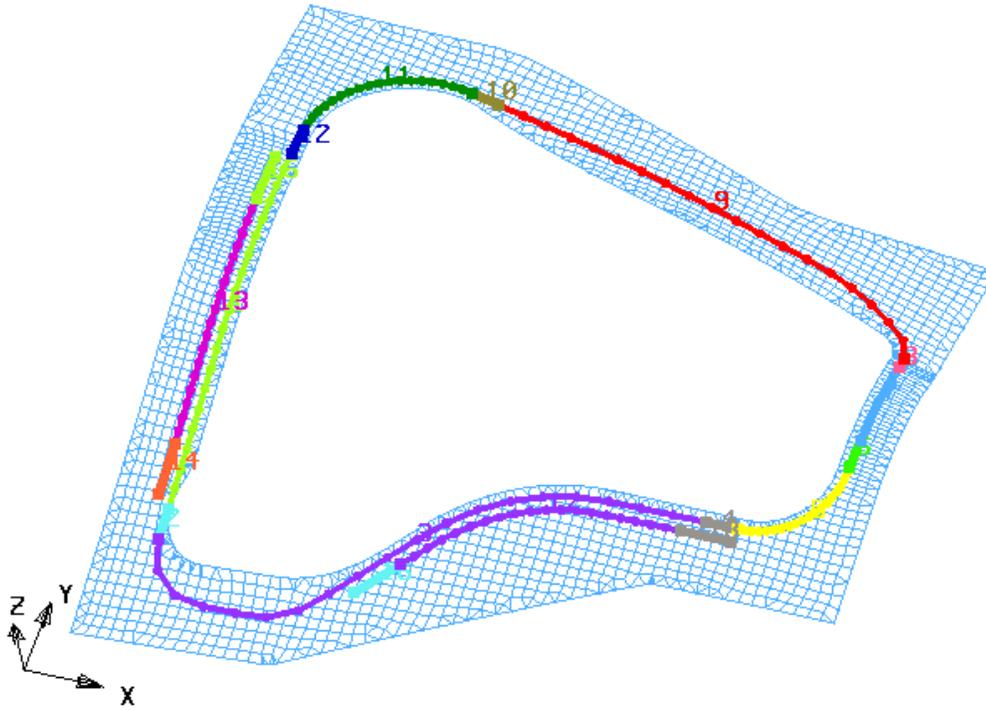


Figure 10.4.12 Before locking drawbeads on BINDER



ETA/DYNAFORM

Figure 10.4.13 After locking drawbeads on BINDER

After drawbead is locked on a target part, the name of target part is displayed in the LOCK ON box (as illustrated in Figure 10.4.14) whenever a drawbead is selected from the Drawbead list.

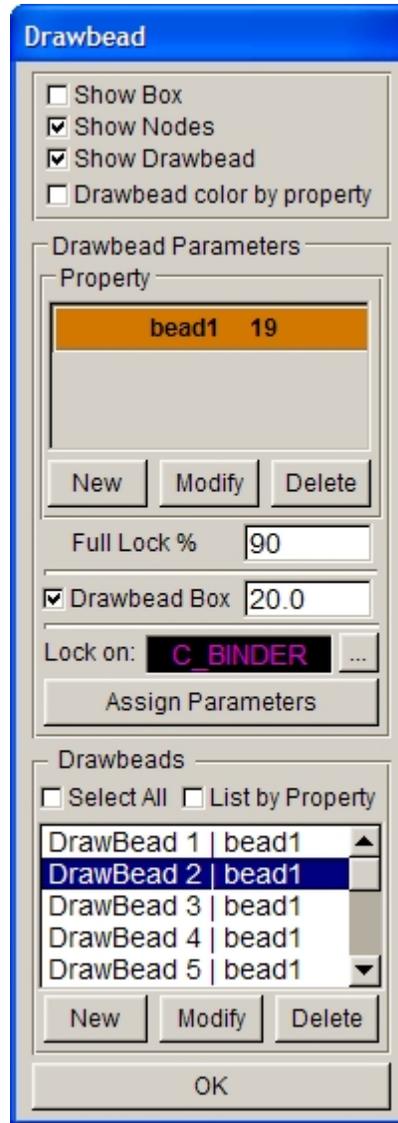


Figure 10.4.14 Lock on Part

**NOTE:** If drawbeads are generated via the menu *Setup/AutoSetup/Tools/Drawbeads*, it is 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 remaining drawbeads are not locked on rigid part, a dialog box as illustrated in Figure 10.4.15 is displayed.

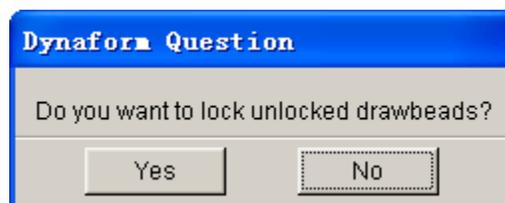


Figure 10.4.15 eta/DYNAFORM Question dialog box

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.

### 10.4.2.5 ASSIGN PARAMETERS

This function allows the user to assign parameters to a selected drawbead. First, the user can select one or multiple drawbead from the drawbead list. Then, select a property from the property list and assign full lock force, drawbead box size and lock on drawbead to the specified parts. Finally, click on ASSIGN PARAMETERS button to assign the specified parameters to the selected drawbead.

## 10.4.3 CREATE DRAWBEAD

After a drawbead property is defined, the user can continue with generation of drawbead. In the DRAWBEADS field, there are some functions such as New, Modify and Delete, as shown in Figure 10.4.16.

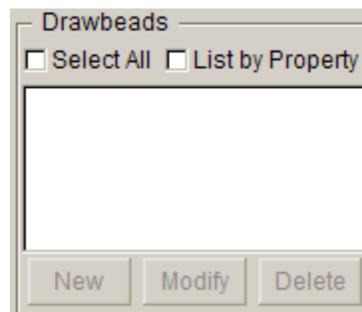


Figure 10.4.16 Drawbeads interface

### 10.4.3.1 NEW DRAWBEAD

This function enables the user to create drawbead using one of the two methods illustrated in Figure 10.4.17. First method is to select the existing geometrical lines in from the current database. Click on Select bead line option and the APPLY button to display the Select Line dialog box illustrated in Figure 10.4.18. The user can select one or multiple lines to define drawbead. The other method is to create new geometrical lines. Click on Create bead line option and the APPLY button to display the Create Line dialog box illustrated in Figure 10.4.19. The user can create lines using the following methods: Node, Point, Cursor on Element and Cursor on Surface.

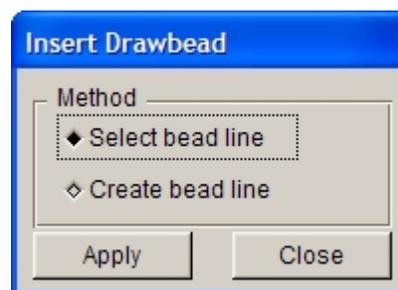


Figure 10.4.17 Insert drawbead methods

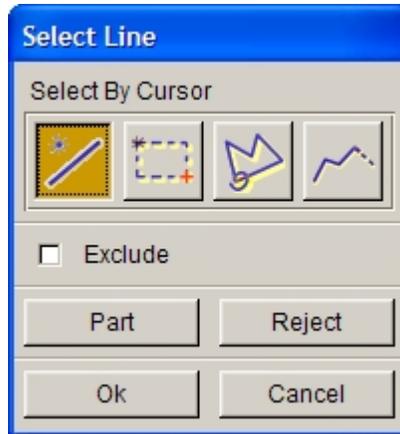


Figure 10.4.18 Select Line dialog box

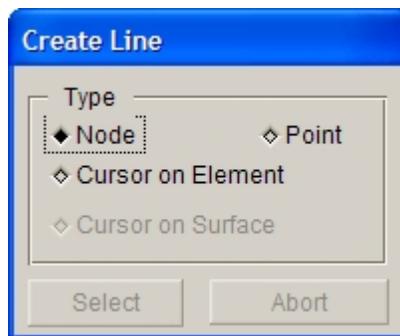


Figure 10.4.19 Create Line dialog box

**NOTE:** Prior to creating a new drawbead, the user has to select a property from the drawbead property field. The selected property is assigned as the property for new drawbead.

The DRAWBEAD INPUT dialog box illustrated in Figure 10.4.20 is displayed after a line is selected. The function provided in the dialog box enables the user to set the number of point along each drawbead line.

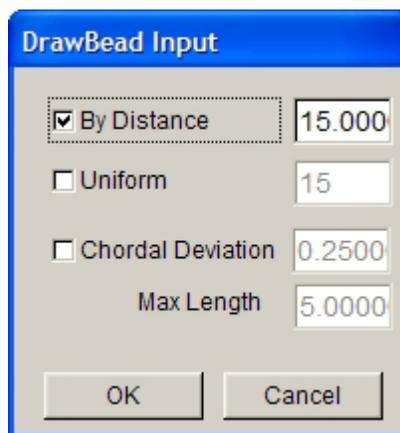


Figure 10.4.20 Define the number of points along drawbead line

**BY DISTANCE**

Define number of points along line by setting the constant distance between two adjacent points on line. The default distance is 15 (mm).

**UNIFORM**

The specified number of points is evenly spread along the line. The default number of points is set as 15 on each line. Refer to Figure 10.4.21a for drawbead line with uniform point distribution.

**CHORDAL DEVIATION**

The number of points along 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 along the drawbead line. The default CHORDAL DEVIATION is set as 0.25. Refer to Figure 10.4.21b for drawbead line using point distribution based on CHORDAL DEVIATION.

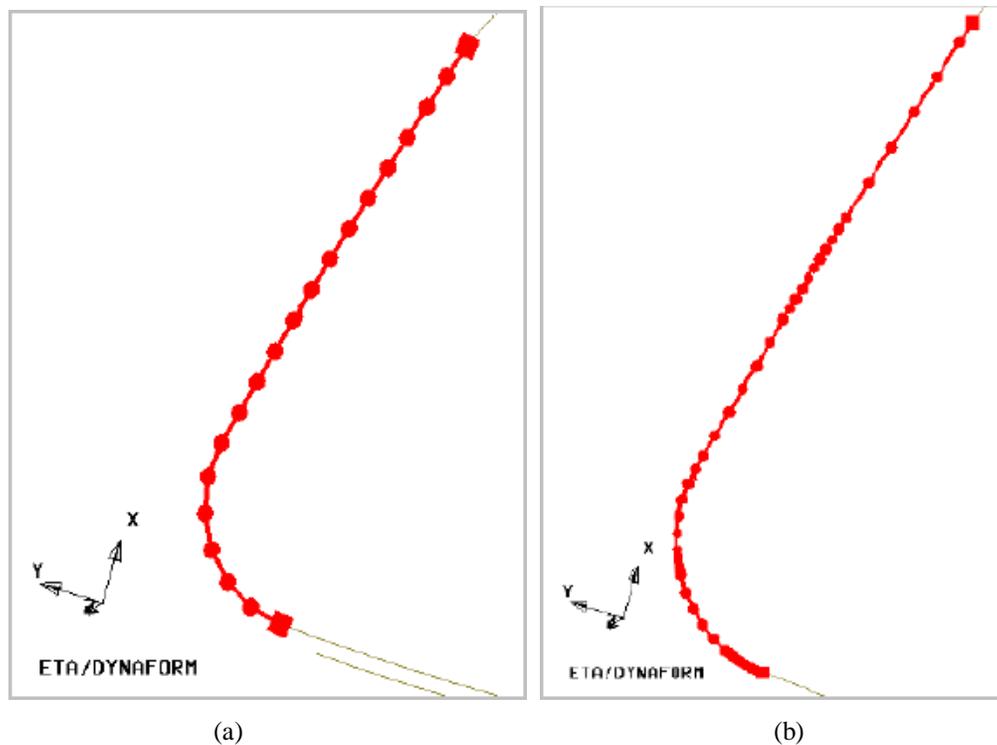
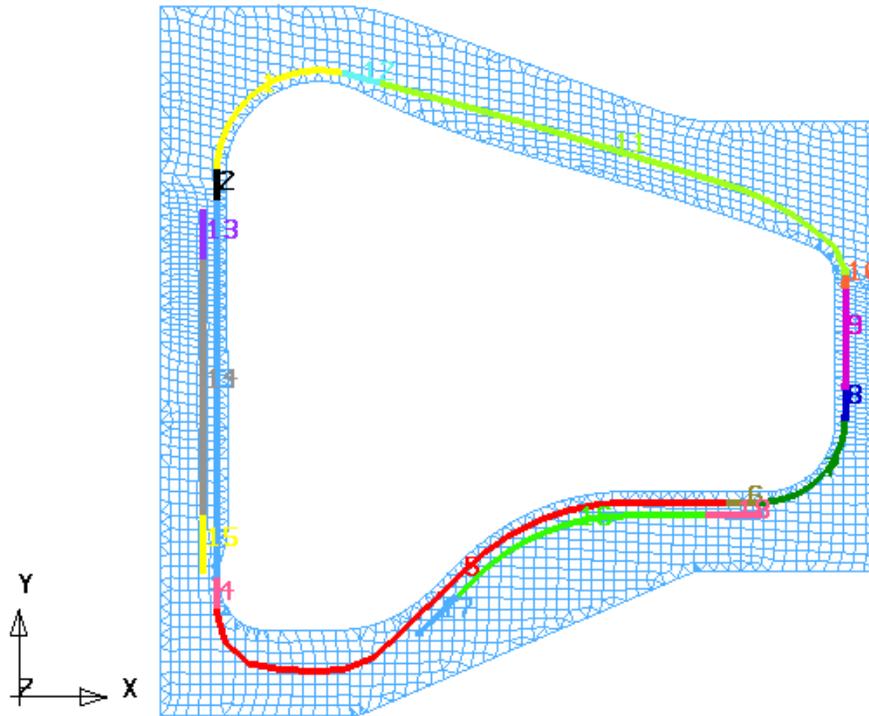


Figure 10.4.21 The distribution of points on drawbead line (a) Uniform (b) Chordal deviation

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 is identified with a unique name such as Drawbead 1, Drawbead 2 and so on. As illustrated in Figure 10.4.22, the corresponding ID number is displayed next to the drawbead on the display screen.



**ETA/DYNAFORM**

Figure 10.4.22 Drawbead name

**DRAWBEAD LIST**

All defined drawbeads are listed in the Drawbead List illustrated in Figure 10.4.23. 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 display screen (in white).



Figure 10.4.23 Drawbead List

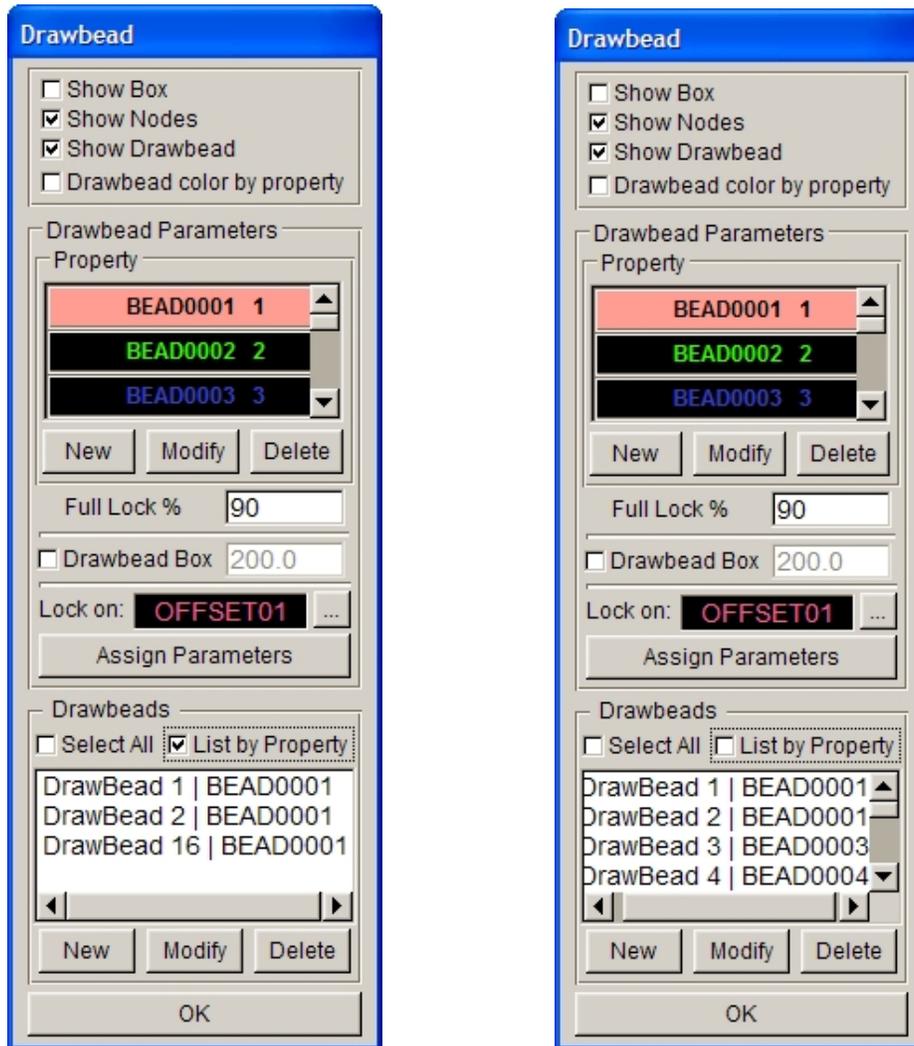
**SELECT ALL**

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, drawbeads sharing the current property are displayed on the DRAWBEAD LIST. Please see Figure 10.4.24.



(a) (b)  
Figure 10.4.24 List by Property option (a) Check (b) Uncheck

**10.4.3.2 MODIFY**

This function allows the user to modify drawbead through EDIT BLINE, RESPACE, COMBINE, SPLIT and TRIM. The user can select drawbead following by clicking on MODIFY button to display a dialog box illustrated in Figure 10.4.25.



Figure 10.4.25 Modify Drawbead

- EDIT BLINE

This function allows the user to edit drawbead line. Select this option and click on APPLY button. The program will pop up Edit Line dialog box for the user to edit line with the corresponding functions. Refer to the relevant sections for detailed description.

- RESPACE

This function allows the user to respace the points along drawbead. After this option is selected, click on APPLY button following by selecting a drawbead. The program displays the DRAWBEAD INPUT dialog box illustrated in Figure 10.4.20.

- COMBINE

This function enables the user to combine two adjacent drawbeads into one drawbead, without reselect drawbeads.

1. Click on the left mouse button to select a drawbead from the display screen.
2. Select another drawbead near the first drawbead.

An example is illustrated in Figure 10.4.26.

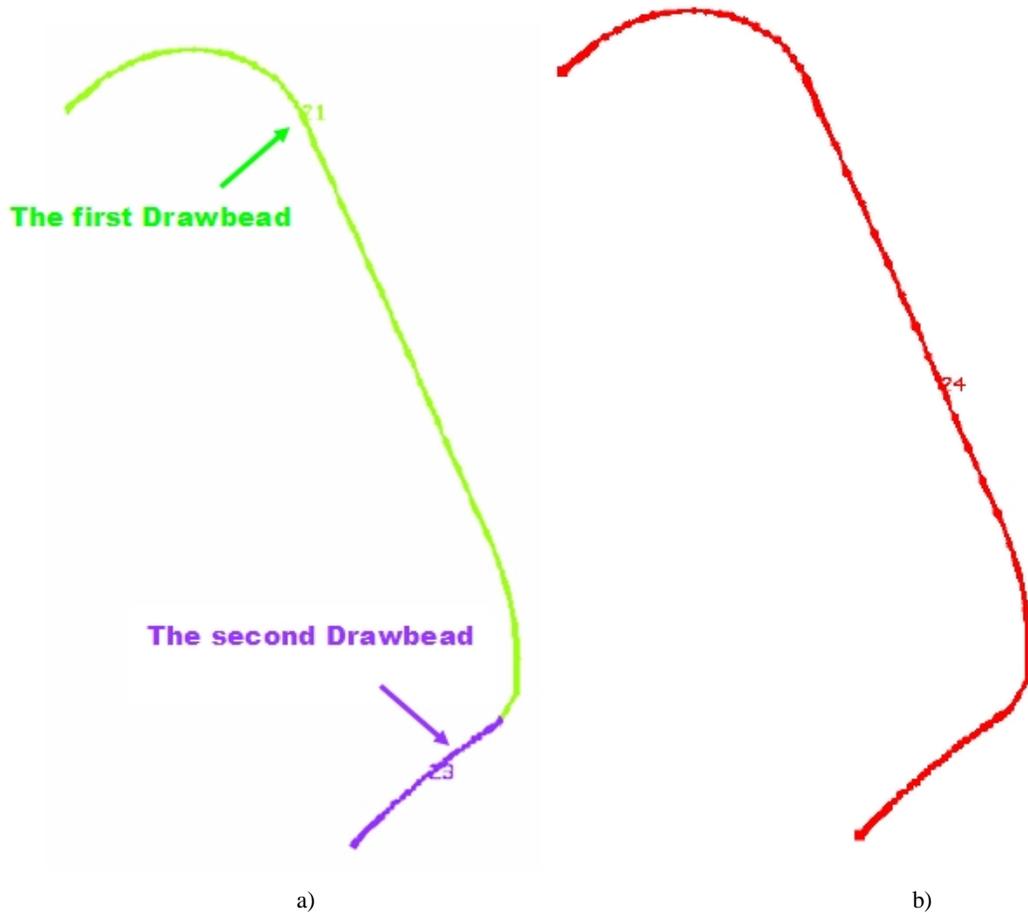


Figure 10.4.26 Combine drawbead (a) Before (b) After

**NOTE:** This function enables the user to combine two adjacent drawbeads of separate property. If of property the two adjacent drawbeads are the same, the property remains unchanged after drawbeads are combined together. If property of the two adjacent drawbeads are different, the property of **FIRST** selected drawbead is used for the combined drawbead.

- **SPLIT**

This function enables the user to divide a drawbead at selected location. Select this option and click on APPLY button. After the function is activated, the user selects a 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 property drawbeads are similar to the original drawbead. Figure 10.4.27 illustrates SPLIT drawbead operation.

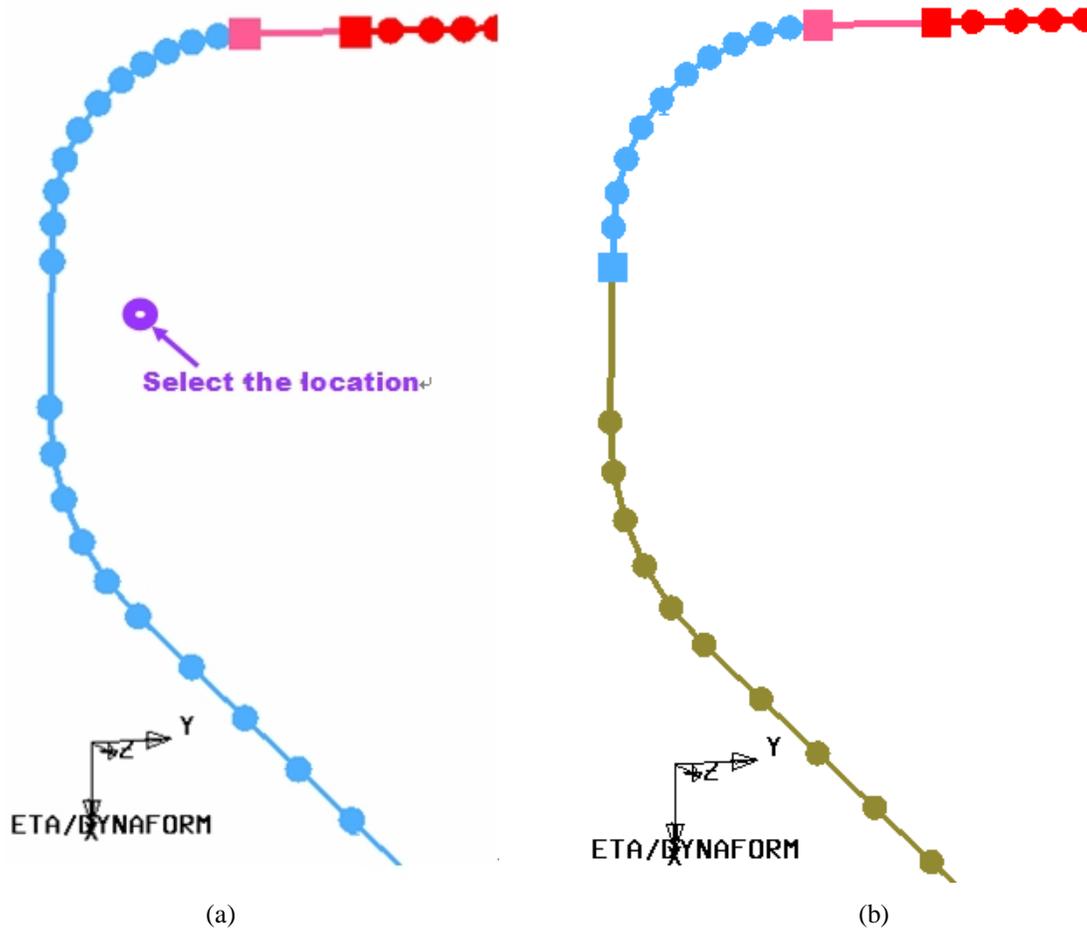


Figure 10.4.27 Split drawbead (a) Before (b) After

- TRIM

This function allows the user to trim drawbead without reselecting drawbead. Firstly, the user can select a trimming location on drawbead. Then, select another node as the trimming direction. If the node on the left side of trimming location is selected, the left side of drawbead is trimmed; if the node on the right side of trimming location is selected, the right side of drawbead is trimmed.

### 10.4.3.3 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.

## 10.5 BLANK GENERATOR

This function is mainly utilized for meshing flat blanks and tube. The dialog box is shown in Figure 10.5.1.

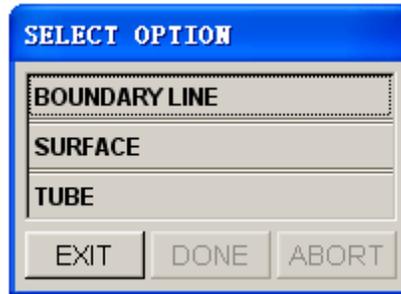


Figure 10.5.1 Blank Generator Options

### 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 10.5.2 is displayed.

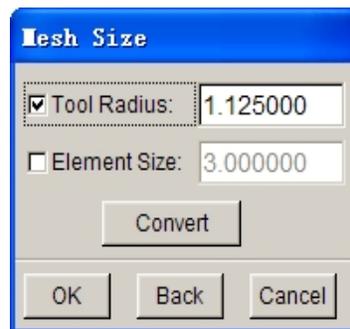


Figure 10.5.2 Mesh Size dialog box

**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 illustrated in Figure 10.5.3 is displayed after the user select OK.

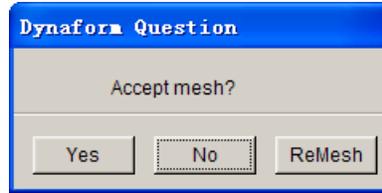


Figure 10.5.3 eta/DYNAFORM Question dialog box

**YES** – Accept blank mesh.

**NO** – Cancel the operation.

**REMESH** – The MESH SIZE dialog box is displayed. The user can modify the radius/element size.

**NOTE:**

1. *The selected boundary lines should be lying on the same plane. Otherwise, blank mesh is generated on the least square plane of the selected line.*
2. *The orientation of 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.*

**SURFACE**

This function enables the user to create a flat blank mesh by selecting an existing surface. The meshing operation is quite similar to blank mesh generation using the boundary lines.

**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 illustrated in Figure 10.5.4 is displayed.

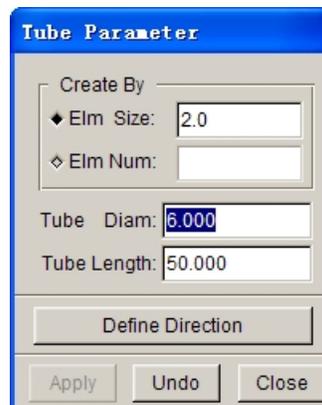


Figure 10.5.4 Tube Parameter dialog box

**ELEMENT SIZE** –Enable the user to input average element size of tube mesh. The default element size is set as 2.0.

**ELEMENT NUM** –Enable the user to input number of elements along the perimeter of tube.

**TUBE DIAM** – Enable the user to input diameter of tube.

**TUBE LENGTH** – Enable the user to input overall length of tube.

**DEFINE DIRECTION** – Enable the user to define the direction of tube mesh.

## 10.6 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 10.6.1.

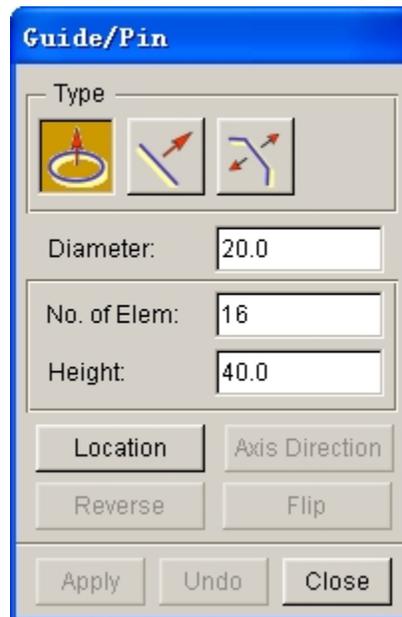


Figure 10.6.1 Guide and Pin dialog box

**DIAMETER** –Enable the user to define diameter of locator pin. The default diameter is set as 20 (mm).

**NO. OF ELEM.** –Enable the user to define number of elements along the circumference of locator pin or number of elements along the length of guide.

**HEIGHT** – Enable 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.

### 10.6.1 LOCATOR

It is a straight column pin. An illustration of locator pin is shown in Figure 10.6.2.

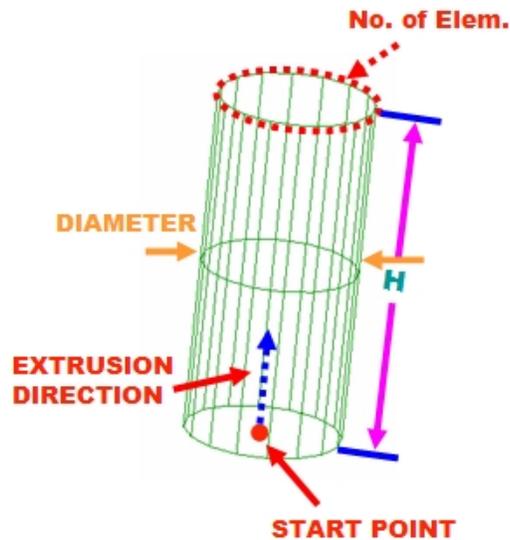


Figure 10.6.2 Locator Pin

**LOCATION** –This function enables the user to specify 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 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 user can edit direction of extrusion by hitting DIRECTION button. The VECTOR dialog box illustrated in Figure 10.6.3 is displayed to enable modification of direction of extrusion.

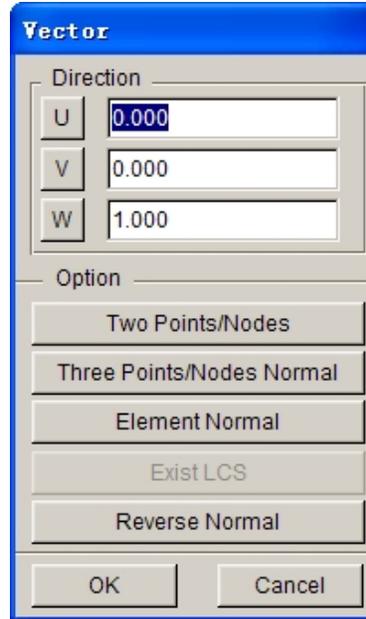


Figure 10.6.3 Vector dialog box

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 10.6.4.

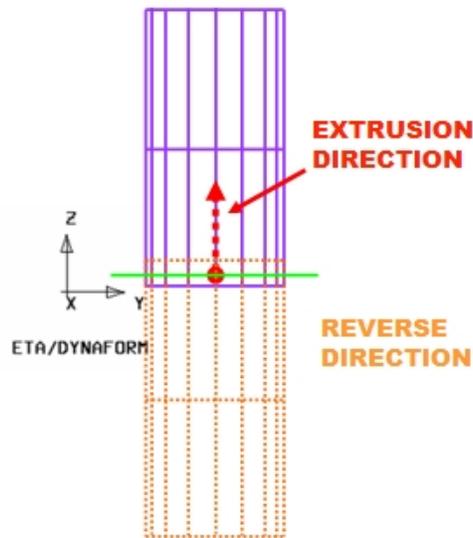


Figure 10.6.4 Reverse Locator Pin

## 10.6.2 GUIDE ONE

It is the straight guide without flange. An illustration of Guide One is shown in Figure 10.6.5.

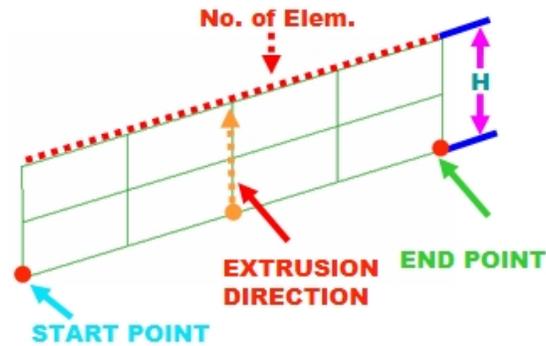


Figure 10.6.5 Guide One

The user should define location and direction of extrusion to generate straight guide without flange. The location is specified by start and end point/node, as illustrated in Figure 9.6.5.

### 10.6.3 GUIDE TWO

It is the straight guide with flange. An illustration of Guide Two is shown in Figure 10.6.6.

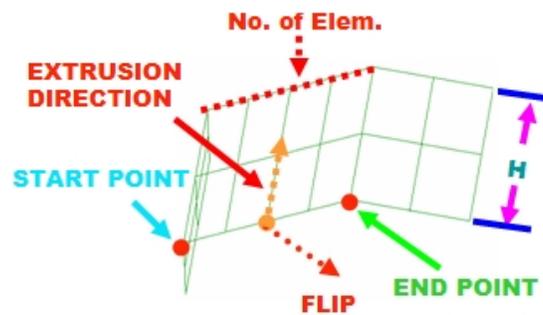


Figure 10.6.6 Guide One

Similarly to Guide One, the location and direction of extrusion should be defined. Moreover, the user can modify the direction of the flange through the FLIP and REVERSE buttons. Illustrations of Guide Two with REVERSE and FLIP result are shown in Figure 10.6.7 and Figure 10.6.8, respectively.

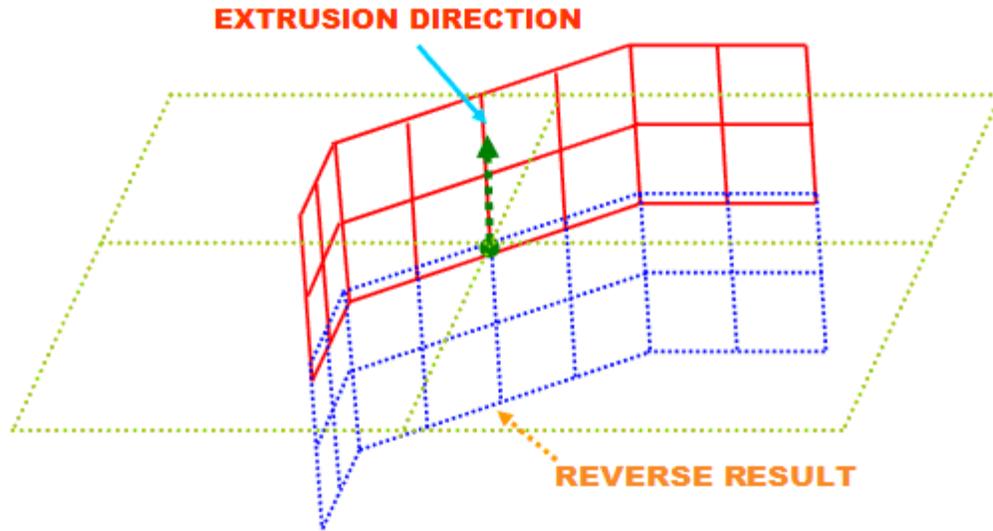


Figure 10.6.7 Reverse result

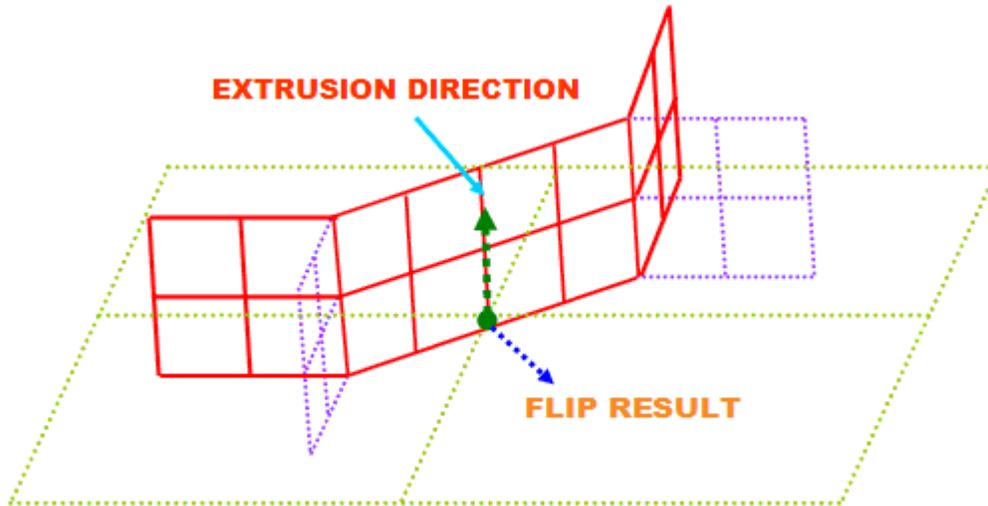


Figure 10.6.8 Flip result

**APPLY** – Create locator pin or guide.

**UNDO** – Undo last operation.

**EXIT** – Exit GUIDE & PIN dialog box.

## 10.7 DEFINE BLANK (CTRL+B)

This function allows the user to define blank material and properties for stamping simulation. The DEFINE BLANK dialog box is shown in Figure 10.7.1.

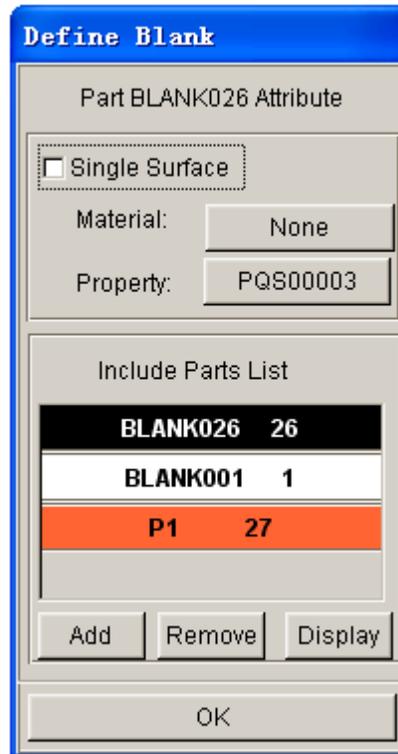


Figure 10.7.1 Define Blank dialog box

### SINGLE SURFACE

This option allows the user to activate the single surface contact interface. Activating the single surface contact interface is recommended to accurately predict the double metal folding condition during stamping simulation.

### ADD PART

This function allows the user to add parts to blank definition. The procedure is similar to adding parts to tool definition.

### DEFINE BLANK MATERIAL

This function allows the user to define the mechanical properties of blank. Refer to Section 10.9 for information about defining mechanical properties of blank material.

### DEFINE BLANK PROPERTY

This function allows the user to define shell element properties and blank thickness. Refer to Section 10.10 for information about defining shell element properties.

## 10.8 BLANK OPERATION

This menu contains eight different submenus which are used to manipulate blank geometry, view deformed blank result, map blank result, etc. The submenus are listed below:

- BLANK AUTO POSITION
- BLANK MAPPING
- RESULT MAPPING
- DYNAMIC CONTOUR
- TRIM
- TIP
- TAILOR WELDED
- LANCING

### 10.8.1 BLANK AUTO POSITION

This function is aimed to automatically reposition blank with reference to a selected tool and given contact gap. The blank can be positioned along X, Y or Z coordinates in either global or local coordinate system. The dialog box is shown in Figure 10.8.1.



Figure 10.8.1 Auto Blank Position dialog box

## 10.8.2 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 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 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 flat blank with coarse mesh is selected, the following message is printed in the message prompt window:

***SELECT COARSE MESH FOR THE DEFORMED BLANK***

The user selects deformed blank with coarse mesh that is produced by gravity loading simulation. deformed blank with the coarse mesh is selected, the following message is printed in the message prompt window:

***SELECT FINE MESH FOR THE FLAT BLANK***

The user selects flat blank with the refined mesh. Both flat blanks must share identical dimensions. eta/DYNAFORM will automatically map refined blank mesh to deformed blank shape. The deformed blank with finer mesh is then used for subsequent binder wrap and/or draw simulation.

## 10.8.3 RESULT MAPPING

The purpose of this function is two-fold:

- To map forming results to car position part mesh for structural analysis.
- To tip a blank without interfering with 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 illustrated in Figure 10.8.2. 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 checkbox COORDINATE TRANSFORM option, 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.

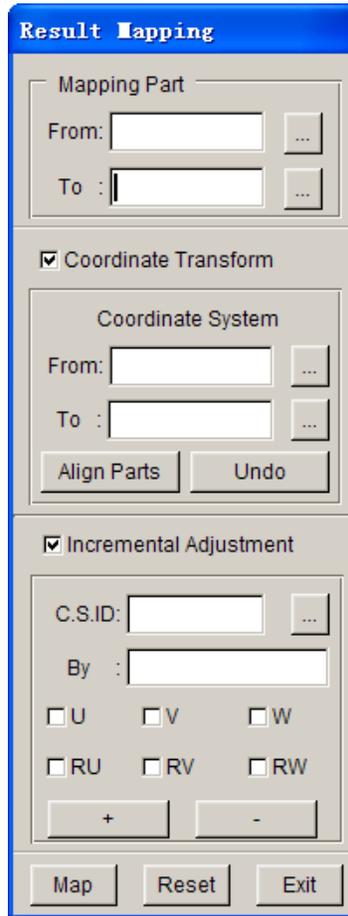


Figure 10.8.2 Result Mapping dialog box

### 10.8.4 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 database, this function is automatically disabled.

### 10.8.5 TRIM

This function is used to trim a blank using one or multiple closed/unclosed trim line(s). The interface is illustrated in Figure 10.8.3. The operation of this function is similar to TRIM ELEMENT function in Section 5.3.10.

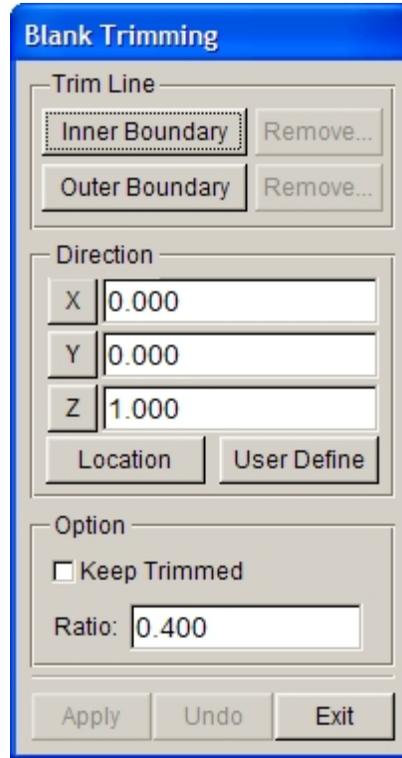
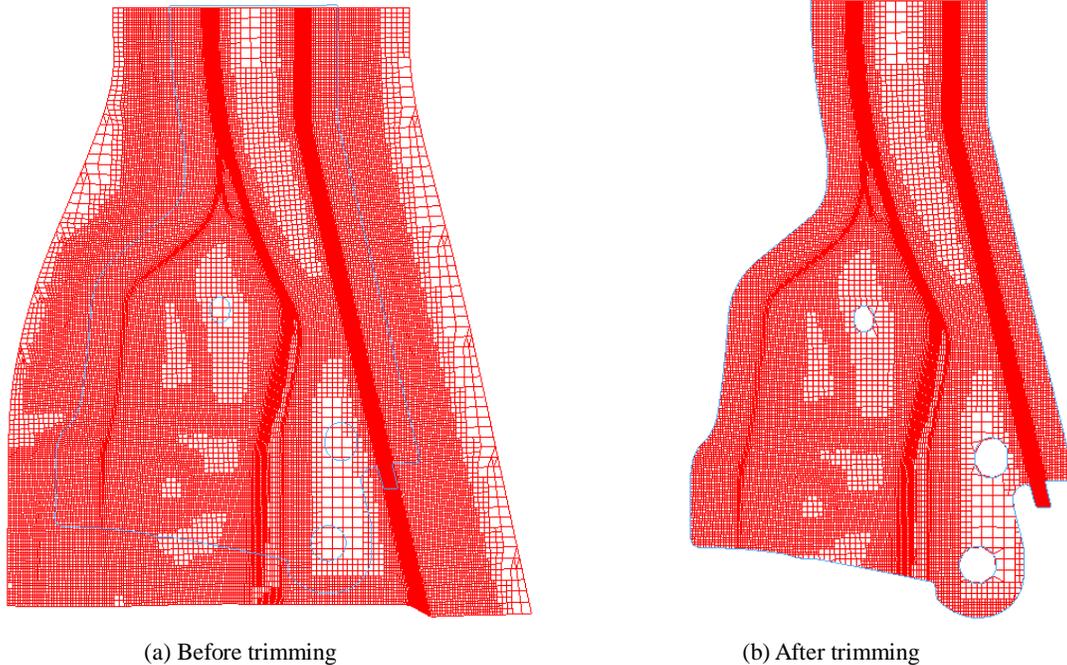


Figure 10.8.3 Blank Trimming dialog box

A typical example for Blank Trimming is illustrated in Figure 10.8.4.



(a) Before trimming

(b) After trimming

Figure 10.8.4 Trimming with Multiple Closed Trim Line

**NOTE:** *Prior to performing the trimming operation, the user must define the blank properties.*

### 10.8.6 TIP

This function allows the user to tip the blank according to a user-defined coordinate system. The dialog box is shown in Figure 10.8.5.

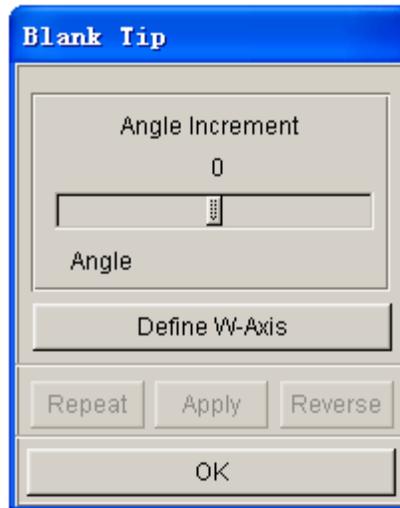


Figure 10.8.5 Blank Tipping dialog box

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.

### 10.8.7 TAILOR WELDED

This function allows the user to weld together parts of different material and/or thickness. If the TAILOR WELDED function is selected, the SELECT PART dialog box illustrated in Figure 4.9.1 is displayed. The following message is printed 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.

### 10.8.8 LANCING

This function allows user to carry out lancing operations by splitting the selected elements along one or multiple given paths. The interface is illustrated in Figure 10.8.6. The operation of this function is similar to the one of LANCE ELEMENT functions described in Section 5.3.11..

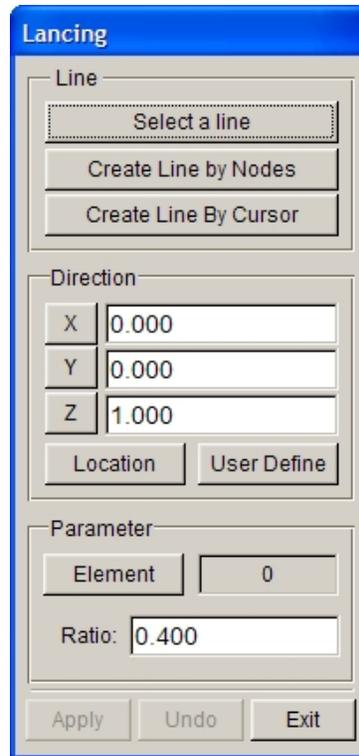


Figure 10.8.6 Select Method dialog box

**NOTE:** *Prior to performing the lancing operation, the user must define the blank and its properties.*

## 10.9 MATERIAL

The MATERIAL dialog box, as illustrated in Figure 10.9.1, allows the user to create, modify, and delete material definition. In addition, the user can import and export any defined materials.

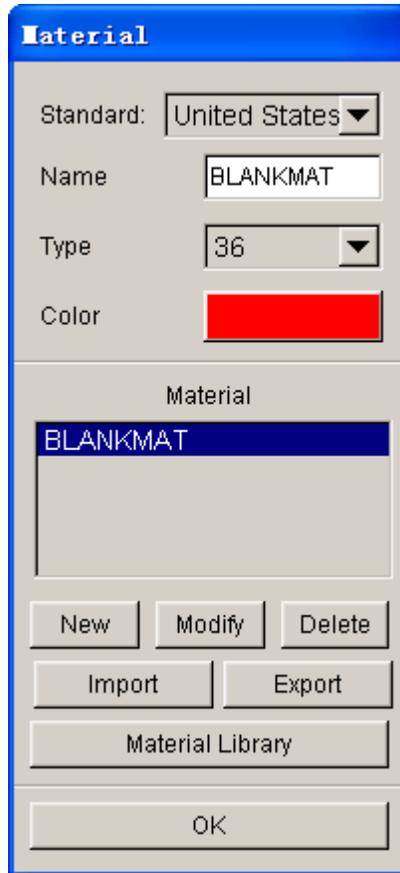
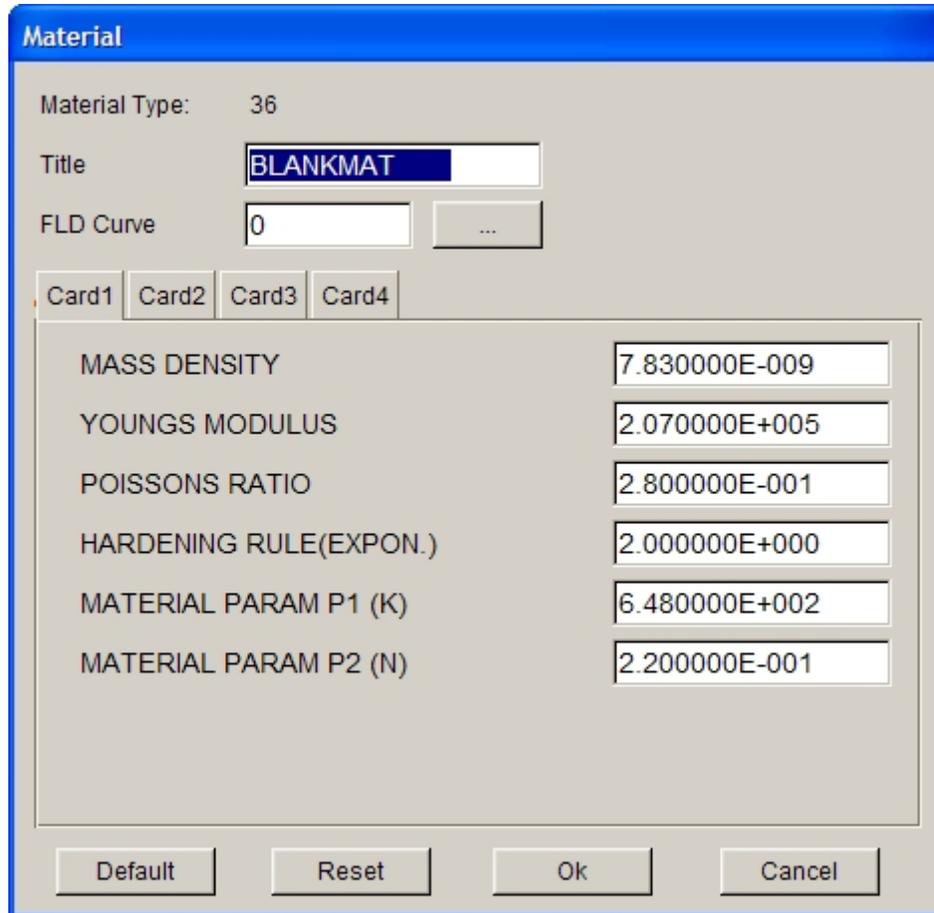


Figure 10.9.1 Material dialog box

The user can select type of material model by clicking the drop down button shown in Figure 10.9.1. The default material model is set to type #36. Refer to the LS-DYNA User's Manual for detailed information about material models supported by eta/DYNAFORM.

### 10.9.1 NEW (MATERIAL)

This function enables the user to create a new material model, as well as defining the associated parameters in the table illustrated in Figure 10.9.2.



The dialog box titled "Material" contains the following fields and controls:

- Material Type: 36
- Title: BLANKMAT
- FLD Curve: 0
- Card selection: Card1, Card2, Card3, Card4
- Parameter table:
 

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
- Buttons: Default, Reset, Ok, Cancel

Figure 10.9.2 Material Model Parameter Table

**NOTE:** The FLD CURVE NUMBER is listed at the end of every material table. This item is not needed for stamping simulation, except MAT-39 is chosen in stamping simulation. 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.

## 10.9.2 MODIFY (MATERIAL)

This function allows the user to edit the parameters of a selected material model using the table shown in Figure 10.9.2.

## 10.9.3 DELETE (MATERIAL)

This function is utilized to remove a defined material model from the database.

## 10.9.4 IMPORT

This function allows the user to read in a material from a file with “.mat” extension. Choose a material file and select OK to close the IMPORT MATERIAL window. The imported material is displayed in the MATERIAL window.

## 10.9.5 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 10.9.3.

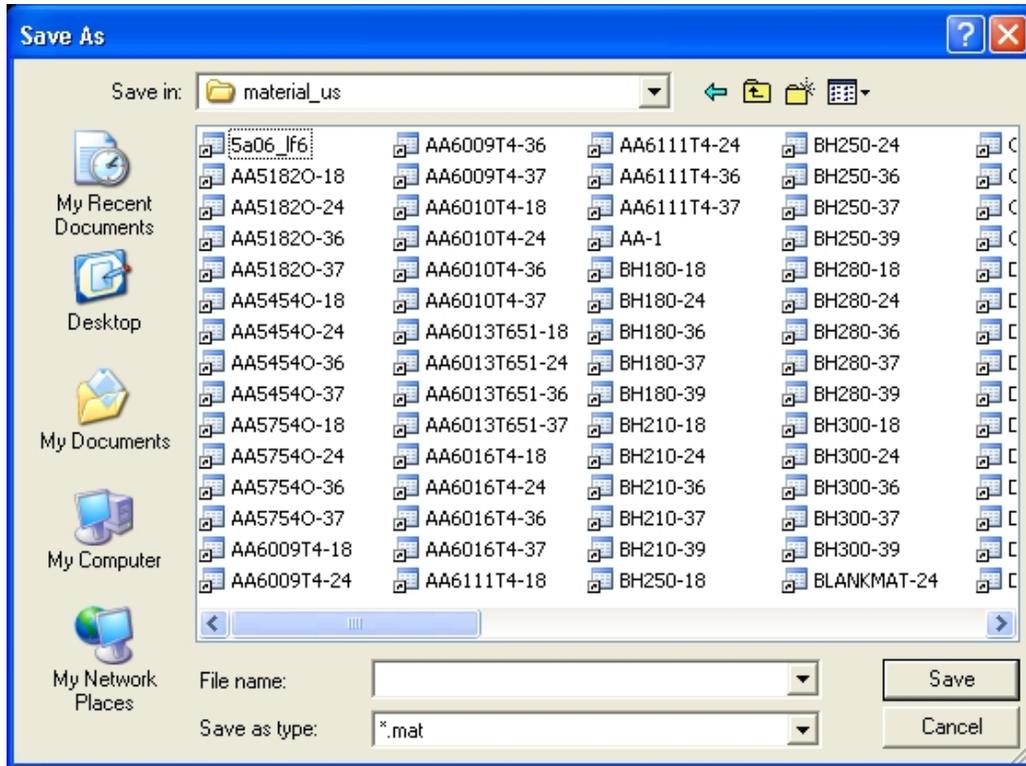


Figure 10.9.3 Export Material Window

## 10.9.6 MATERIAL LIBRARY

This function allows user to select a material directly from the generic material library embedded in eta/DYNAFORM. In the eta/DYNAFORM 5.6, 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” material files and DF\_MATERIAL\_INDEX.dat file. Each “.mat” material file consists of the LS-DYNA material model. The DF\_MATERIAL\_INDEX.dat is the index of all material files.

As shown in Figure 10.9.4, the user can select any material from the menu and the relative parameters are displayed in the MATERIAL PARAMETERS table.

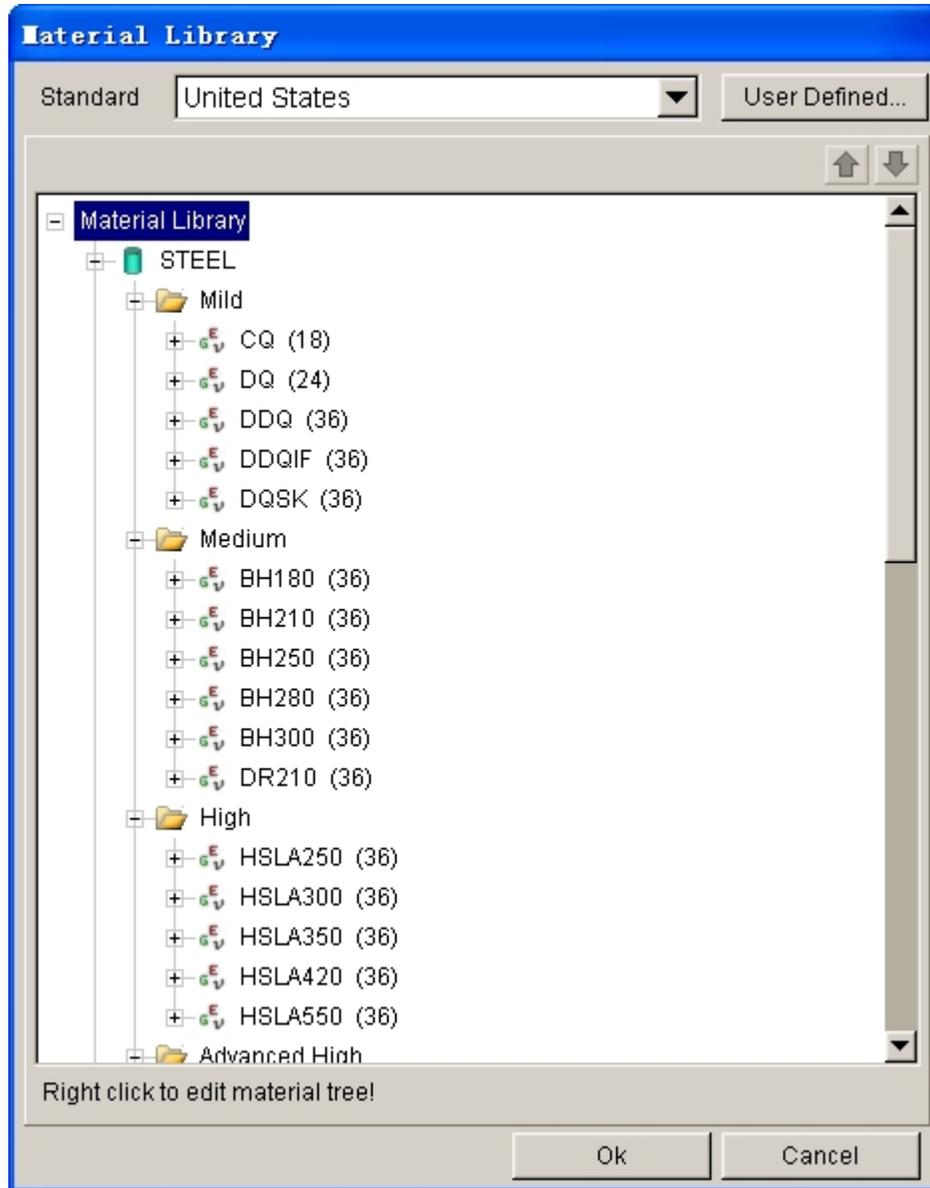


Figure 10.9.4 Material Library window

**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!*

## 10.10 PROPERTY

The functions in this menu are used to define and modify physical properties of blank. As shown in Figure 10.10.1, 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.

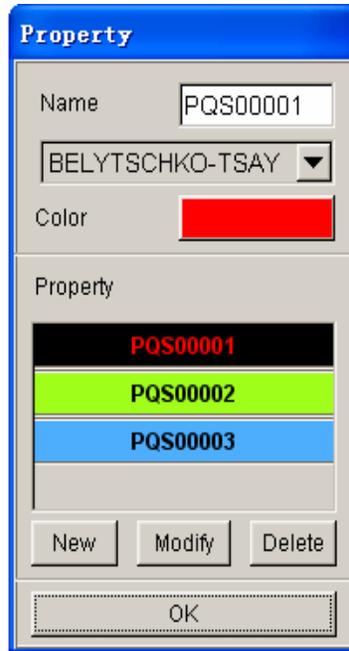


Figure 10.10.1 Property dialog box

### 10.10.1 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 10.9.

## 10.11 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 10.11.1.

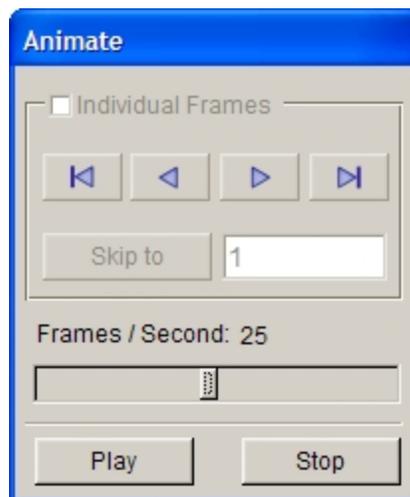


Figure 10.11.1 Define Animation dialog box

**SLIDER**

Allow the user to set number of frames for the animation.

**PLAY**

Click on PLAY button to animate all frames.

**INDIVIDUAL FRAMES**

Allow the user to animate tool motion using individual frames. Once this option is selected, the user can click on Forward and Backward buttons to view the animation. The SKIP TO button illustrated in Figure 10.11.2 is enabled.

**SKIP TO**

Allow the user to directly animate the tool motion of a frame. Type in a frame number in the input box and click on SKIP TO button to animate the tool motion of this frame.

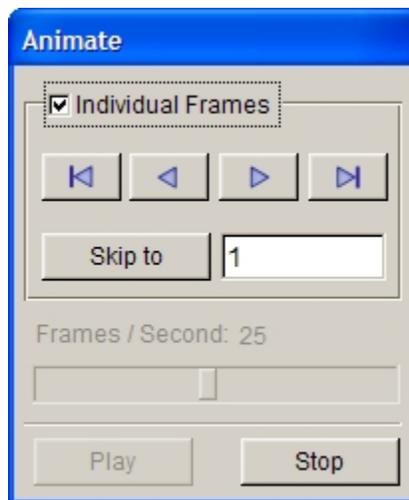


Figure 10.11.2 Individual Frames

## 10.12 TOOLS ON/OFF

This function allows the user to show/hide individual or all tools. The TOOLS ON/OFF dialog box is shown in Figure 10.12.1.

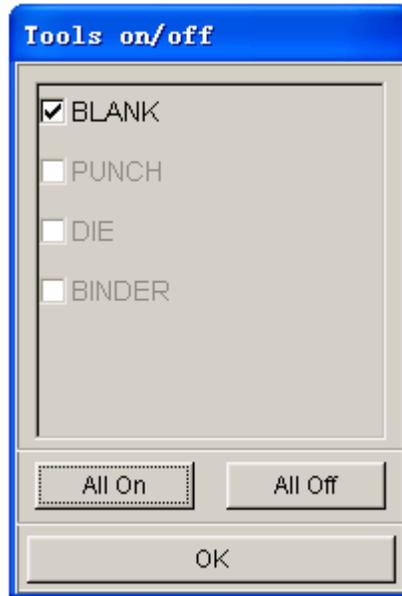


Figure 10.12.1 Tools On/Off dialog box

## 10.13 SUMMARY

This function enables the user to review 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 10.13.1.

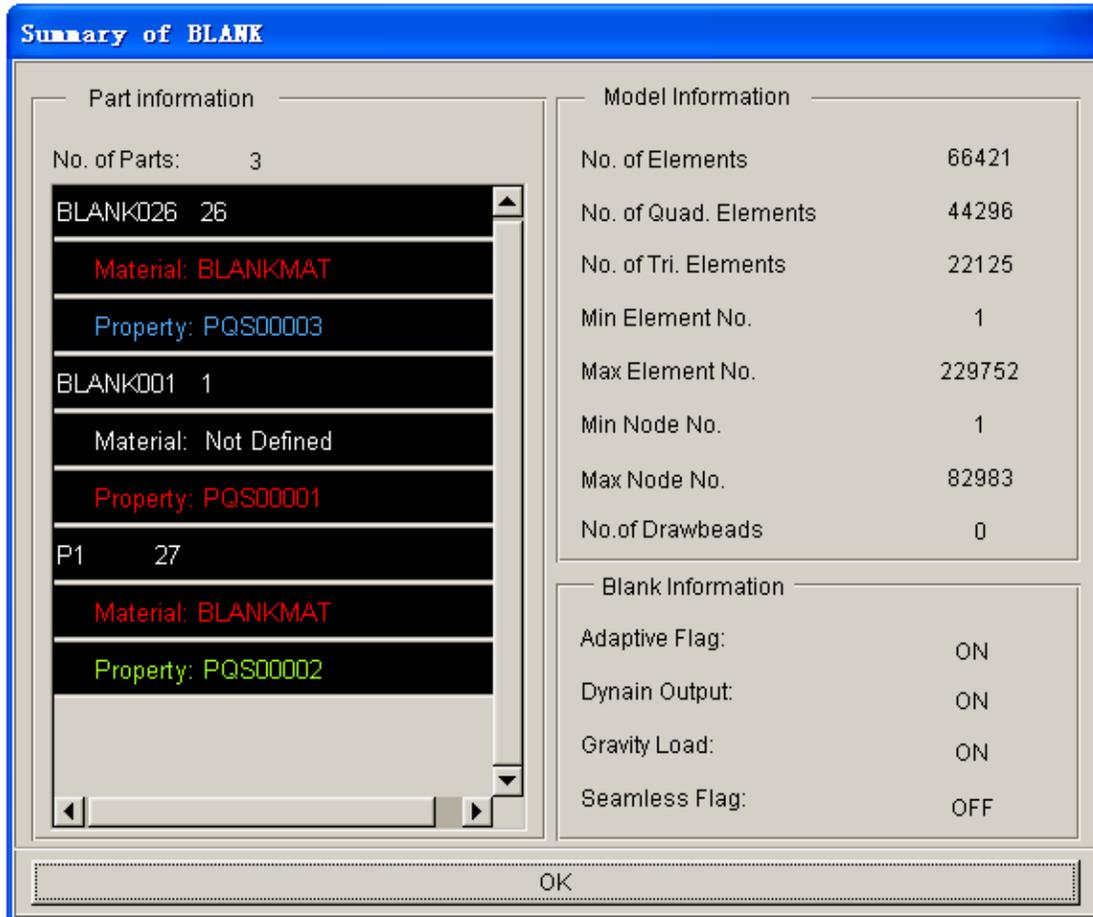


Figure 10.13.1 Tool Statistic Message window

## Chapter 11

# OPTION MENU

This menu consists of various preprocessing utilities that are toggle-switch activated as shown in Figure 11.1.



Figure 11.1 Option Menu

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

## 11.1 MESH CONTROL

The options provided in this dialog box, as illustrated in Figure 11.1.1, are used to control mesh generation.

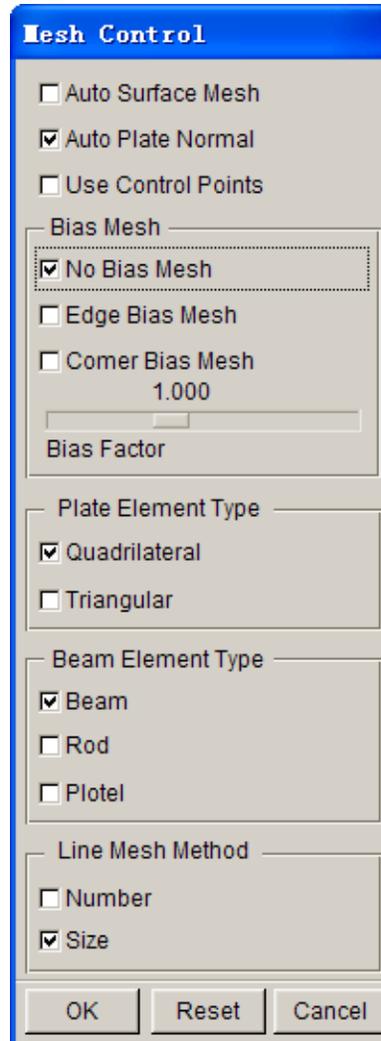


Figure 11.1.1 Mesh Control dialog box

### **AUTO SURFACE MESH (toggle)**

Enable the user to directly mesh the displayed surfaces using the SURFACE MESH function.

### **AUTO PLATE NORMAL (toggle)**

Enable the user to mesh the part with consistent element normal.

### **11.1.1 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.

### 11.1.2 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 option illustrated in Figure 11.1.1 is toggled on, the user can define the bias factor by dragging the slider. An example of mesh with edge bias mesh control is illustrated in Figure 11.1.2b.

**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.*

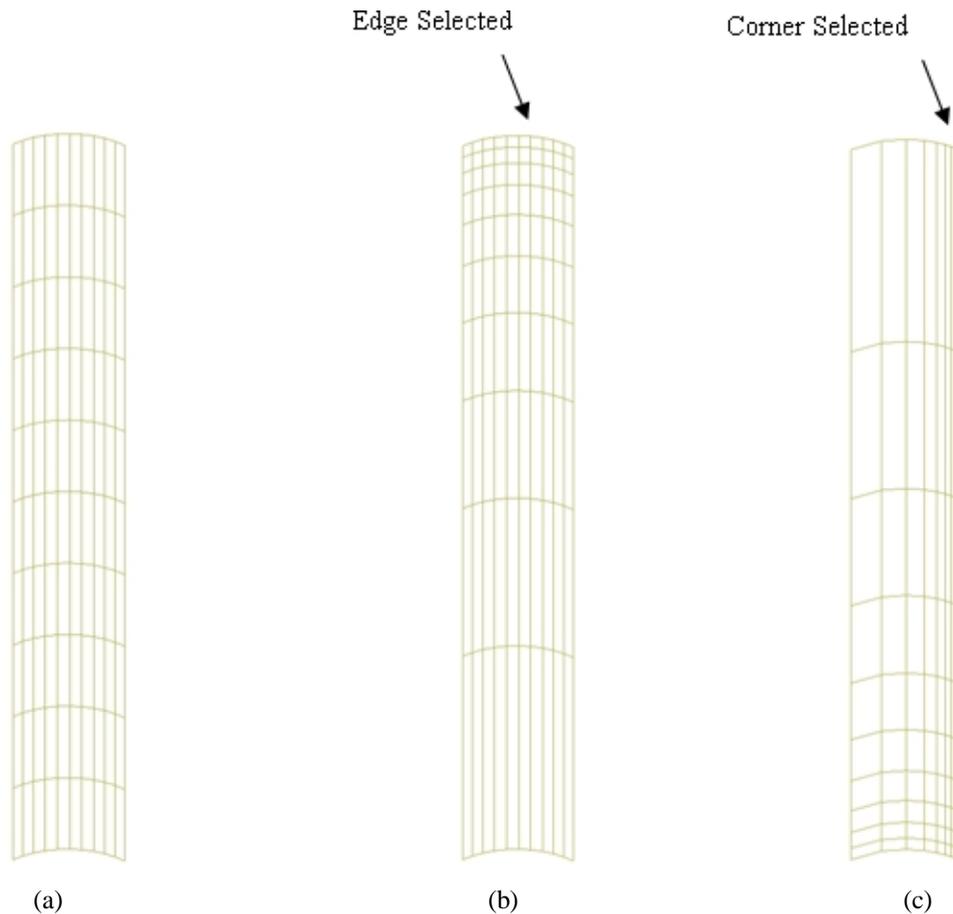


Figure 11.1.2 An example of mesh with bias mesh control: (a) No Bias (b) Edge Bias (c) Corner Bias (Bias Factor=1.765).

### 11.1.3 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 option illustrated in Figure 11.1.1 is toggled on, the user can define the bias factor by dragging the slider. An example of mesh with corner bias mesh control is illustrated in Figure 11.1.2c.

**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.

### 11.1.4 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.

### 11.1.5 LINE MESH METHOD

Please refer to Section 5.3.5 for detailed information about LINE MESH METHOD.

## 11.2 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 illustrated in Figure 11.2.1.

**NOTE:** The FILE DIALOG STYLE option is inactive in WINDOW.

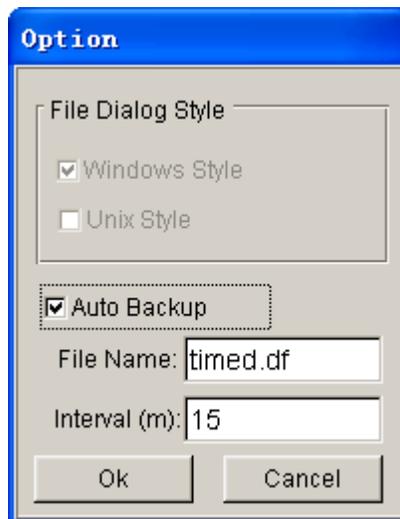


Figure 11.2.1 Option dialog box

If WINDOWS STYLE is checked, the file dialog style is shown as in Figure 11.2.2

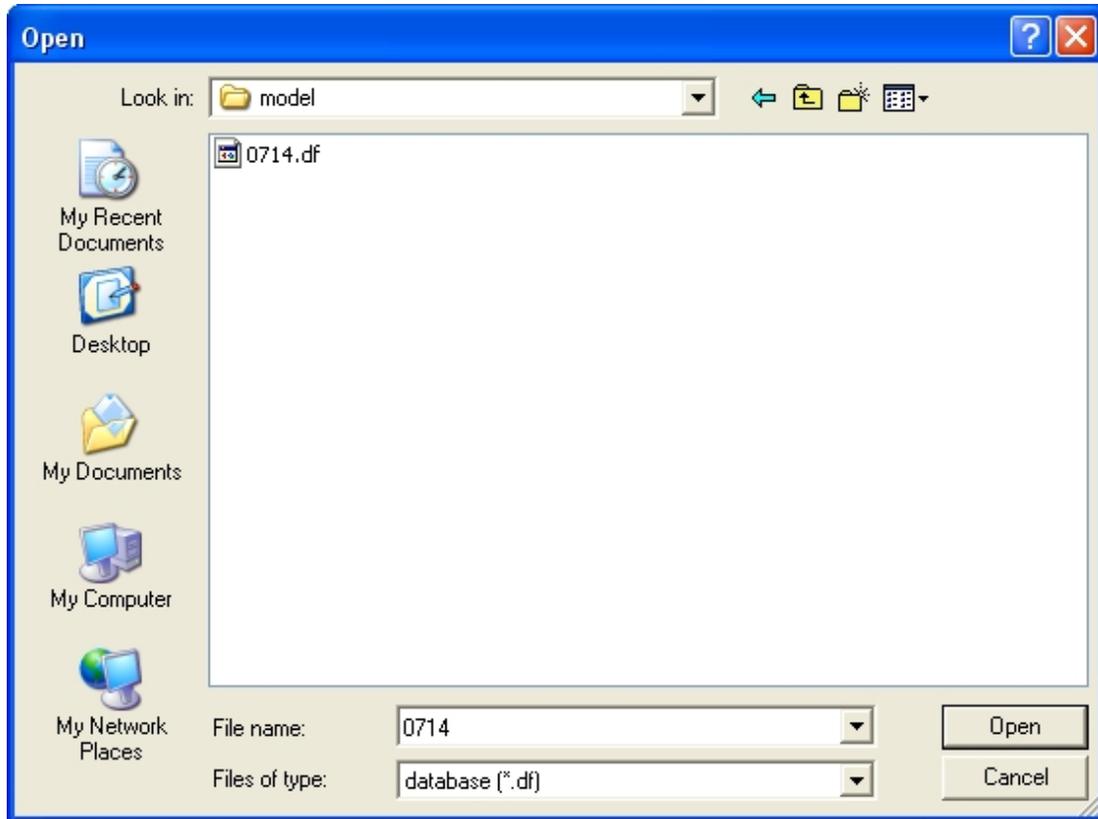


Figure 11.2.2 Windows Dialog Style

If UNIX STYLE is checked, the file dialog style is shown as in Figure 11.2.3.

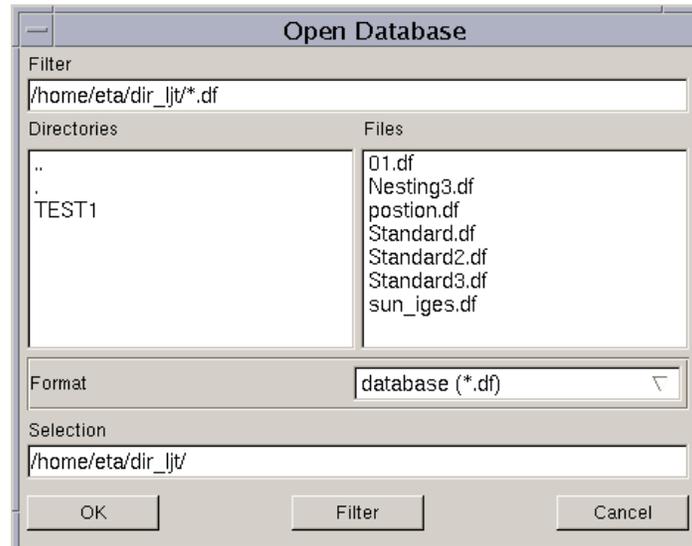


Figure 11.2.3 UNIX Dialog Style

## 11.3 SYSTEM LANGUAGE

This function allows the user to change the language in the menu, prompt and icon tips displayed on the screen. The dialog box is illustrated in Figure 11.3.1.



Figure 11.3.1 Select Language dialog box

Click on the drop down button to select a 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 illustrated in Figure 11.3.2 is displayed.

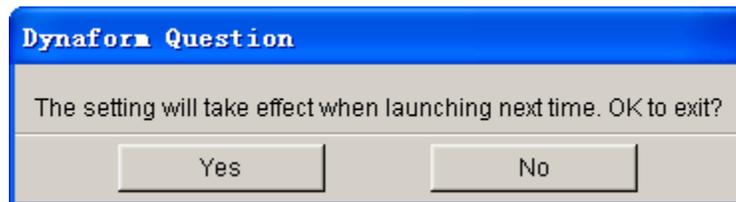


Figure 11.3.2 Select Language Confirmation Window

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.

## 11.4 MATERIAL LIBRARY STANDARD

There are four different types of material library supported in eta/DYNAFORM. The user can set the default material library from the SET STANDARD dialog box as illustrated in Figure 11.4.1. 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. Private is the save path for the user-defined material library.

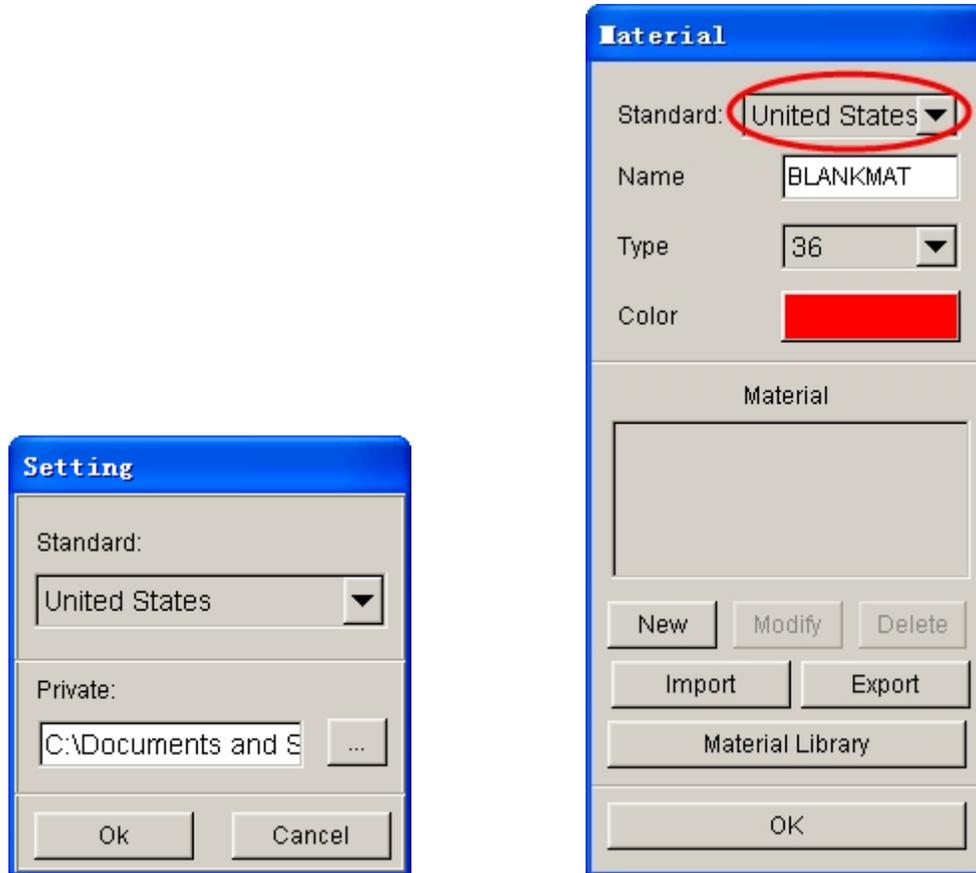


Figure 11.4.1 Set the default material standard

## 11.5 LEFT MENU

The pop-up menu is default located on the left of the mainly GUI of the eta/DYNAFORM. The user can change the style and make the pop-up menu located on the right of the mainly GUI as the Figure 11.5.1 shown.

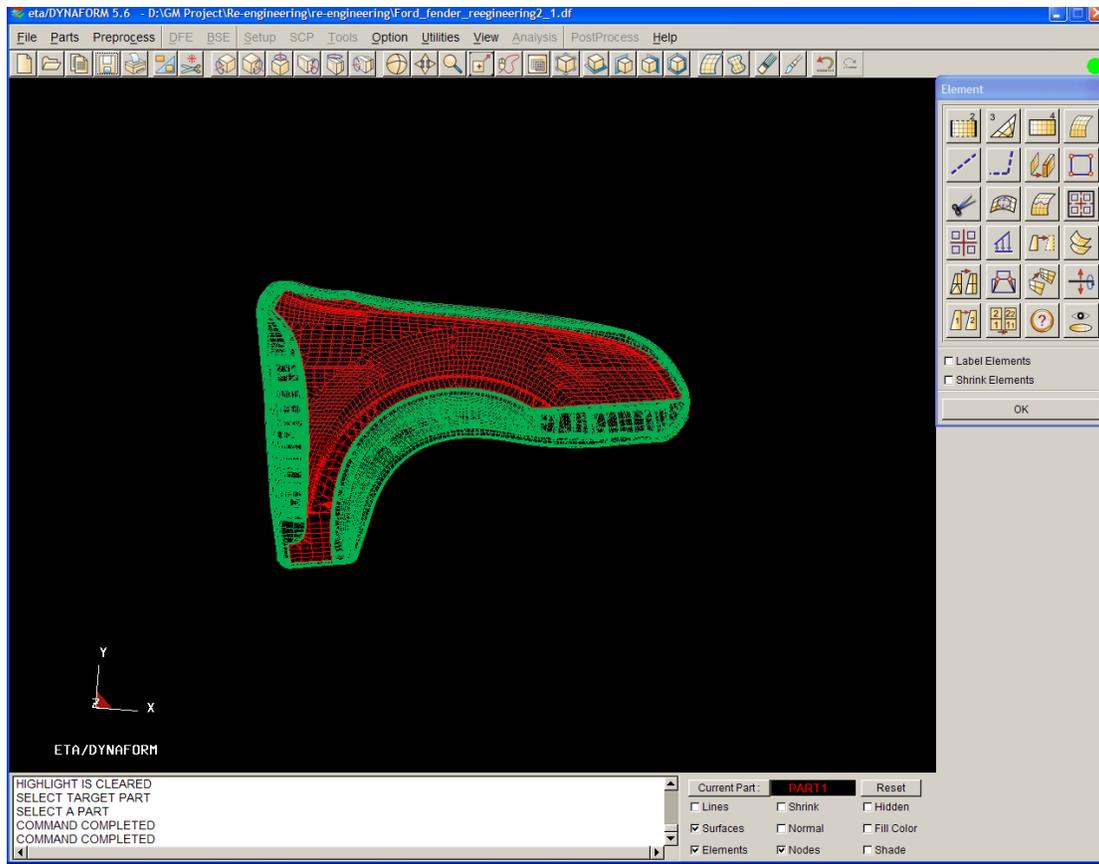


Figure 11.5.1 Left menu option

## 11.6 SHOW ICON TIPS (toggle)

This function allows the user to read/identify the name of each icon on the menu bar and dialog box. Hover cursor over an icon to display the tips on the bottom-right of each icon, such as illustrated in Figure 11.6.1.



Figure 11.6.1 Show Tips

## Chapter 12

# 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.

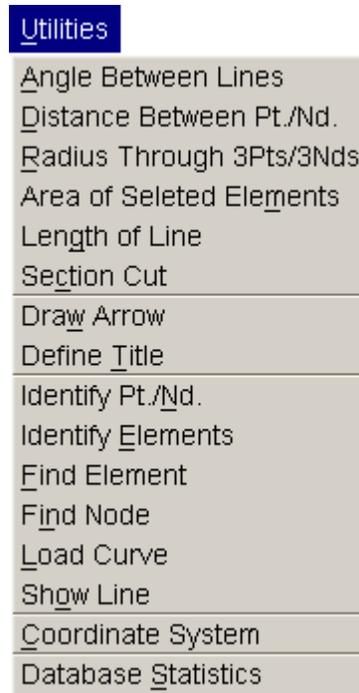


Figure 12.1 Utilities Menu

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

## 12.1 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***

## 12.2 DISTANCE BETWEEN POINTS/NODES

This function enables the user to measure the distance (in mm) between two points/nodes, or a point/node and line or surface. The program display a dialog box as shown in Figure 12.2.1 for the user to select the measuring type.

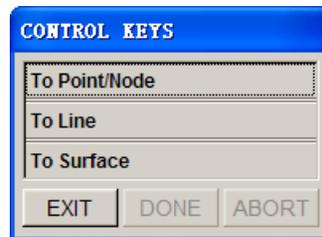


Figure 12.2.1 Measuring Type

### TO POINT/NODE

Measure the distance between two points/nodes. Once this function is selected, the program pops up a dialog box for the user to select two points/nodes on the screen by cursor and measure the distance between them. The user can also define two points/nodes by entering their coordinates. The program will display the measurement result on the screen and simultaneously display the DISTANCE, DX, DY and DZ values in the message prompt window.

***DISTANCE=185.465, DX=157.975, DY=97.165, DZ=0.000***

### TO LINE

Measure the shortest distance from point/node to line. Once this function is selected, the program prompts the user to select a point/node. The user can select point/node by mouse cursor or by entering the coordinates, and then select a line on the screen. The program will prompt the user for the shortest distance from this point/node to the line:

***DISTANCE=80.728, DX=69.490, DY=40.977, DZ=-3.000***

Simultaneously, the points which are close to the selected point/node on line are highlighted on the screen. The program pops up an eta/DYNAFORM Question dialog box as shown in Figure 12.2.2 to determine whether to create a node at the shortest distance. The user can select YES to create a node or select NO to continue the measurement.

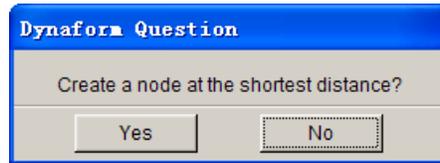


Figure 12.2.2 eta/DYNAFORM Question dialog box

### TO SURFACE

Measure the shortest distance from point/node to surface. Once this function is selected, the program prompts the user to select a point/node. The user can select point/node by mouse cursor or by entering the coordinates. Then select a surface on the screen. The program will prompt the user for the shortest distance from this point/node to the surface:

**DISTANCE=28.193, DX=24.824, DY=13.327, DZ=-1.000**

Simultaneously, the points which are close to the selected point/node on surface are highlighted on the screen. The program pops up an eta/DYNAFORM Question dialog box as shown in Figure 12.2.2 to determine whether to create a node at the shortest distance. The user can select YES to create a node or select NO to continue the measurement.

## 12.3 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***

## 12.4 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***

## 12.5 LENGTH OF LINE

This function allows user to measure length of the selected line. After selecting this function, the SELECT LINE dialog box as illustrated in Figure 12.5.1 is displayed.

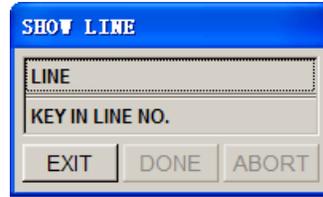


Figure 12.5.1 Select Line dialog box

If LINE option is selected, eta/DYNAFORM prompts the user to select a line on the display screen. If KEY IN LINE NO. option is selected, the LINE NUMBER dialog box illustrated in Figure 12.5.2 is displayed. The user can select the line by typing line ID in input data field.

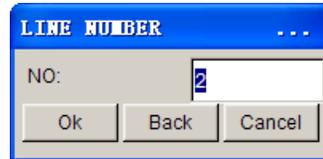


Figure 12.5.2 Line Number dialog box

The length of the selected line is printed in the message prompt window in the following format:

***LINE SEGMENT LENGTH IS: 232.3619***

## 12.6 SECTION CUT

Refer to Section 5.1.7 Section Line for detailed description of this function.

## 12.7 DRAW ARROW

This function allows the user to draw arrows near a specific location on the display area, as shown in Figure 12.7.1.



Figure 12.7.1 Draw Arrow

The user selects two locations on the display screen. 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.***

## 12.8 DEFINE TITLE

This function allows the user to enter a title or text at any location of the displayed screen. The dialog box is shown in Figure 12.8.1.

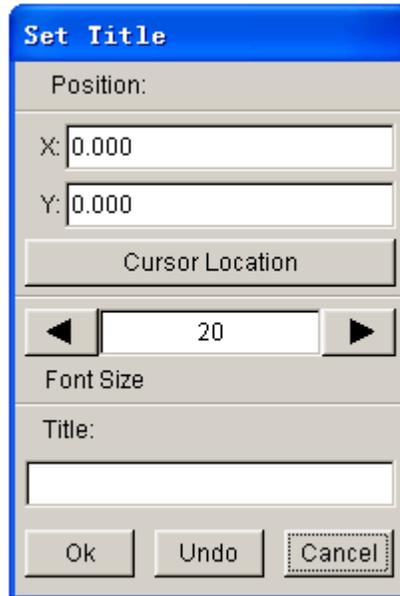


Figure 12.8.1 Define Title dialog box

- Type in caption in the Title data input field.
- Click arrow buttons to set the font size of the title.
- Click OK to display the title on the display screen.

The user can define the position of the title by entering the screen coordinates of the X, Y values or by selecting CURSOR LOCATION.

## 12.9 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.

## 12.10 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.

## 12.11 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.

## 12.12 FIND NODE

This function allows the user to find the X, Y, Z coordinates of the specified node by typing the node identification number. This function is also found in the PREPROCESS/NODE dialog box.

## 12.13 LOAD CURVE

The options in this menu are utilized to generate or modify the load curves. The options are shown in Figure 12.13.1.

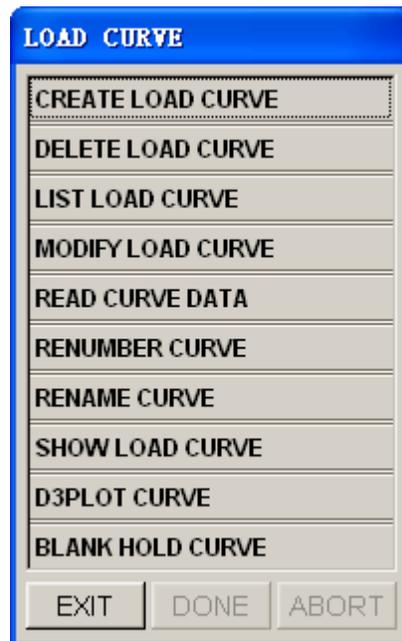


Figure 12.13.1 Load Curve Options

### 12.13.1 CREATE CURVE

This function allows the user to manually create a load curve. As shown in Figure 12.13.2, 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.

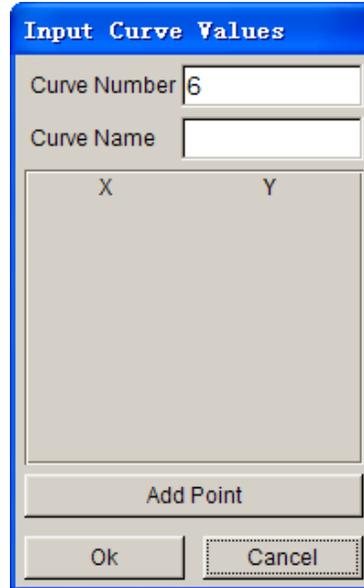


Figure 12.13.2 Create Curve dialog box

### 12.13.2 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 illustrated in Figure 12.13.3. 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.

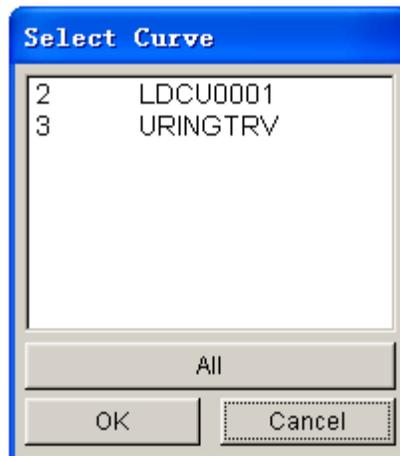


Figure 12.13.3 Select Curve dialog box

- An eta/DYNAFORM QUESTION dialog box, as shown in Figure 12.13.4, is displayed.

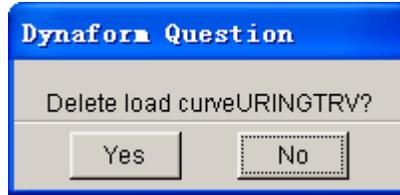


Figure 12.13.4 eta/DYNAFORM Question dialog box

- Select YES to delete the selected curve(s).
- Select NO to reject the choice.

### 12.13.3 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 12.13.5.

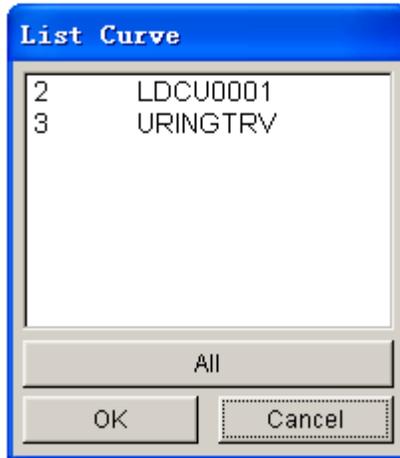


Figure 12.13.5 List Curve dialog box

### 12.13.4 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 illustrated in Figure 12.13.6, is displayed.

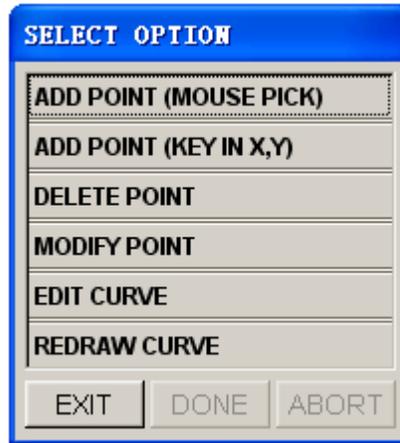


Figure 12.13.6 The Modify Curve dialog box

#### **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**

Remove selected data points.

#### **MODIFY POINT**

There are several options to modify data points using the SELECT OPTION dialog box illustrated in Figure 12.13.7.

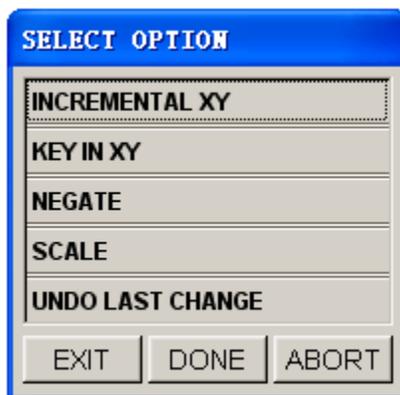


Figure 12.13.7 Modify Point Options

- **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 are 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 illustrated in Figure 12.13.8.

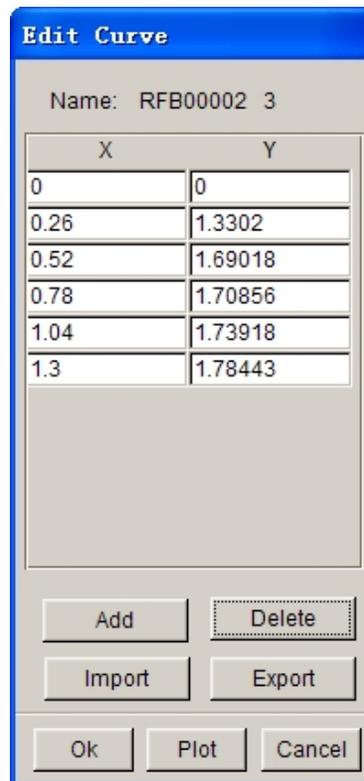


Figure 12.13.8 Edit Curve dialog box

### REDRAW CURVE

This function redraws the modified curve in the display screen.

### 12.13.5 READ CURVE DATA

This function enables the user to import an existing curve's data into the eta/DYNAFORM database.

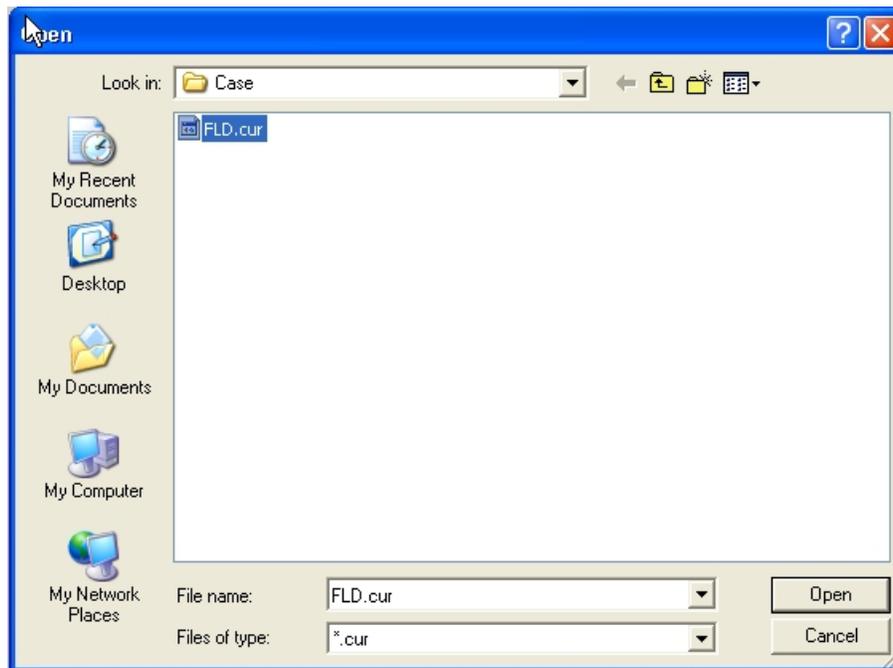


Figure 12.13.9 Open Curve File Window

**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.

### 12.13.6 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 illustrated in Figure 12.13.10.

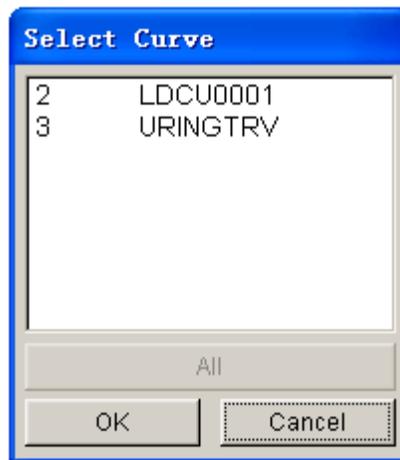


Figure 12.13.10 Select Curve dialog box

- Type in the new ID in input data field of the displayed dialog box.

### 12.13.7 RENAME CURVE

This function enables the user to individually rename the load curves.

- Select a load curve from the list in the SELECT CURVE dialog box.
- Enter the new name in the next dialog box.

### 12.13.8 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 12.13.11.

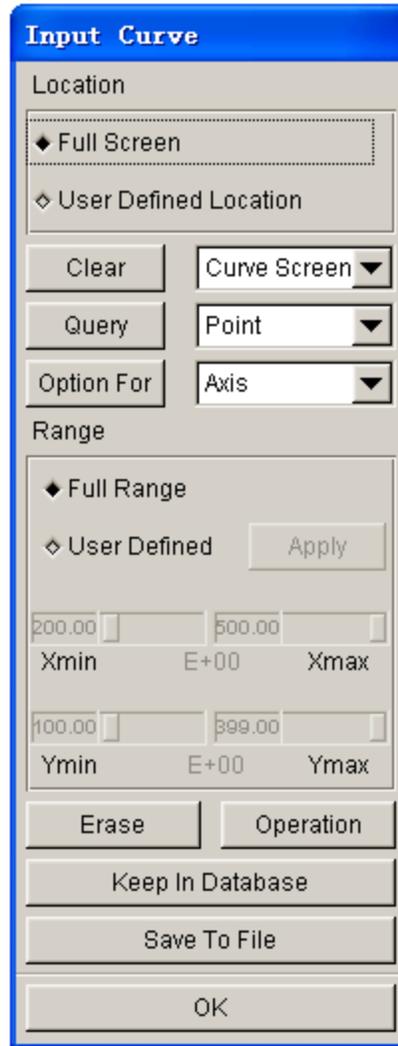


Figure 12.13.11 Show Curve Options dialog box

Select the load curve from the SELECT CURVE dialog box illustrated in Figure 11.13.10.

#### 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 displayed model.

- **ETA LABEL**  
Clear label at the bottom left 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 12.13.12 ~ 12.13.16.

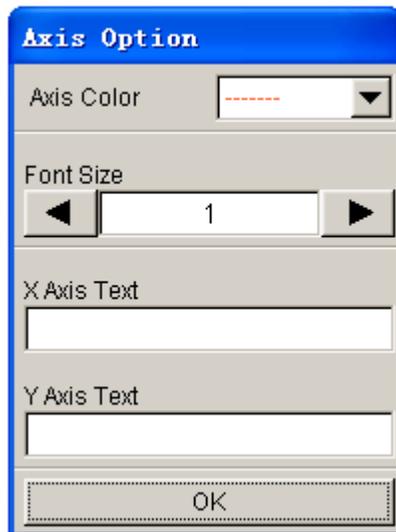


Figure 12.13.12 Axis Options

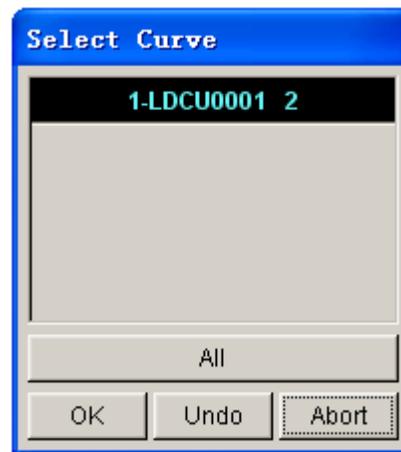


Figure 12.13.13 Select Curve

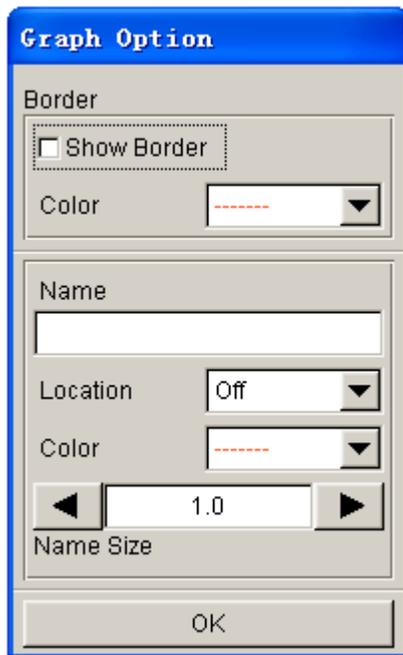


Figure 12.13.14 Graph Options

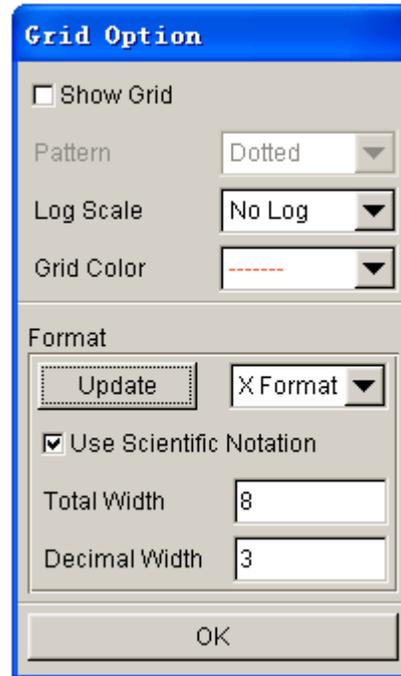


Figure 12.13.15 Grid Options



Figure 12.13.16 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 values of the four sliders. Hit APPLY button to reset the range.

**OPERATION**

This function allows the user to apply mathematical operations to the displayed curve. The options are illustrated in Figure 12.13.17.

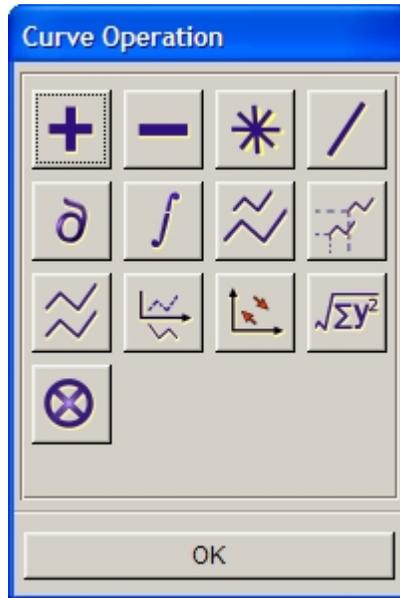


Figure 12.13.17 Curve Operation dialog box

After an operation is selected, the SELECT CURVE dialog box as illustrated in Figure 12.12.18 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***

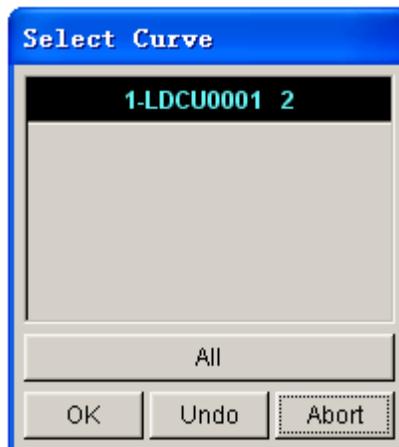


Figure 12.13.18 Select Curve dialog box



Create a new curve in the curve screen of which the values of Y are the sums of the selected curves. Two or more curves must be selected.



Create a new curve using the differences between two selected curves.



Create a new curve using the product of two selected curves.



Create a new curve using quotients of two selected curves.



Create a new curve that is derivative of the selected curve with respect to time.



Create a new curve that is 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.

**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 12.13.19.

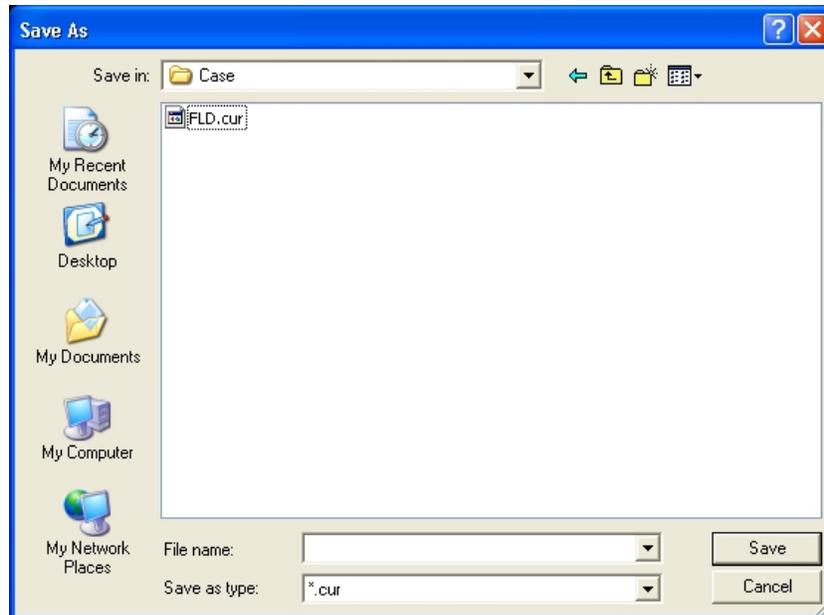


Figure 12.13.19 Save Curves Window

**12.13.9 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 12.13.20 is displayed.

**NOTE:** The D3plot Curve function is available only in Traditional Setup and the Quick Setup (QS).

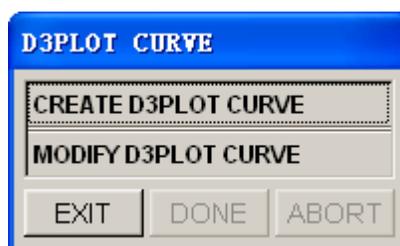


Figure 12.13.20 D3plot Curve dialog box

**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.

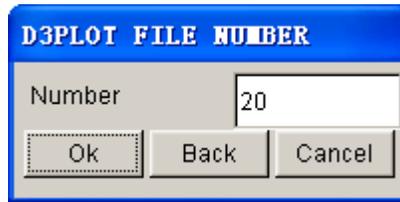


Figure 12.13.21 Define the D3plot file number

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 Figure 12.13.22.

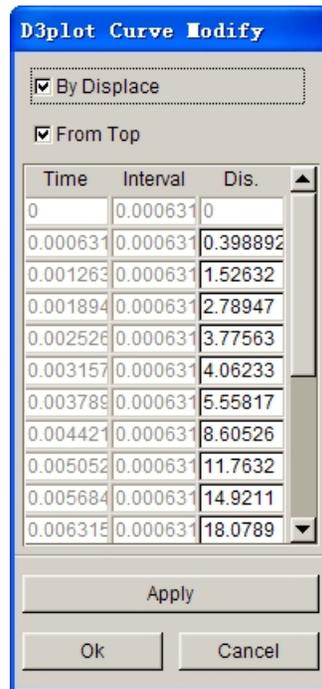


Figure 12.13.22 D3PLOT curve edit dialog box

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.

### 12.13.10 BLANK HOLD CURVE

This function allows the user to import curves with XLS format. This curve is the motion curve picked by the mechanical hand in stamping production.

## 12.14 SHOW LINE

This function is used to identify any existing line and its direction. Refer to Section 5.1 for detailed information.

## 12.15 COORDINATE SYSTEM

The functions in this menu are used to create and modify local coordinate systems. The functions are shown in Figure 12.15.1.

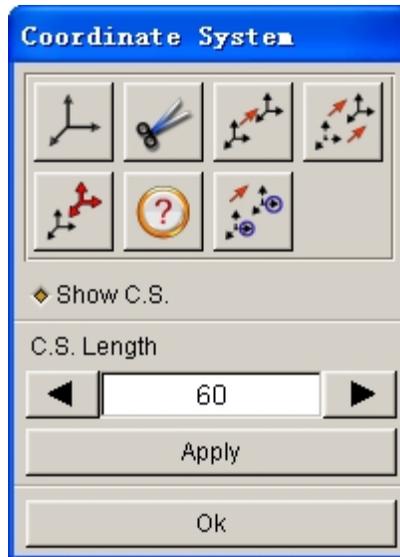


Figure 12.15.1 Coordinate System dialog box

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.

### 12.15.1 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 12.15.2. 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.

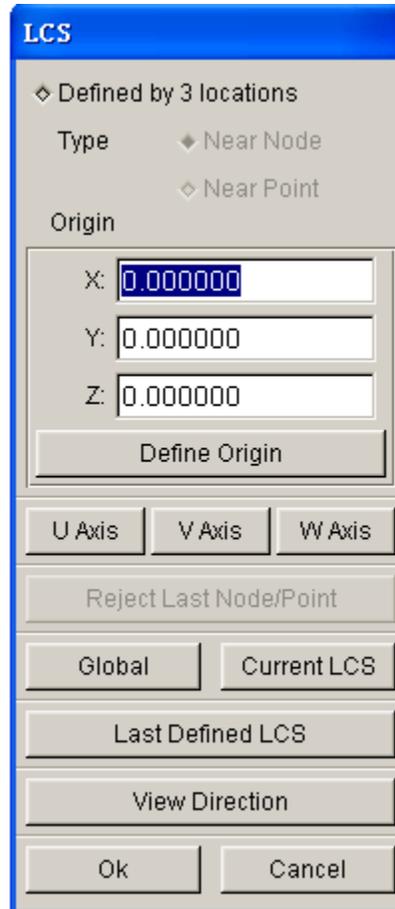


Figure 12.15.2 LCS dialog box

### DEFINED BY 3 LOCATIONS

This function enables user to define a local coordinate system by specifying three nodes/points.

**Near Node:** Define LCS by selecting three nodes. The first selected node is defined as the origin of the LCS. The coordinate value of the defined origin in Global system is also shown. The user can manually edit the coordinate.

**Near Point:** Define WCS by selecting three geometry points. The first selected point is defined as 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 one node/point is defined, the defined node/point sets the origin of the LCS. The local U, V, W axis are parallel to the global X, Y, Z axis, respectively.*

*If two nodes/points are defined, the first selected node/point sets the origin of the LCS. The local W axis is the vector orientation from the first node/point to the second node/point.*

*If three nodes/points area defined, the first selected node/point sets the origin of the LCS. The local U*

*axis is the vector orientation from the first node/point to the second node/point. The selected three nodes/points determines 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 by 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.

## 12.15.2 DELETE COORDINATE SYSTEM



This function enables the user to remove selected local coordinate system from the database. The SELECT LCS dialog box illustrated in Figure 12.15.3 is displayed.



Figure 12.15.3 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

Key in starting and ending coordinate system ID and the increment in displayed dialog box.

### REJECT

Reject last selection.

**NOTE:** To remove the local systems/definitions from individual nodes, the user must reassign the nodes to the GLOBAL COORDINATE SYSTEM.

## 12.15.3 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.

### 12.15.4 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 shown in Figure 12.15.4 is displayed.

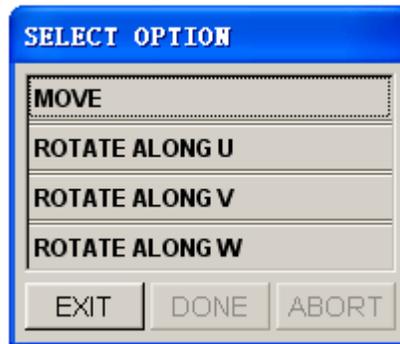


Figure 12.15.4 Modify C.S Options

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.

### 12.15.5 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.

### 12.15.6 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.



### 12.15.7 LCSLCS ENT TRANSFORM

This function allows the user to copy lines, surfaces, elements and nodes as well as their thickness, stress, strain information from one LCS to another, as shown in Figure 12.15.5.

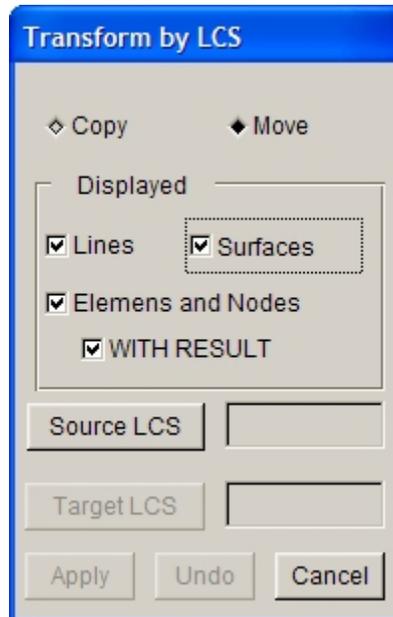


Figure 12.15.5 Transform by LCS

- Select COPY or MOVE.
- Select combination of Lines, Surfaces, Elements and Nodes or With Result options.
- Click on SOURCE LCS button to select the source coordinate system.
- Click on TARGET LCS button to select the coordinate system to be transformed.
- Click on APPLY button to copy or move the selected lines, surfaces, elements and nodes as well as their thickness, stress and strain information from the source LCS to the target LCS.

## 12.16 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 12.16.1.

The screenshot shows the 'Database Statistics' window with the following data:

Geometry Information		Model Information	
Lines	0	Nodes	8779
Points	0	Max Node ID	10987
Surfaces	0	Elements	8348
Edge Points	0	Max Element ID	10532
Surface Boundaries	0	Materials	1
Control Points	0	Properties	1
Real Parameters	0	Parts	10
Integer Parameters	0	Max Part ID	14
Element Information		Local Coordinates	5
Shell	8348	Tool Information	
Solids	0	Blank	Defined
Beams&Rods	0	Punch	Defined
Nodal Rigid Bodies	0	Die	Defined
Spot Welds	0	Binder	Defined
Spring/Dampers	0	User Defined Tools	4
Masses	0	Other Information	
Thick Shells	0	Nodes with Initial Velocity	0
Shell Element Information		Drawbeads	0
Quad.Elements	8236	Ok	
Tri.Elements	112		
Solid Element Information			
Hexahedron Elements	0		
Wedge Elements	0		
Tetrahedron Elements	0		

Figure 12.16.1 Database Statistics window

## Chapter 13

# 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 13.1.

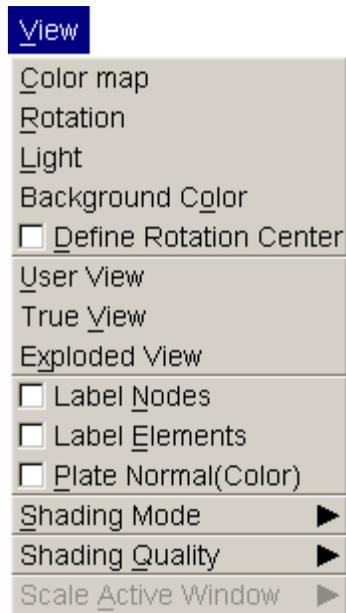


Figure 13.1 the View Options Menu

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

## 13.1 COLOR MAP

This function is used to change the default color in the color bar. The dialog box is illustrated in Figure 13.1.1.

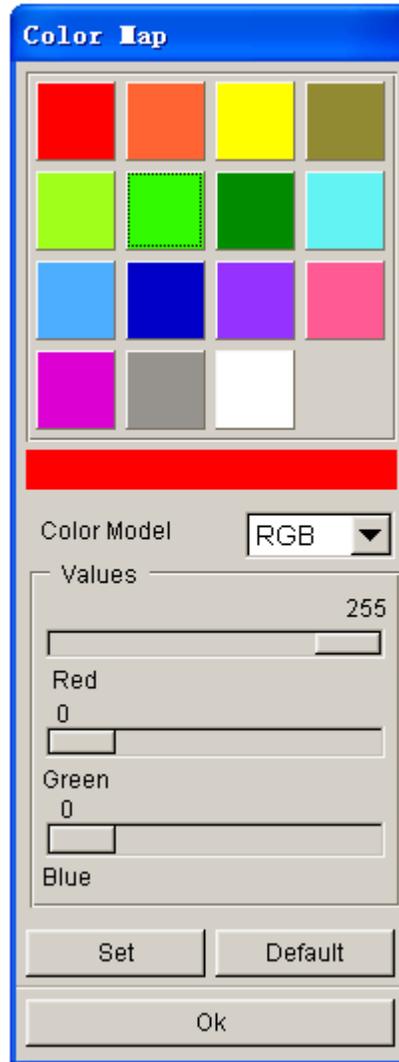


Figure 13.1.1 Color Map dialog box

- 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.

## 13.2 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 13.2.1.

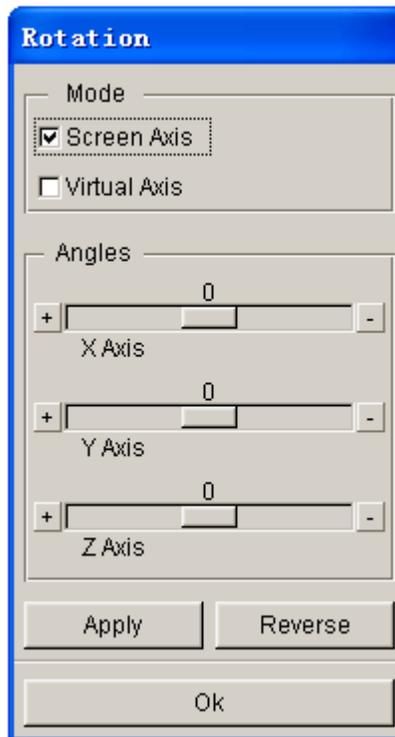


Figure 13.2.1 Rotation dialog box

- **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. SECEEN Y  
Axis direction is from the bottom to the top of the terminal screen.
  3. SECEEN 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.

## 13.3 LIGHT

This function allows the user to simulate the real color of parts according to light conditions in eta/DYNAFORM. An object itself is not luminous, and it is usually illuminated by three types of light: Ambient, Diffuse and Specular. The light source is formed by various lights with different intensity and none of them is purely formed by one type of light.

- **POSITION**

This function enables the user to uniformly move a light source direction with a consistent intensity. The dialog box is shown in Figure 13.3.1. 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.

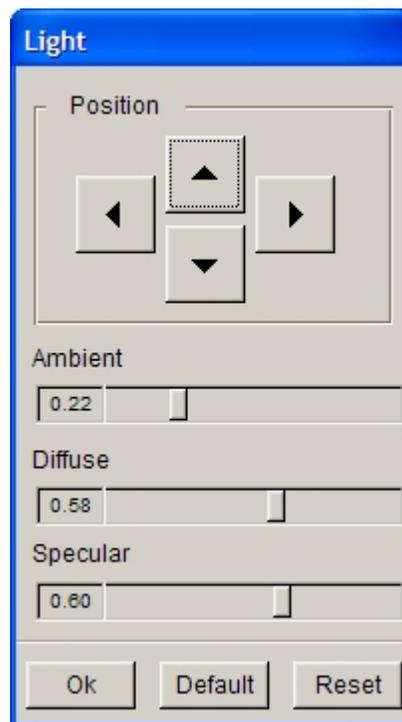


Figure 13.3.1 The Light Source dialog box.

- **AMBIENT**

The ambient light does not come from a specific direction. It comes from a light source, but the light reflects everywhere in the room or scene, without direction. For the object illuminated by ambient

light, the surfaces along all directions are illuminated evenly. The user can control the intensity of ambient light using slider on the interface.

- **DIFFUSE**

Diffuse light comes from a specific direction and is reflected evenly on a surface. However, the object surface directly illuminated by the light is still brighter than the surfaces swept by light from a certain angle. The user can control the intensity of diffuse light using slider on the interface.

- **SPECULAR**

Similarly to diffuse light, the specular light also has direction. Its reflection angle is very sharp and is along a specific direction. High intensity specular light tends to form a bright spot (speckle) on the illuminated surface. The user can control the intensity of specular light using slider on the interface.

## 13.4 BACKGROUND COLOR

This function allows the user to change the background color of display screen. Once this option is selected, the Background Color dialog box illustrated in Figure 13.4.1 is displayed.

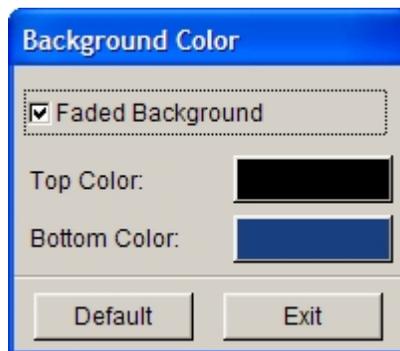


Figure 13.4.1 Background Color

- **FADED BACKGROUND**

Once this option is toggled on, **BOTTOM COLOR** is activated. The program will allow the user to define the background color as faded color. The user can change the color using color button. Click on this button to display a color palette as shown in Figure 13.4.2. The user can select the desired background color from this palette. If Faded Background option is not selected, **BOTTOM COLOR** button will be disabled. The background color is same as the top color.

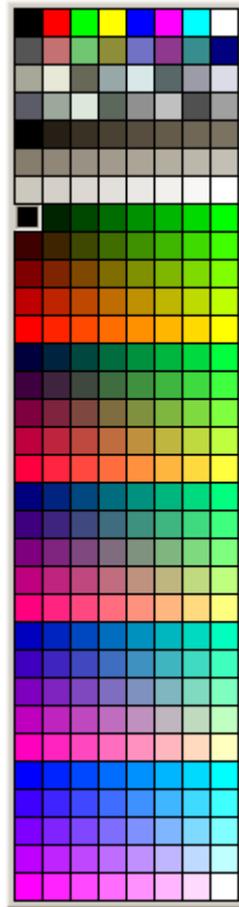


Figure 13.4.2 Color Palette

- **TOP COLOR**  
Define background color for top half of the graphic display window.
- **BOTTOM COLOR**  
Define the background color for bottom half of the graphic display window.
- **DEFAULT**  
Define background color as black.

## 13.5 DEFINE ROTATION CENTER

If this option is toggled on, the user can define the rotation center prior to rotating the model. If the button is hit, 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 shortcut key (CTRL+ left mouse button) is used 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.

## 13.6 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 13.6.1.

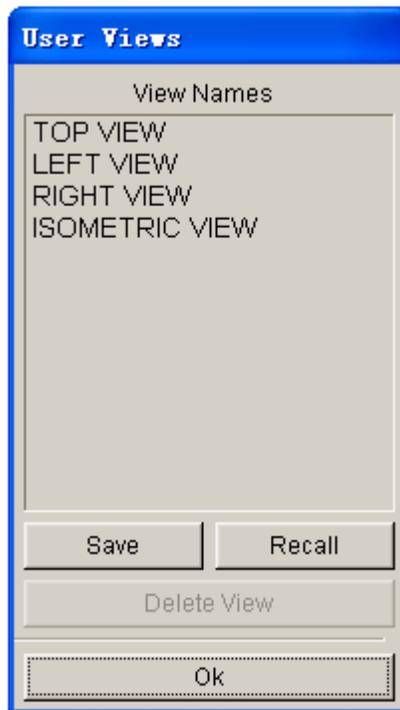


Figure 13.6.1 User Views dialog box

- SAVE**  
 Save 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 illustrated in Figure 13.6.2.



Figure 13.6.2 Save View dialog box

- RECALL**

Display the selected view from the list.

- **DELETE VIEW**  
Remove the selected view from the list.

## 13.7 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 illustrated in Figure 13.7.1 is displayed.

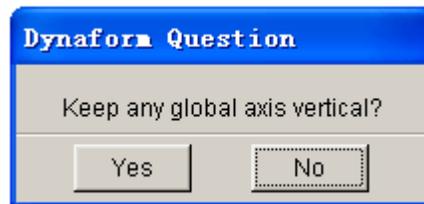


Figure 13.7.1 DYNAFORM Question dialog box

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).

## 13.8 EXPLODED VIEW

This function allows the user to clearly view the relative relationship between components after assembly. The dialog box is illustrated in Figure 13.8.1. The user can click on the drop-down menu to select explode option: Part or Tool. The slider is used to control the EXPLODE SCALE. The user can also click on IDENTIFY NODE and IDENTIFY ELEMENT buttons to identify node and element. Refer to Section 12.9 and 12.10 for detailed description about IDENTIFY NODE and IDENTIFY ELEMENT.

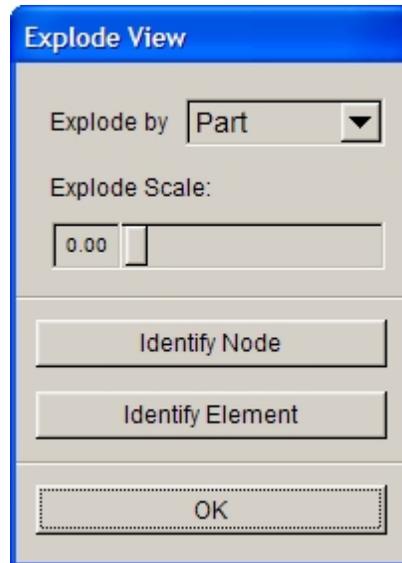


Figure 13.8.1 Explode View

## 13.9 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.

## 13.10 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.

## 13.11 PLATE NORMAL (Color)

This function enables the user to display the normal direction of the elements by color.

## 13.12 SHADING MODE

As shown in Figure 13.12.1, 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.

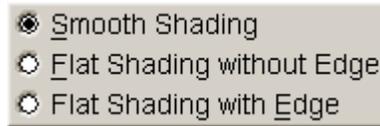


Figure 13.12.1 Shading Mode Menu

### 13.13 SHADING QUALITY

As shown in Figure 13.13.1, 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 to the performance of the computer or the need for high resolution images.

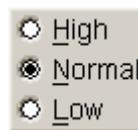


Figure 13.13.1 Shading Quality Submenu

### 13.14 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 14

# ANALYSIS

The functions provided in the ANALYSIS menu, as illustrated in Figure 14.1 allow the user to submit an analysis or to generate an output file.

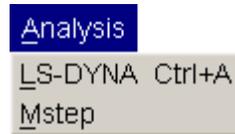
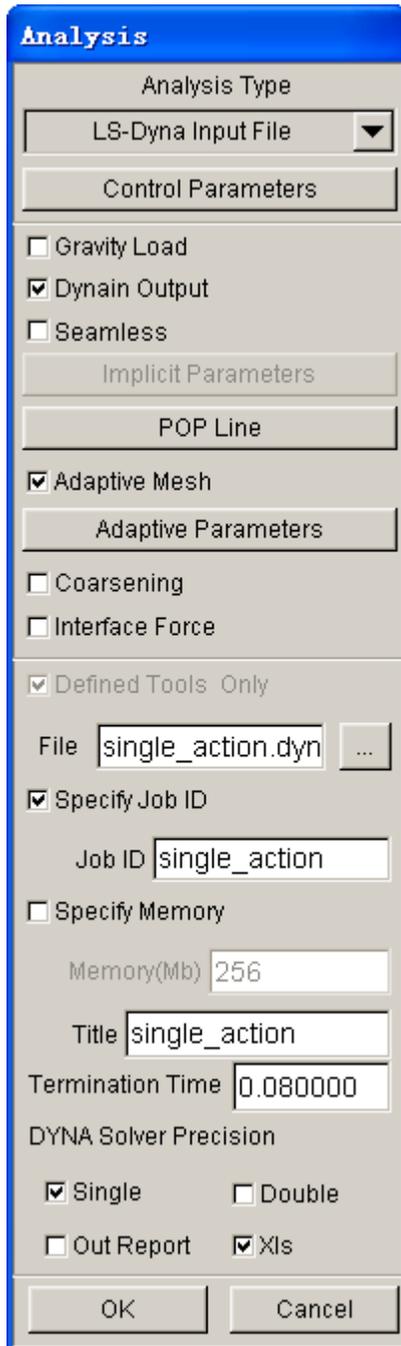


Figure 14.1 Analysis menu

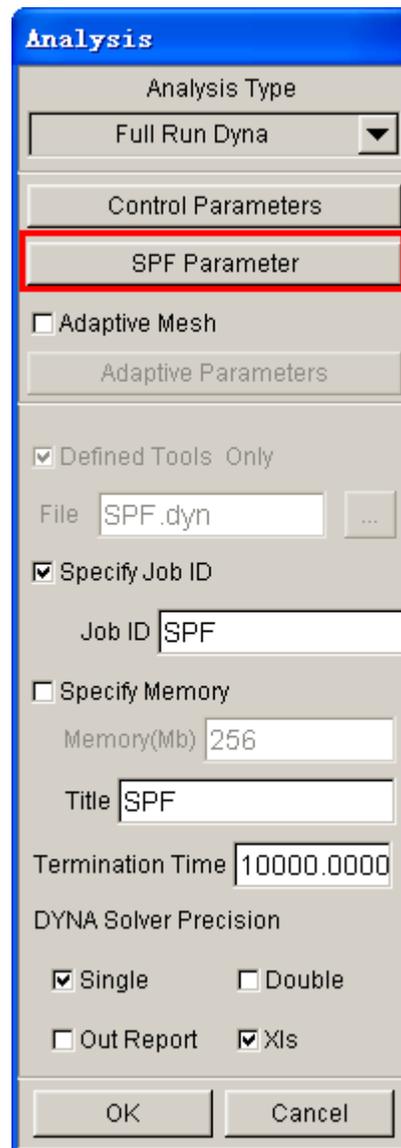
A detailed description of each function in the menu is given in the following sections.

### 14.1 LS-DYNA (CTRL+A)

The LS-DYNA submenu enables the user to set the parameters for stamping simulation, gravity loading and springback analysis. These parameters are written into the analysis output files. If SINGLE/DOUBLE ACTION draw type is chosen, the ANALYSIS dialog box illustrated in Figure 14.1.1a is displayed. If SUPERPLASTIC draw type is chosen, dialog box illustrated in Figure 14.1.1b is displayed.



(a)



(b)

Figure 14.1.1 Analysis dialog box

(a) LS-DYNA Input

(b) Full Run DYNA

### 14.1.1 ANALYSIS TYPE

The user can select the type of analysis output by clicking on the pull-down menu There are three types of analysis output:

1. LS-DYNA INPUT FILE

This function outputs the setup in LS-DYNA input deck.

2. FULL RUN DYNA

This function outputs the setup in LS-DYNA input deck and launches the LS-DYNA to run the current job.

**NOTE:** The user should set the path of LS-DYNA solver in Tools/Analysis Setup menu.

3. JOB SUBMITTER

This function outputs the setup in LS-DYNA input deck and submits simulation to Job Submitter illustrated in Figure 14.1.2. The user can manage the job in Job Submitter.

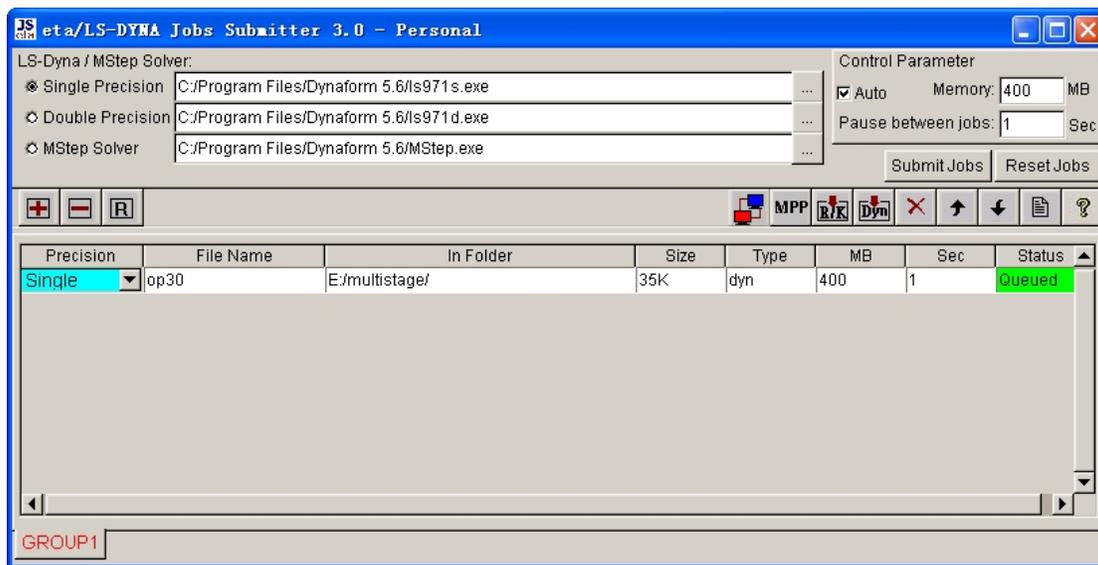


Figure 14.1.2 Job Submitter GUI

### CONTROL PARAMETERS

Set the control parameters for running a stamping simulation using LS-DYNA. The user can edit the parameters by clicking on the CONTROL PARAMETERS button from the ANALYSIS dialog box. The DYNA3D CONTROL PARAMETER dialog box illustrated in Figure 14.1.3 is displayed.

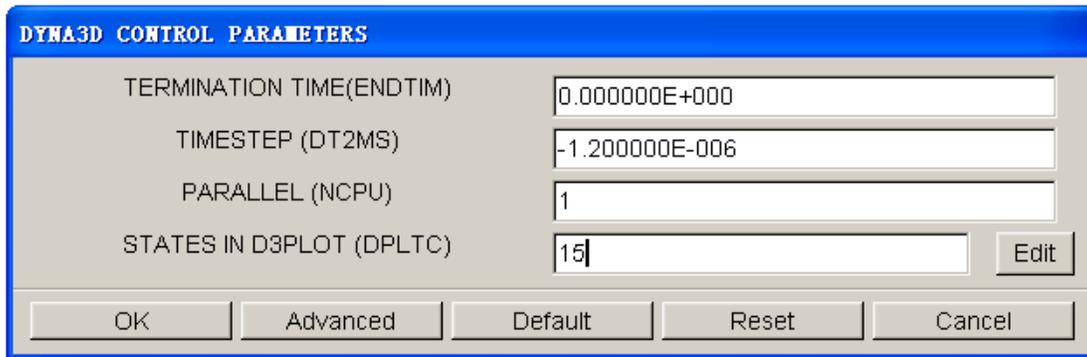


Figure 14.1.3 Control Parameters dialog box

**TERMINATION TIME**

Determine by duration of motion curve. The user may manually define termination time.

**TIMESTEP (DT2MS)**

Time step size for mass scaled solutions. Please refer to keyword \*CONTROL\_TIME\_STEP described in LS-DYNA User's Manual.

**PARALLEL (NCPU)**

Allow the user to define the number of processors used to perform stamping simulation. This function requires PCs equipped with multiple processors, a shared memory (parallel process) version of the LS-DYNA solver, and valid license. Please refer to keyword \*CONTROL\_PARELLEL described in LS-DYNA User's Manual.

**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 keyword \*DATABASE\_BINARY\_D3PLOT described in LS-DYNA User's Manual.

**ADVANCED OPTIONS****HOURGLASS (IHQ)**

Type of hourglass viscosity. Please refer to keyword \*CONTROL\_HOURGLASS described in LS-DYNA User's Manual.

**CONTACT (SLSFAC)**

Scale factor for sliding interface penalties. Please refer to keyword \*CONTROL\_CONTACT described in LS-DYNA User's Manual.

**CONTACT (ISLCHK)**

Initial penetration check in the contact surfaces. Please refer to keyword

\*CONTROL\_CONTACT described in LS-DYNA User's Manual.

#### **CONTACT (SHLTHK)**

Shell thickness considered in SURFACE-TO-SURFACE and NODE-TO-SURFACE contact options. Please refer to keyword \*CONTROL\_CONTACT described in LS-DYNA User's Manual.

#### **CONTACT (PENOPT)**

Penalty stiffness value option. Please refer to keyword \*CONTROL\_CONTACT described in LS-DYNA User's Manual.

#### **TERMINATION (ENDMAS)**

Percent change in the total mass for termination of calculation. Please refer to keyword \*CONTROL\_TERMINATION described in LS-DYNA User's Manual.

#### **INTEGRATION (MAXINT)**

Number of shell integration points. Please refer to keyword \*DATABASE\_EXTENT\_BINARY described in LS-DYNA User's Manual.

#### **OUTPUT TIMES (ENDTIM/DT)**

Multiplier which determines cycle time, DT. Please refer to keyword \*CONTROL\_TERMINATION described in LS-DYNA User's Manual.

#### **ACCURACY STRESS (OSU)**

Global flag for objective stress updates. Please refer to keyword \*CONTROL\_ACCURACY described in LS-DYNA User's Manual.

#### **ACCURACY NODE (INN)**

Invariant node numbering for shell elements. Please refer to keyword \*CONTROL\_ACCURACY described in LS-DYNA User's Manual.

**OK** – Accept the control parameters.

**DEFAULT** - Reset all values to their default settings.

**RESET** - Restore to previously defined value.

**ADVANCED** - Display advanced control parameters.

**CANCEL** – Abort dialog box.

### **14.1.2 GRAVITY LOAD (toggle)**

Activate gravity force during simulation.

### 14.1.3 DYNA OUTPUT (DYNAIN) (toggle)

Create a DYNAIN file at the end of each stamping simulation. If the checkbox of this option is toggled on, the keyword INTERFACE\_SPRINGBACK\_DYNA3D is printed in the input deck. The DYNAIN file contains the deformed blank results such as thickness, stress, strain, etc.

### 14.1.4 SPRINGBACK (SEAMLESS)

Activate the seamless explicit/implicit switch during stamping simulation. The solver will activate implicit springback analysis after the explicit stamping simulation is completed. If checkbox of SEAMLESS option is toggled on, the user can set the springback constraint and define IMPLICIT parameters illustrated in Figure 14.1.4.



Figure 14.1.4 Implicit Parameters dialog box

**NOTE:** Refer to keyword \*CONTROL\_IMPLICIT\_(OPTION) described in LS-DYNA User's Manual.

### 14.1.5 POP LINE

This function enables the user to define the punch opening line for visualization purpose when processing forming result in the eta/POST. The POP line information is output to the index file.

**NOTE:** This function is developed specially for postprocessing result in eta/Post. When the result file (D3plot) is read in eta/Post, the related information listed in “.idx” file such as POP line information is read by eta/Post.

The user can click on the POP line button to display the POP line dialog box illustrated in Figure 14.1.5.

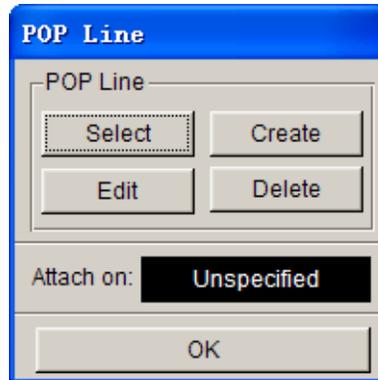


Figure 14.1.5 Define POP LINE dialog box

### SELECT POP LINE

This function enables user to define the POP line through selecting the existing single or multiple line segments available in current database. The SELECT LINE dialog box is displayed after the Select button is hit. Each time the POP LINE dialog box is displayed, the defined POP line is highlighted on display screen.

### CREATE POP LINE

This function enables user to create the POP line from the Addendum defined in DFE module. Prior to using this function, the user has to create the addendum using DFE. 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 5.1.15 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 reference lines will not be deleted from the database.*

### ATTACH ON

This function enables user to attach POP line to a selected part from the Part list. In eta/POST, the POP line moves together with the defined part. Usually, the POP line is attached to DIE.

After all parameters of the POP line are defined, the user can click the OK button to exit the POP line dialog box. When LS-DYNA input deck is output from eta/DYNAFORM, the POP line information is written into an index file with “.idx” extension.

### 14.1.6 ADAPTIVE MESH (toggle)

If this option is toggled on, the mesh refinement scheme is activated during the stamping simulation. The user can edit the adaptive parameters by clicking on the ADAPTIVE PARAMETERS button. The default parameters are listed in ADAPTIVE CONTROL PARAMETERS dialog box illustrated in Figure 14.1.6. The ADVANCED button enables the user to set other adaptive parameters.

**NOTE:** Refer to keyword *\*CONTROL\_ADAPTIVE* described in *LS-DYNA User's Manual* for detailed information about each parameter.

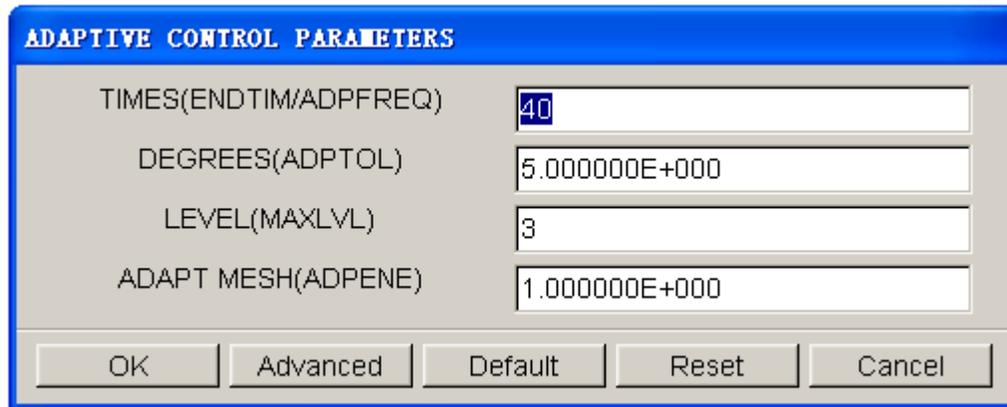


Figure 14.1.6 Adaptive Control Parameters dialog box

### 14.1.7 DEFINED TOOLS ONLY (toggle)

If the option is toggled on, only defined tools are output to the input deck. Otherwise, all parts in the eta/DYNAFORM database are output.

### 14.1.8 SPECIFY OUTPUT FILENAME

The user can specify a file name of the output. The default file name is similar to the name of current database. The user can directly input any file name in the FILE NAME FIELD or click the button to select the path for saving the file.

### 14.1.9 SPECIFY JOB ID (toggle)

This is a new feature for LS-DYNA970 or latest version to create prefix to the result files. If the option is toggled on, the input data field of Job ID is activated allowing the user to enter specific JOB ID for the analysis. Default JOB ID is the database file name without the “.df” extension.

### 14.1.10 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 words and write the value in the field of the \*KEYWORD card in the input deck.

### 14.1.11 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.

### **14.1.12 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.

### **14.1.13 DYNA SOLVER PRECISION**

#### **SINGLE**

This option is enabled 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 enabled 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 basic setup information into a file for reference and communication purpose. The default file name is similar to the name of the current database. The default file format is XLS. If the checkbox for XLS option is toggled off, the file is saved in CVS format.

#### **XLS**

This option allows the user output the setup information in XLS format. A typical example of XLS file is illustrated in Figure 14.1.7.

⊕ <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	Integration 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			
Dies-Binder clearance(mm)	4.00	Dies-Punch clearance(mm)	44.00	
Binder Begin Time				
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 14.1.7 A typical XLS file

## 14.2 LS-DYNA (SPRINGBACK)

If the draw type is set as SPRINGBACK (TOOLS/ANALYSIS SETUP/DRAW TYPE), the Analysis dialog box illustrated in Figure 14.2.1 is displayed.

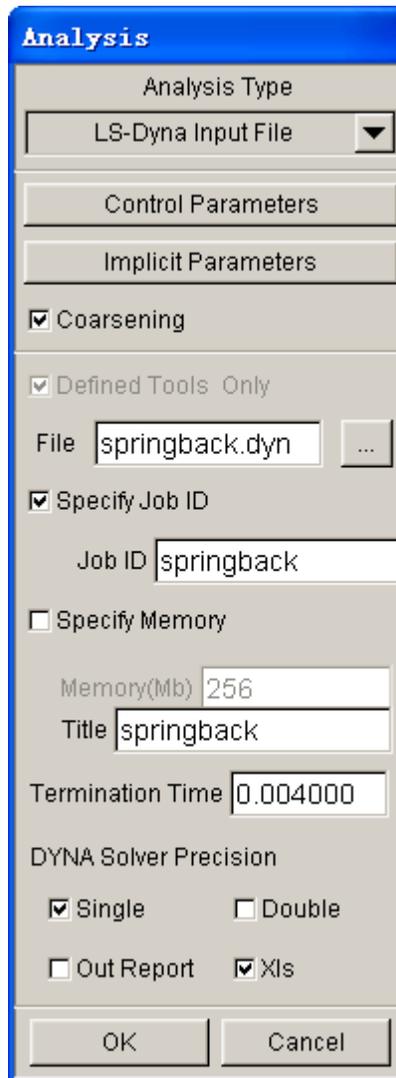


Figure 14.2.1 Springback Analysis dialog box

These functions are discussed in Section 14.1, LS-DYNA.

**NOTE:** Refer to the LS-DYNA User's Manual for more information.

## 14.3 LS-DYNA (GRAVITY LOADING)

If the draw type is set to GRAVITY (TOOLS/ANALYSIS SETUP/DRAW TYPE), the Analysis dialog box illustrated in Figure 14.3.1 is displayed.

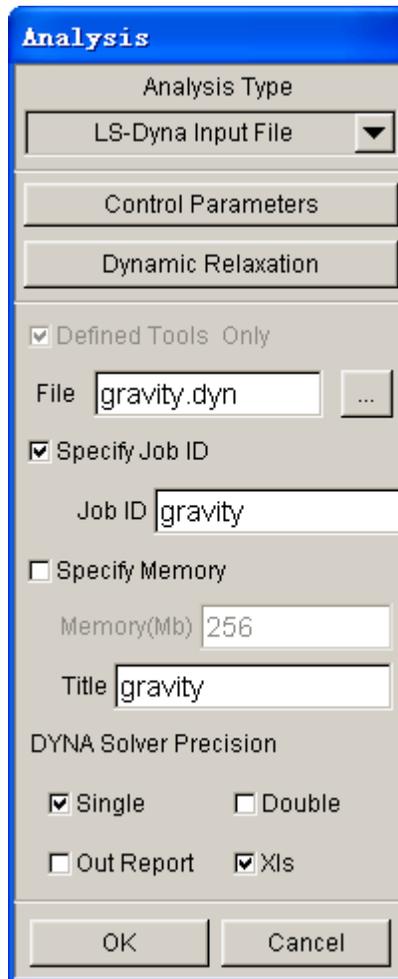


Figure 14.3.1 Gravity Loading dialog box

**NOTE:** Refer to keyword *\*CONTROL\_DYNAMIC\_RELAXATION* described in *LS-DYNA User's Manual* for detailed information about each parameter.

## 14.4 MSTEP

MSTEP is a brand new modified-one-step solver embedded in eta/DYNAFORM to quickly estimate formability and blank outline in the early stages of the stamping design and development cycle.

Please refer to Section 7.2 for detailed information about MSTEP.

## 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 971. It will run in Windows 98, 2000, and XP environments. It is not recommended for usage with earlier versions of Windows.

**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**

<b>NAME</b>	<b>TYPE</b>
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 reliable 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: +1 (248) 729 3010

Fax: +1 (248) 729 3020

E-mail: [support@eta.com](mailto:support@eta.com)

The eta/DYNAFORM Team  
Engineering Technology Associates, Inc