eta/DYNAFORM
User's Manual

Version 5.2

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INTRODUCTION

eta/DYNAFORM Version 5.2 software is an **LS-DYNA-based sheet metal forming simulation solution package** developed by Engineering Technology Associates, Inc. This specialty CAE software combines the analysis power of LS-DYNA Versions 960 and 970 with the streamlined pre- and post-processor functions of eta/DYNAFORM. 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 cycle 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 solver approaches for fluid and solid structural problems. This 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 cost of producing high quality panels and 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 panel stamp in areas such as ***cracking***, ***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 NT operating systems of 1998 and above. For LINUX, the RedHat version 7.2 and above is supported.

Special features of eta/DYNAFORM Version 5.2 include:

Die Face Engineering – Significantly enhanced features and functions to support die face geometry:

- Inner Binder function enables the user to create multi-binders.
- Enhanced edit function of the binder and the binder trim function. The binder surface of two side of the symmetry is smoother in the joint.
- The grid function is added to select point when define section line for Free Form Binder.
- Enhanced the addendum function such as copy, move and symmetry the profile, split and show the addendum, Elongation of one profile, create profile line and addendum surface.

Added MSTEP function

MSTEP – An iterative One-Step code to support blank size estimate applications and quick formability assessment. Enables the user to gain the forming result and the original outline of blank in few minutes. Evaluation Report can create a detailed BSE result report for the user.

Added Nesting function

Nesting of the blanks account for the carrier, supporting progressive tooling applications. One-up nesting, Two-pair nesting, Two-up nesting, Two different blanks nesting methods are developed according to production requirement. The utilization of the blank and other nesting information can be print in the Nesting report.

Default Parameters and Configuration Control

eta/DYNAFORM provides a two-level authorization system to enable the supervisor or super-user to implement default values and parameters in addition to customizing processes and applications.

The DYNAFORM DEFAULT file, located in both the program installation directory and the user's home directory, allows a user to modify default parameters including file extensions, contact parameters, adaptive mesh parameters, output control parameters, etc.

A detailed explanation regarding the default parameters is provided at the beginning of the DYNAFORM DEFAULT file.

Enhance partly functions in Pre-processor

1. Added circular region option for node, element and surface selection.
2. Added create node on line function in pre-processor.
3. Enhanced the edit function for Load Curve
4. Updated DYNA input file to support LS-DYNA 970

Updated DYNA input file to support LS-DYNA 970

The DYNA file created by DF 5.2 can support LS-DYNA 970.

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 the draw die simulation of an S-channel using LS-DYNA that describes the process of setting up and running a stamping simulation.

A step-by-step procedure guides users through the complete process and flow of the 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. Binder Wrap Simulation
Case 2. NUMISHEET'93 Fender Simulation
Case 3. Cup Draw Simulation with Adaptive Mesh
Case 4. S-Rail NUMISHEET'96 Simulation with Spring Back
Case 5. Hydro Forming 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 spring back simulation.

The **User's Manual** contains function descriptions specific to the DYNAFORM interface.

For further details on LS-DYNA, refer to the manuals for these codes published by the Livermore Software Technology Corporation.

CHAPTER 1
STRUCTURAL OVERVIEW

1.1 GENERAL

eta/DYNAFORM Version 5.2 is a complete Graphic User Interface (GUI) package that is operated on Windows (95 and above) and on UNIX/Linux based workstations (W/S) including IBM, HP, DEC-Alpha, SGI and SUN-SOLARIS platforms using various popular operating systems. The model generation and input file preparation of a typical stamping simulation are done in the eta/DYNAFORM preprocessor, but the results can be processed in another environment. Refer to the Post-processor User's Manual. 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 Windows and UNIX platforms.

1.2 MENU BAR

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

Figure 1.1

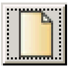
FILE MANAGER	Imports and exports data from eta/DYNAFORM.
PART CONTROL	Organizes Parts (or layers).
PREPROCESS	Contains the menu of preprocessing functions, lines/points, elements, nodes, check functions, boundary conditions.
DFE	DIE FACE ENGINEERING provides tools to build an addendum and binder.
BSE	BLANK SIZE ENGINEERING includes a modified-one-step (MSTEP) scheme based unfold function, Blank Development functions, and Blank Nesting function.
QuickSetup	The QuickSetup menu provides a streamlined, easy way to set up certain standard stamping simulations.
TOOLS	Creates, defines, and modifies tooling definitions.
OPTION MENU	Contains various options to control the mesh and file window style.
UTILITIES	Includes several supplemental functions for identifying entities.
VIEW OPTIONS	Adjusts the display of the items and view angle on the screen.
ANALYSIS	Defines the control parameters, outputs the DYNA input deck, and submits jobs.
PostProcess	Starts eta/POST for post-processing analysis results and automatically closes DYNAFORM.

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 clicks on these icons to activate the functions instead of browsing the various menus. See Figure 1.2.

Figure 1.2 Tool bar



NEW

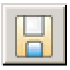
Creates a new database file.

OPEN

Opens a database.

IMPORT

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

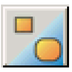


SAVE

Updates the current database.

PRINT

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



PART 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 free nodes on which there are no associated 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 SX and SY. Moving the mouse up/down manipulates SX. Moving the mouse left/right manipulates SY. Moving the mouse diagonally combines the movements of both commands. Clicking the left mouse button stops the rotation. This function can be executed by simultaneously pressing the Ctrl key and left mouse button.

PAN

This command enables the user to translate the model by following the movement of the cursor. If the cursor is moved off the screen, it reappears at the center of the screen. Clicking the left mouse button stops the pan. This function can be executed by simultaneously pressing the Ctrl key and middle mouse button.

CURSOR ZOOM

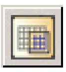
The user picks a point about which to zoom. The model is centered about this point, and the user may move the cursor up or down to zoom in or out. This function can be executed by simultaneously pressing the Ctrl key and right mouse button.

WINDOW ZOOM

The user defines the corners of the zoom window by positioning the cursor on the display screen. The user presses the left mouse button and drags the cursor diagonally down until the desired window size is reached. Releasing the left button displays the entities included in the window in full screen.

FREE HAND ZOOM

The user defines a free region by pressing the left mouse button and dragging the cursor on the display screen. Releasing the left button displays the entities included in the region in full screen.



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 over 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 the model to include all entities that are currently displayed. FILL automatically zooms in or out until the model fits the viewing area of the screen.


TOP VIEW
Automatically displays the model from the TOP or in the XY-plane.


LEFT VIEW
Automatically displays the model from the LEFT or in the XZ-plane.


RIGHT VIEW
Automatically displays the model from the RIGHT or in the YZ-plane.


ISOMETRIC VIEW
Automatically displays the model from the isometric plane (120-degree isometric).


CLEAR
Removes 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 an additional step (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 represent the element normal).


UNDO
Undoes the last operation. If there is nothing to undo, the icon is disabled.


REDO
Redoes the last operation. If there is nothing to redo, the icon is disabled.

1.4 DISPLAY WINDOW

eta/DYNAFORM breaks the screen into six distinct regions. The regions are used to receive input or display messages for the user. See Figure 1.3. The six regions are illustrated and described below:

1. DISPLAY AREA
   This area displays models and graphs.

2. MENU BAR
   This menu displays commands and command options.

3. ICON BAR
   Gives the user easy access to the most often used functions of eta/DYNAFORM.

4. DIALOG WINDOW DISPLAY
   Once the user selects a command from the MENU BAR, a corresponding dialog window with the appropriate functions is displayed in this area.

5. DISPLAY OPTIONS
   This group of commands is always displayed and can be used at any time during an eta/DYNAFORM session.

6. PROMPT AREA
   eta/DYNAFORM displays comments and messages.

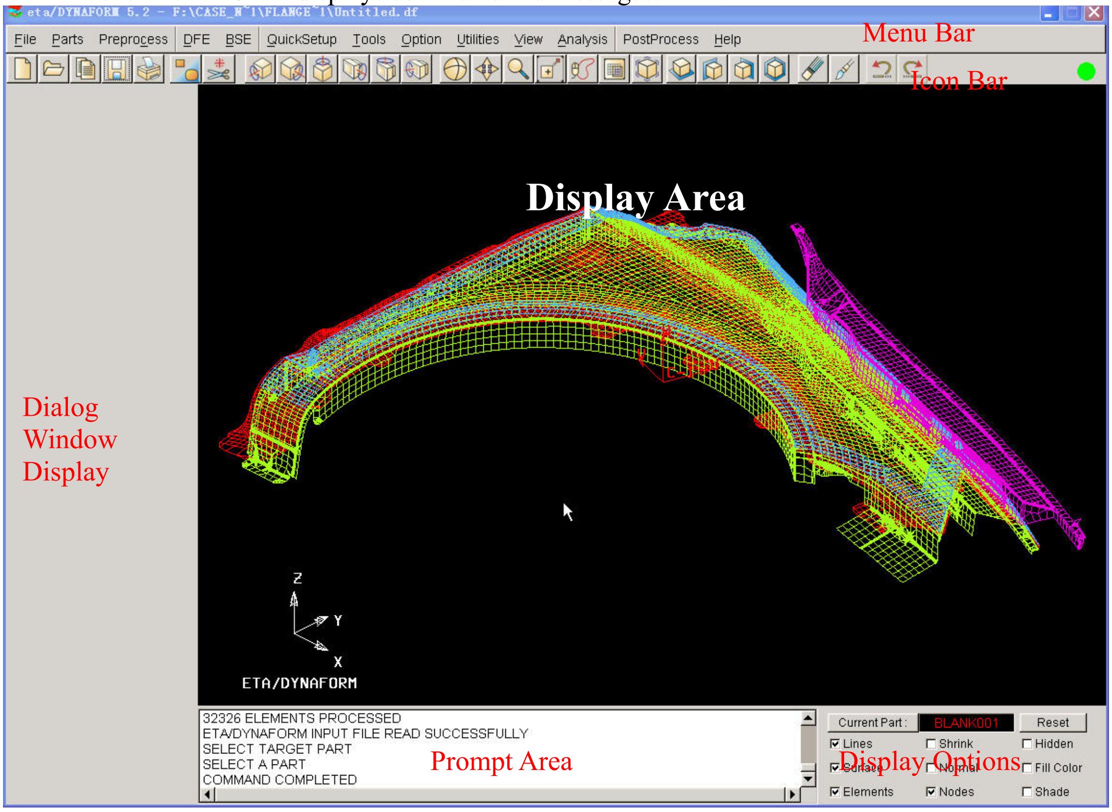
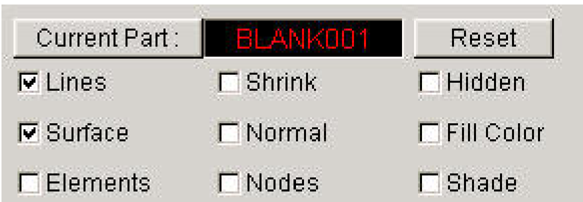


Figure 1.3 Interface of eta/DYNAFORM 5.2

1.5 DISPLAY OPTIONS

The DISPLAY OPTIONS window, located in the bottom right corner of the screen, displays the current part and contains the following commonly used functions.



RESET
Restores all options to the default.

LINE (toggle)
Toggles the lines ON and OFF.

SURFACE (toggle)
Toggles the surfaces ON and OFF.

ELEMENT (toggle)
Toggles the elements ON and OFF.

SHRINK (toggle)
Creates a plot with elements reduced in size by 20%. SHRINK ELEMENTS is also useful for locating any missing elements in a shell or solid structure.

NORMAL (toggle)
This function enables the user to display the normal direction of an element with an arrow that is 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.

NODE (toggle)
Toggles the nodes ON and OFF.

HIDDEN (toggle)
This function improves the integrity of the 3D simulation of a model. The user may toggle the hidden lines ON/OFF while using the SHADING and HIDDEN SURFACE commands. 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 the displayed elements with a designated color. This function, when used alone, is unable to accurately represent the depth perspective of a model. The parts may appear to warp or penetrate each other. However, the FILL COLOR command displays an accurate 3D perspective of the parts when used in conjunction with the HIDE PLOT option that is described in the following section.

SHADE (toggle)
Displays the elements as if they were illuminated by a light source. Elements 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 selects the desired function button by using the mouse pointer and depressing the left mouse button. In certain functions, the middle mouse button finishes some operations, such as creating a line and selecting nodes.

1.7 SPECIFICATIONS

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

150,000	LINES
900,000	POINTS
8,000	SURFACES
600,000	Edge Points (SURFACE)
16,000	Boundaries (SURFACE)
500,000	ELEMENTS
500,000	NODES
1,000	PROPERTIES
1,000	MATERIALS
1,000	PIDS
1,000	LOCAL COORDINATE SYSTEMS
8	LENGTH OF PART, MATERIAL, PROPERTY NAME

1.8 GEOMETRY DATA

eta/DYNAFORM directly reads in IGES, VDA and eta/DYNAFORM/FEMB geometry data (line and surface). It also directly reads in CATIA4 & 5 and UG part with native library.

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:

1. eta/DYNAFORM database file name: *filename.**df***
2. eta/DYNAFORM geometry data file name: *filename.**lin***
3. IGES geometry data file name: *filename.**igs***, **or** *filename.**iges***
4. VDA surfaces data file name: *filename.**vda***, **or** *filename.**vdas***
5. Data Exchange file name: *filename.**dxf***
6. Stereolithography file name: *filename.**stl***
7. ACIS file name *filename.**sat***
8. CATIA4 database file name: *filename.**model***
9. CATIA5 database file name: *filename.**CATPart***
10. STEP file name: *filename.**stp***
11. UG database file name: *filename.**prt***
12. Pro-E database file name: *filename.**prt*** or *filename.**asm***
13. NASTRAN input file name: *filename.***nas** *or filename.**dat***
14. LS-DYNA input file name: *filename.**dyn***
15. LS-DYNA model file name: *filename.**mod***
16. LS-DYNA input file name: *filename.**k***
17. LS-DYNA input file name: ***dynain***
18. ABAQUES input file name: *filename.**inp***

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 can click 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

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.

ABORT- Aborts the current function and exits the dialog box.
APPLY- Executes the current function without leaving the dialog box.
BACK- Returns the user to the previous dialog box.
CANCEL - Rejects the current operation.
CLOSE - Closes the current dialog box.
DONE - Finishes the current step in the dialog box and allows the user to proceed to the next step.
EXIT - Exits the current dialog box.
OK - Accepts the data in the dialog box and forwards the user to the next step.
UNDO - Rejects the last step of the operation.
REJECT - Rejects the previous selection.

1.11 PROPERTY TABLES

The tables in eta/DYNAFORM allow the user to enter properties for materials, elements, draw beads, etc. See Figure 1.4. The data fields contained in the table can be edited in several different ways. Single-clicking the mouse button inserts the cursor at a specific point in the data field; single clicking and dragging the cursor highlights a select portion of the field; double clicking highlights the entire field. Keystroke entry then inputs the new value. Altered values will be remembered by the table and will remain until reset. Most parameters have been organized into two groups: standard and advanced. Only the standard parameters are shown on screen (see figure below). At the bottom of the table the OK, ADVANCED, DEFAULT, RESET, and CANCEL buttons allow the user to accept or reject the data. The ADVANCED button is implemented in the following dialog boxes: MATERIAL, PROPERTY, CONTACT PARAMETERS, CONTROL PARAMETERS and ADAPTIVE PARAMETERS.

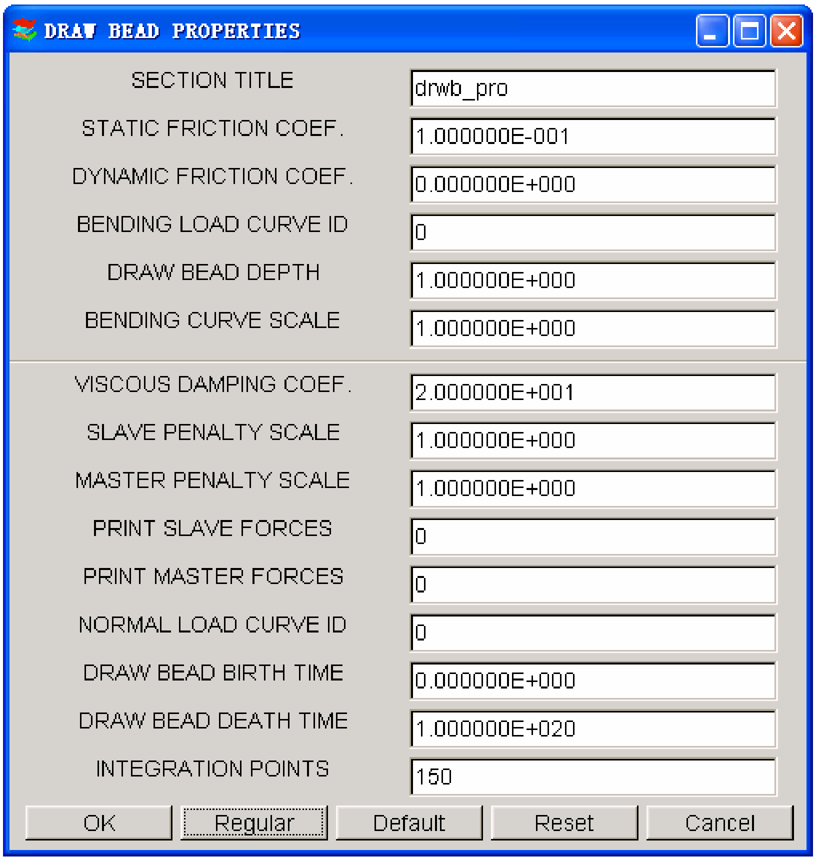


Figure 1.4 Draw Bead Properties window

OK - Accepts the currently displayed values and exits the table.

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

DEFAULT - Sets all fields to their default values.

RESET – Resets the last altered field to its previous state.

CANCEL – Exits the table without entering any altered values.

1.12 CONFIGURATION FILE

Three configuration files (**.dynaformDefault**, **dynaformhardcopydefault** and **dynaform.ini**) are created to set all the control parameters. They are, by default, located in the directory of current user's "My Document" on PC. They are located in the user's home directory on UNIX/LINUX. This file requires super-user privilege to modify.

DynaformDefault primarily controls the DYNAFORM default setting of analysis parameters. Each parameter has a control flag to control permission for modification. The format of each control parameter appears as follows:

"Keyword value permission key".

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

Dynaformhardcopydefault controls the DYNAFORM default setting of the print. The format of each control parameter appears as follows:

"Keyword value"

The user can modify the default settings of most parameters related to the print.

dynaform.ini is provided to save the customized setting in the database or in the user's home directory. The format of each parameter is as follows:

```
#DYNAFORM5.2 configure file
    #[keywords]      value
    #[FILE_DIALOG_STYLE]: 0 - WINDOWS style, 1 - UNIX STYLE
    #[SYSTEM_LANGUAGE]: 0 - ENGLISH, 1 - CHINESE, 2 - JAPANESE, 3 - KOREAN
    #[AUTOBACKUPTIME]: <=0 - don't autobackup, >0 - backup time( minutes )
    #[AUTORESTART]: 0 - don't autorestart, 1 - autorestart.
    [FILE_DIALOG_STYLE]
    0
    [SYSTEM_LANGUAGE]
    0
    [LAST_DATABASE_PATH]
    D:/DYNAFORM_MATERIAL/DFE_Tutorial_manual/Case4_HoodInner/
    [LAST_IMPORT_PATH]
    D:/dynaform52/dir_dfe/
    [LS-DYNA_SOLVER]
    D:\DYNAFO~2\lsdyna.exe
    [AUTOBACKUPFILE]
    timed.df
    [AUTOBACKUPTIME]
    0
    [AUTORESTART]
    0
```

Comment lines start with '#'.

[FILE_DIALOG_STYLE] defines the default dialog file style. There are two file dialog styles (Windows and UNIX) under the Option/File option of DYNAFORM 5.2.

[SYSTEM_LANGUAGE] defines the default language for GUI. DYNAFORM supplies four languages.
[LAST_DATABASE_PATH] When saving or opening a database, DYNAFORM 5.2 will automatically locate the last saved or opened database path.

[LAST_IMPORT_PATH] When importing or exporting a database, DYNAFORM 5.2 will locate the last import or export database path automatically.

[LS-DYNA_SOLVER] When full running a model, DYNAFORM will find the solver from the path.

[AUTOBACKUPFILE] defines the auto back file name.

[AUTOBACKUPTIME] defines the time interval of auto backup.

[AUTORESTART] Postprocess can be started directly in DYNAFORM. If the value is 1, the database (*.df file) will be restarted automatically after eta/Post is closed.

CHAPTER 2
GETTING STARTED

DYNAFORM is organized into two major modules, a formability simulation module, and a Die Face Engineering module. Both can begin with geometry data or mesh.

2.1 GETTING STARTED

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

1. The user supplies CAD-IGES or VDA or any other geometry data that will be translated and read directly into eta/DYNAFORM.

2. The user supplies model data (e.g. NASTRAN, LS-DYNA, etc.) that will be read directly into eta/DYNAFORM.

3. CAD or model data is not supplied. Therefore, the user begins with an empty database and generates or digitizes the line data from a drawing.

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

A normal simulation includes the following steps.

2.2 STARTING WITH CAD DATA

The user begins a session by entering the word, DYNAFORM, at the UNIX prompt, or double clicking on the DYNAFORM icon at the Windows/Linux. Once the program is activated, the default database is automatically created and called, Untitled.df.

2.2.1 STARTING WITH CAD *SURFACE* DATA

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

NOTE: If IGES is selected, files having the suffixes .igs and .iges will be listed in the menu area. Please refer to section 1.9 to get more information about the file types.

If Line Data is selected, files having the suffix **.lin** will be listed.

2.2.2 STARTING WITH CAD *LINE* DATA

Users with line data 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, surfaces can be generated with lines via the SURFACE option menu located in the PREPROCESS /SURFACE menu. The surfaces created can be auto meshed via the SURFACE MESH command found in the ELEMENT OPTIONS menu. The line data can be read 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 button to the right of FORMAT and choose the desired format.

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 window.

2.4 GETTING STARTED FROM SCRATCH

This section will help users getting started in eta/DYNAFORM when line or model data is unavailable. Users will typically generate their 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. See also Section 6.1.1 in Chapter 6, 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. See 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 4.3, ELEMENT.

Once the desired geometry has been created, 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/Thickness and Tool/Blank Positioning can also be defined. Once the tools are defined, the interface between the tools will be automatically 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.

After the forming analysis is complete, the result file dynain can be read in and the user can view the formed blank shape. This file is now ready for subsequent springback analysis or multi-step stamping. A trimmed blank model can be written out into a dynain input file from the ANALYSIS/output new dynain file. A blank may be trimmed using the TRIM command of BLANK OPERATIONS in the TOOL DEFINITION menu. Once the analysis is complete, the results can be viewed by reading the d3plot files into the POSTPROCESS.

Please refer to QUICK SETUP to get the detailed simulation procedure using Quick Setup.

Please refer to DFE to get the detailed steps to engineer the die face.

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 window 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.

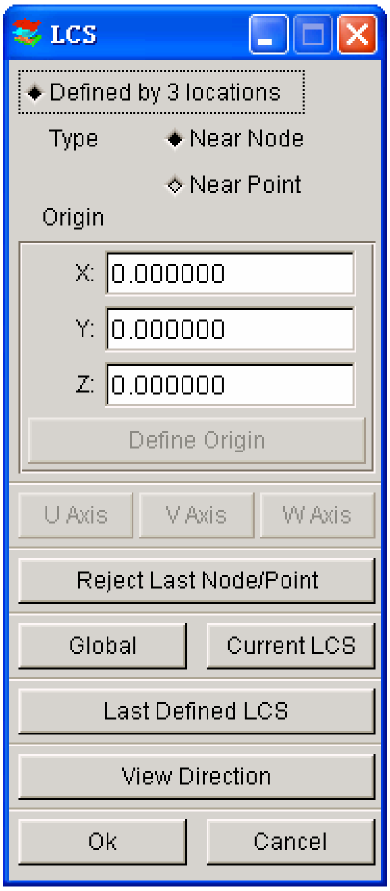


Figure 2.5.1 LCS

GLOBAL

Defines the global coordinate system as the current local C. S.

CURRENT LCS

Creates a local coordinate system as the current local C. S.

LAST

Defines the last coordinate system as the current local C. S.

VIEW DIRECTION

Defines the view directions as the W-Axis of the current local C. S. 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 local C.S

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

1. CHECKED
   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
   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, a dialog is displayed. The choices are ALONG X, ALONG Y, ALONG Z, ALONG U, ALONG V, ALONG W in the next dialog window. Select one, and then 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.

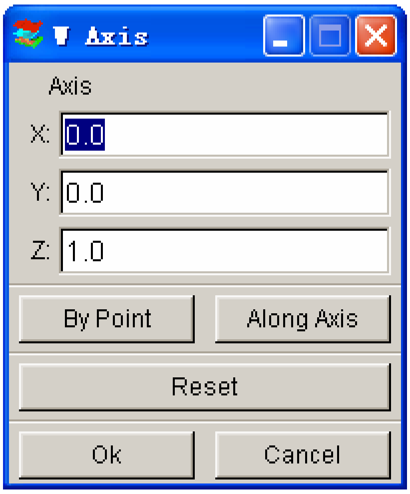


Figure 2.5.2 Define W Axis

CHAPTER 3
FILE MANAGER

The options in this menu open, save, import, export, and print current files. See Figure 3.1.

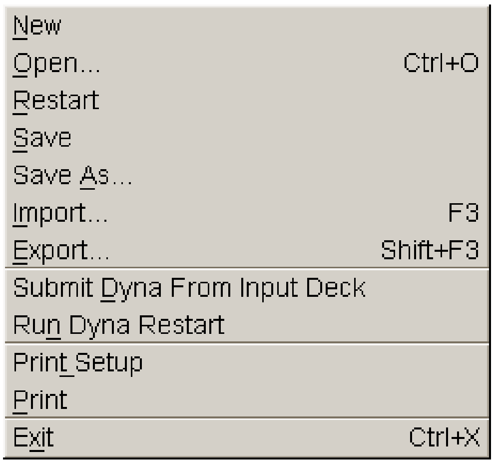


Figure 3.1 File Manager

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 prompts to save the opened file before creating a new one. See Figure 3.1.1.

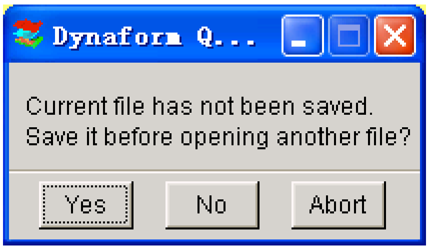


Figure 3.1.1 New File Prompt

- 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 there is already an opened database, eta/DYNAFORM prompts the user to save the current file. See Figure 3.2.1.

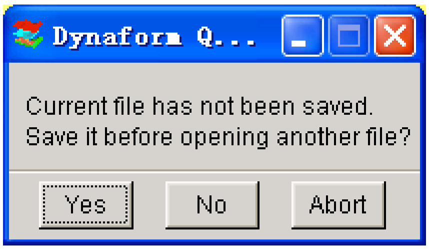


Figure 3.2.1 Open File Prompt

- The user can open a DYNAFORM database by selecting a file name in the open file dialog window. The OPEN DATABASE dialog window is shown in Figure 3.2.2.

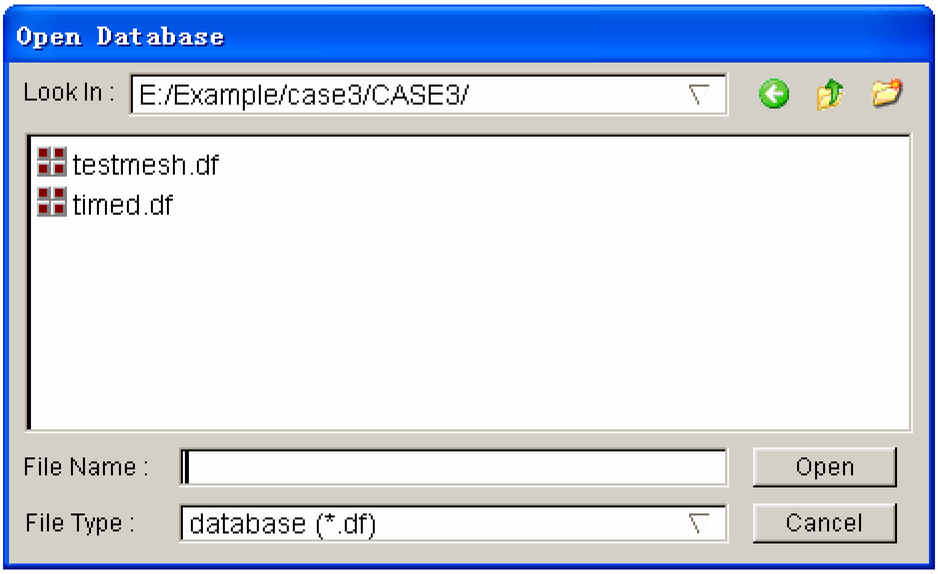


Figure 3.2.2 Open File Prompt

3.3 RESTART

This function allows the user to restart the current database from the previously saved stage. eta/DYNAFORM prompts the user to save the current file. See Figure 3.3.1.

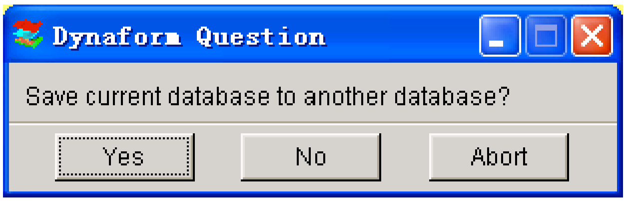


Figure 3.3.1 Restart File Prompt

- 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 updates the current database.

3.5 SAVE AS

This function saves the current database as a new file. See Figure 3.5.1.

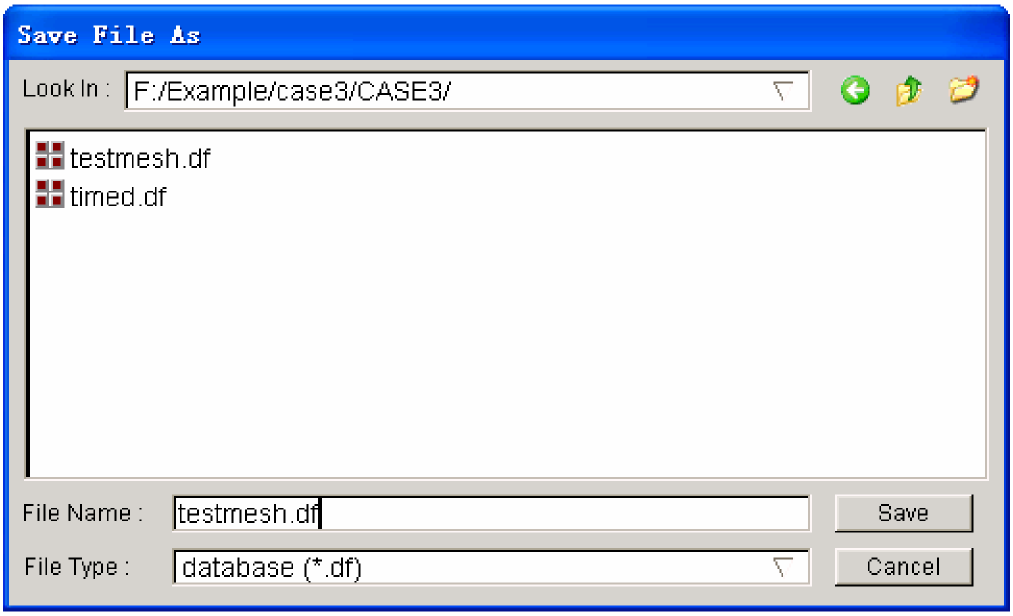


Figure 3.5.1 Save File As

3.6 IMPORT (F3)

This function allows the user to read in CAD or model data. See Figure 3.6.1.

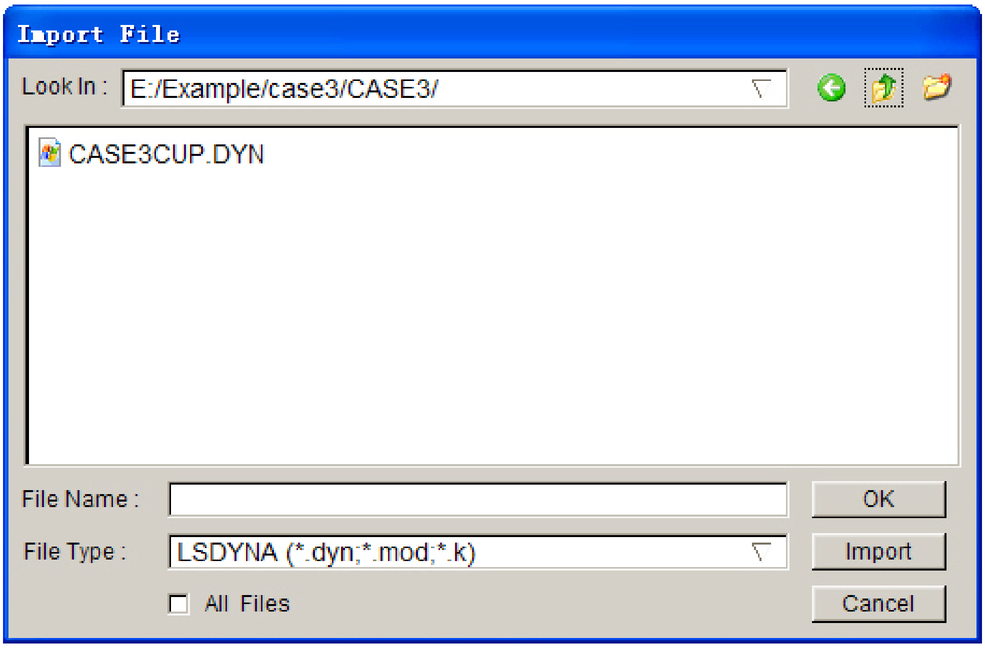


Figure 3.6.1 Import File

FORMAT

Select the downward pointing triangle to open the combo box, and select the proper format. The available formats are:

MODEL DATA FORMAT:	LSDYNA , NASTRAN, and ABAQUS.
CAD DATA FILE:	Line data, IGES, VDA, DXF, STL, ACIS, CATIA, UG, PRO-E, and STEP.
MULTI-STAGE FILE:	DYNAIN.

If ALL FILES (Check Box) is toggled **On**, 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 current eta/DYNAFORM database. The options are similar to the options mentioned above. The formats are:

INDEX

Includes basic information on the model from eta/DYNAFORM, including the unit system, draw type and direction, tools ID and name, and material. This information will be used for post-processing simulation results. The INDEX file will be written automatically when the DYNA input file is written.

NASTRAN

Entire or partial model data.

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

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

3.8 SUBMIT DYNA FROM INPUT DECK

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

1. eta/DYNAFORM prompts the user to select the input file. See Figure 3.8.1.

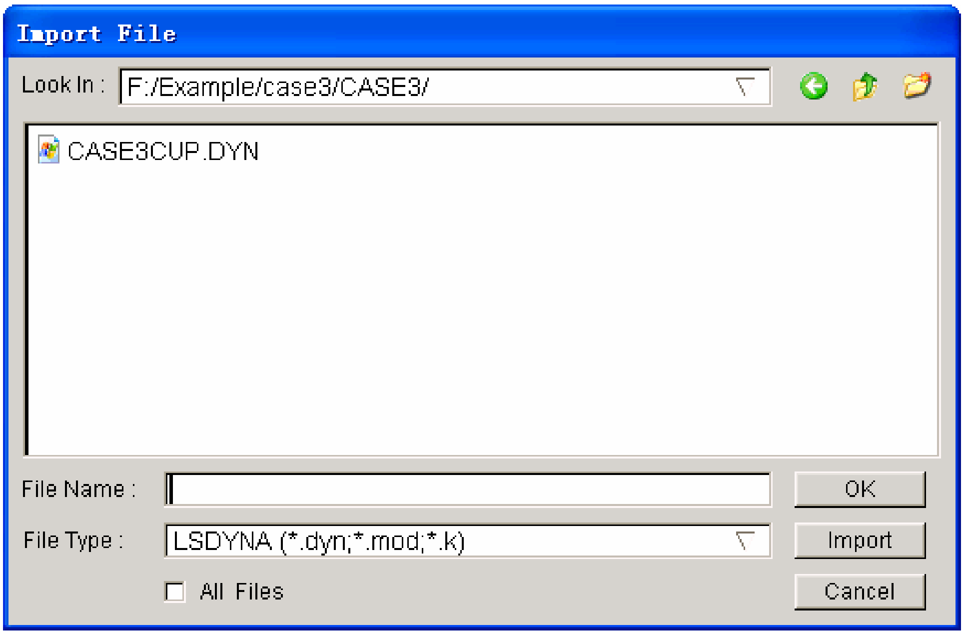


Figure 3.8.1 Select Input File

The user can select the DYNA file or select "All Files" to list all files. Double click the left mouse button on the desired file, or select **Ok** to continue.

2. The SELECT MEMORY window appears. See Figure 3.8.2.
   The default memory is 64MB, but a different memory can be given.

Ok- LSDYNA starts and begins to solve the selected input deck file.
Back- Returns to the SELECT INPUT DECK FILE window.
Cancel- Exits the function.

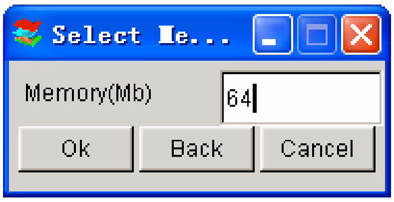


Figure 3.8.2 Select Memory

eta/DYNAFORM locates LSDYNA by the given directory. Refer to Section **8.1, ANALYSIS PARAMETERS,** for more information. If there is no location or the location of LSDYNA is wrong, clicking **Ok** on the SELECT MEMORY window will bring up the SELECT SOLVER window. See Figure 3.8.3.

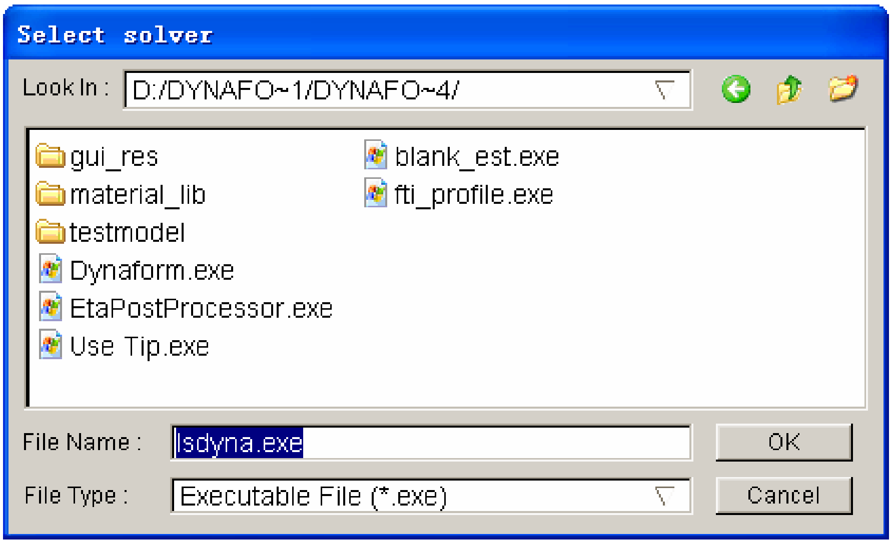


Figure 3.8.3 Select Solver

This option allows the user to locate the LS-DYNA solver.

3.9 RUN DYNA RESTART

This function submits a DYNA restart file to LS-DYNA and starts up LS-DYNA automatically.

1. eta/DYNAFORM prompts the user to select the input file. See Figure 3.9.1.

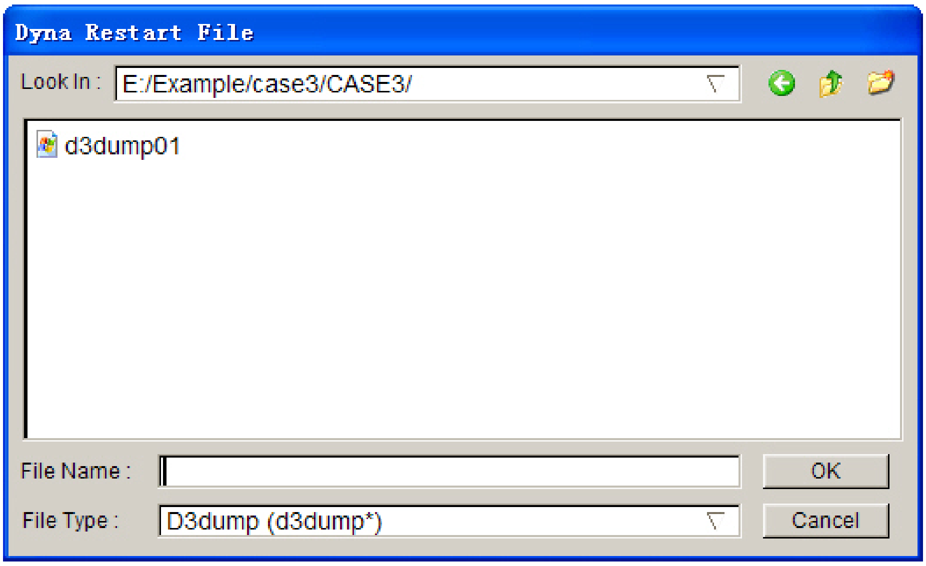


Figure 3.9.1 Select DYNA Restart File

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

Double click the left mouse button on the desired file, or click Ok to continue.

2. The SELECT MEMORY window appears. See Figure 3.8.2.

Note: The user should use the same amount of memory as in the original run.

Refer to Section **3.7**, **SUBMIT DYNA FROM INPUT DECK**, and Section **8.1**, **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.

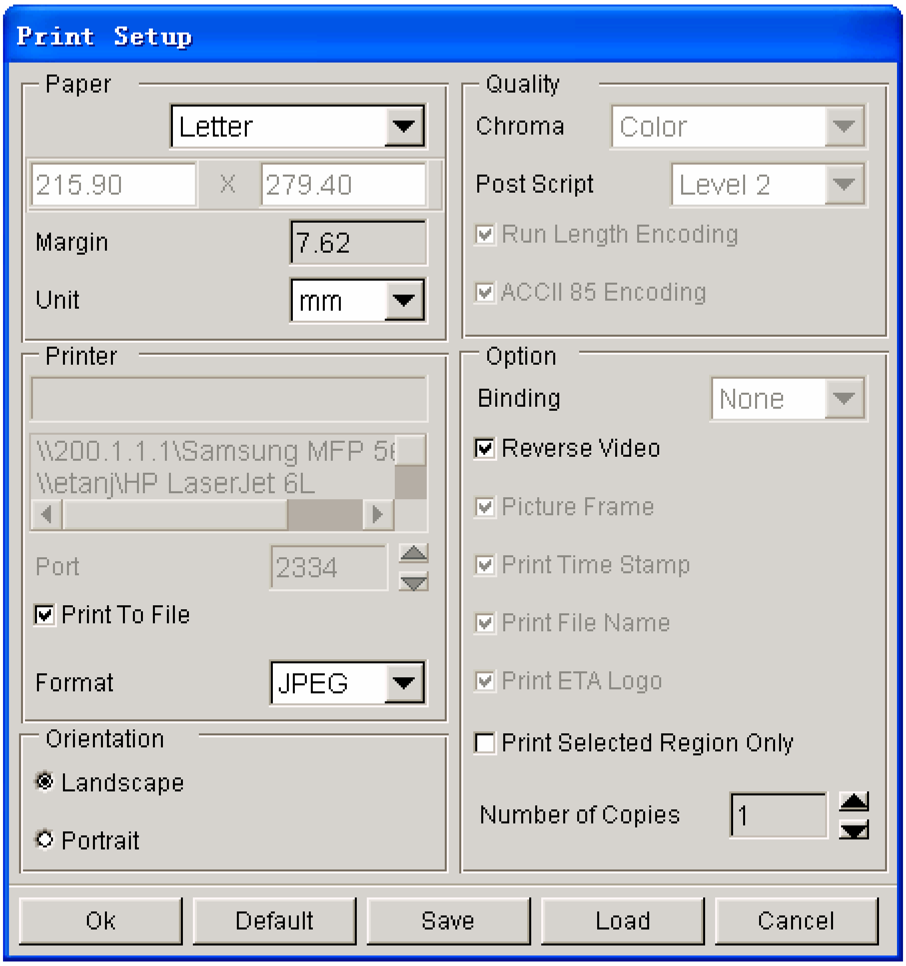


Figure 3.10.1 Print Setup

3.10.1 PAPER

The user can specify a given paper and/or margin size.

- Select the 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.
- Select the drop down choice next to UNIT to choose a unit (inches or mm).

3.10.2 PRINTER

The user can select a printer or a format to print to a file.

- 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

Select the drop down choice next to CHROMA to select the color type.

POSTSCRIPT

There are two levels: level 2 is the default setting, but level 1 should be selected when 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 used for defining printer output and layout on the paper.

BINDING

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

REVERSE VIDEO

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

BOUNDING BOX

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

PRINT STAMP TIME

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

PRINT FILE NAME

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

PRINT ETA LOGO

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

PRINT SELECTED REGION ONLY

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

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

3.10.6 NUMBER OF COPIES

This function allows the user to print multiple copies.

3.10.7 DEFAULT

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

3.10.8 SAVE

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

3.10.9 LOAD

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

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

3.11 PRINT

This 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 in Section 3.10.

3.12 EXIT (CTRL+X)

This function allows the user to exit the program. eta/DYNAFORM prompts to save the current data

CHAPTER 4
PART CONTROL

A part in an eta/DYNAFORM database file is a set of lines, surfaces, and elements. Each part has a unique Part Identification number (PID). The part name is a string consisting of up to eight characters. Presently, the user can create up to 1000 different 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.

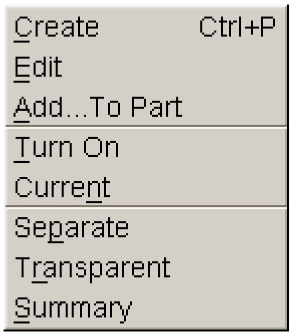


Figure 4.1 Part Control

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

4.1 CREATE PART (CTRL+P)

This function enables the user to make a new part. See Figure 4.1.1

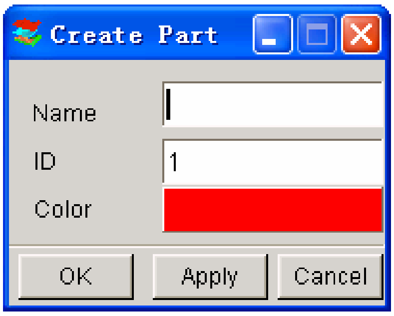


Figure 4.1.1 Create Part

1. Type a new 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. See Figure 4.1.2 for available part colors.

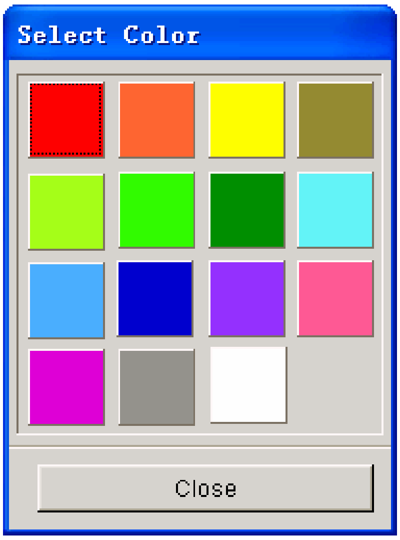


Figure 4.1.2 Select Color

4. Choose either the OK or the APPLY button. Clicking the OK button will create a new part and exit CREATE PART function. Clicking the APPLY button will create a new part and remain in the CREATE PART function allowing the user to continue creating new parts. The new part is also the current part and is shown in the DISPLAY OPTIONS window.

4.2 EDIT PART

The functions in this menu enable the user to modify and delete parts as well as modify part names, ID, and color. See Figure 4.2.1.

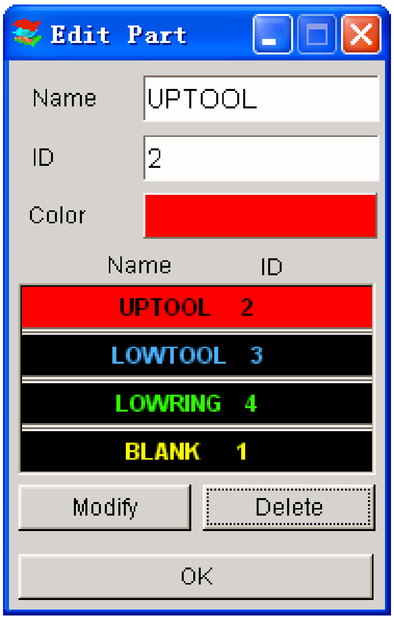


Figure 4.2.1 Edit Part

4.2.1 MODIFY PART

This function allows the user to change the part name, ID, and/or color.

1. Select a part in the part name list.

2. Select MODIFY in order to change the part name, ID, and/or color.

4.2.2 DELETE PART

This function allows the user to remove a selected part. A DYNAFORM QUESTION window will appear to prompt the user to confirm or cancel the deletion.

4.3 ADD … TO PART

This function allows the user to move lines, elements, or surfaces from one part to another. See Figure 4.3.1.

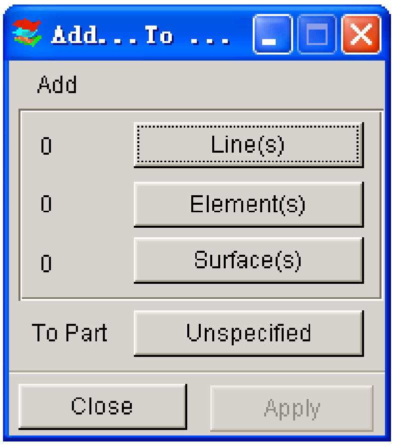


Figure 4.3.1 Add… To Part

4.3.1 ADD LINE

- Select LINE(S), and the SELECT LINE dialog window is displayed. See Figure 4.3.2.

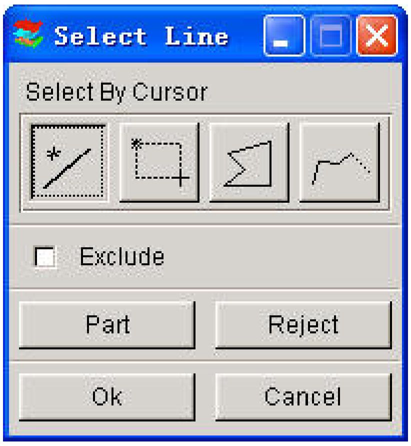
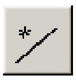
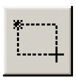
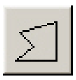
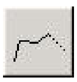


Figure 4.3.2 Select Line

- The SELECT LINE dialog window offers four ways to select a line. The selected line will be highlighted .

1. Select  to pick a line by clicking its location.
2. Select  to pick lines inside a rectangular window. The user can define the rectangle by selecting two points as its diagonal.
3. Select  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  to pick lines by clicking one's location. All lines that are chained (connected) to the selected line will be selected.

5. Select PART to pick lines included in the part.

- Deselect lines.

    1. Toggle ON the EXCLUDE option. The user can exclude the selected lines using the above functions.
    2. Select REJECT, and the last selection will be rejected.

- Select OK or CANCEL.

- To select a target part, select TO PART. See Figure 4.5.1.

- Add a selected line to the part, and then select APPLY.

- To cancel the current adding operation, select CLOSE.

4.3.2 ADD ELEMENT

- Select ELEMENT(S), and the SELECT ELEMENTS dialog window is displayed. See Figure 4.3.3. There are three sets of functions, depending on whether SELECT BY CURSOR or SELECT BY is ON.

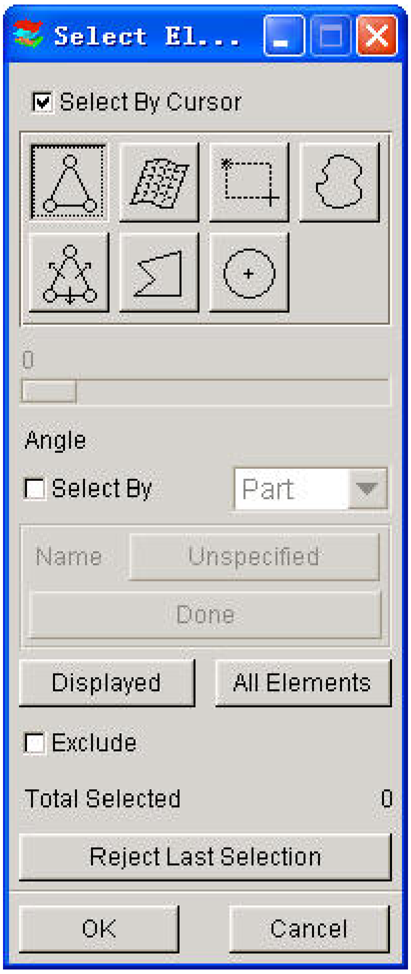
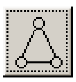
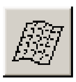
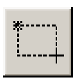


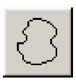
Figure 4.3.3 Select Element

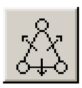
- If SELECT BY CURSOR is toggled ON, the user can select elements in the following four ways.

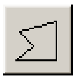
Select  to pick an element by its location to select one element or click left mouse button and move it over the desired elements to select multi-elements.

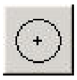
Select  to pick elements on a surface.

Select  to pick elements inside a window.

Select  to pick elements inside a free hand region.

Select  to pick elements by spread. The slider above ANGLE can be used to adjust the angle. This allows the user to select an angle and pick an element. 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.

Select  to pick elements inside a polygon. Click the left mouse button in a sequence to define the polygon. Click right mouse button to reject last point and middle mouse button to complete.

Select  to pick elements inside a circle. Click the left mouse button to define the center of the circle. Hold the mouse button and move the mouse to define the circle. Release the mouse button to complete.

- If SELECT BY is toggled ON, there are three ways to select an element. The user may select by part, type, or range. See Figures 4.3.4 ~ 4.3.6. Click the drop down choice next to the SELECT BY button to choose one of the options.

Figure 4.3.4 Select by Part

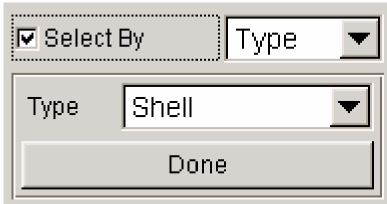


Figure 4.3.5 Select by Type

Figure 4.3.6 Select by Range

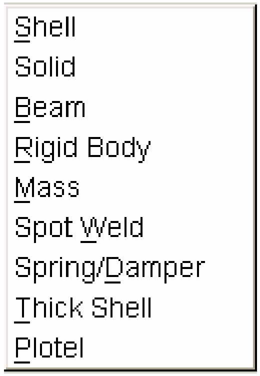


Figure 4.3.7 Element Type

1. SELECT BY PART

   See Section 4.2 to choose a part. All elements in the part will be selected when this option is selected. See Figure 4.3.4.

2. SELECT BY TYPE

   Select a type in the ELEMENT TYPE POP-UP Window. See Figure 4.3.7. All elements of the selected type displayed on the screen will be selected. See Figure 4.3.5.

NOTE: Refer to the LS-DYNA User's Manual for a description of the nine types of elements. Deselecting elements and closing the dialog box operations are similar to those in Section 4.5.1.

3. SELECT BY RANGE

Type the element range in the dialog window. All elements within that range will be selected. See Figure 4.3.6.

- Select DISPLAYED, and all elements in the turned on parts will be selected.

- Select DONE after processing each of the above functions. The total number of selected elements will be shown in the window.

  EXCLUDE and REJECT perform in the same manner as in SELECT LINE.

- Select OK to finish the selection of elements and return to the PART ADD menu.

4.3.3 ADD SURFACE

- Select SURFACE(S), and the SELECT SURFACE dialog window appears. See Figure 4.3.8.

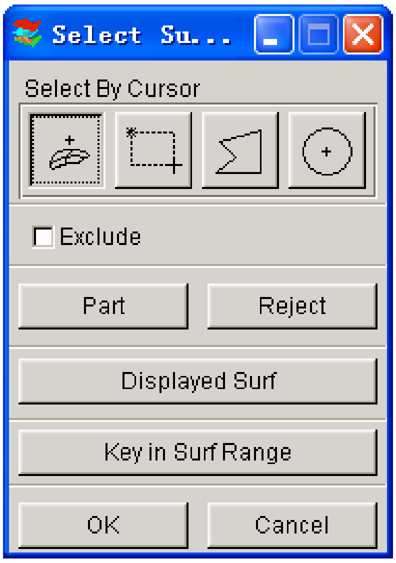


Figure 4.3.8 Select Surface

- **DISPLAYED SURF**

  This function is used to select all displayed surfaces in the graphic window.

- **KEY IN SURF RANGE**

  Select this button and the RANGE OF SURFACE dialog window will display. See Figure 4.3.9. The user can enter values in the displayed fields to select a range of surfaces.

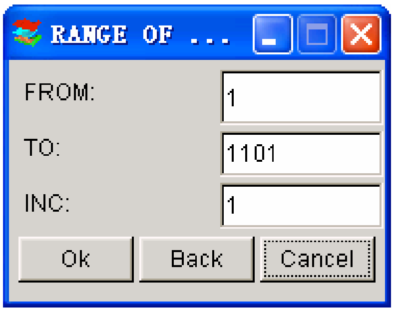


Figure 4.3.9 Define Surface Range

Other functions are the same as those in SELECT LINES.

4.3.4 SELECT THE TARGET PART

After selecting lines, elements, or surfaces, the target parts that will contain those selected entities must be specified.

The program will add the selection to the specified part when APPLY is selected. Select CANCEL to abort the addition to the part.

4.4 TURN ON

This function enables the user to turn the selected parts on or off. The PART TURN ON/OFF dialog window is displayed once the function is selected. See Figure 4.4.1.

- The user can turn off a part by selecting a line, element, or surface. The part can also be toggled on/off by selecting its name in the SELECT BY NAME list.

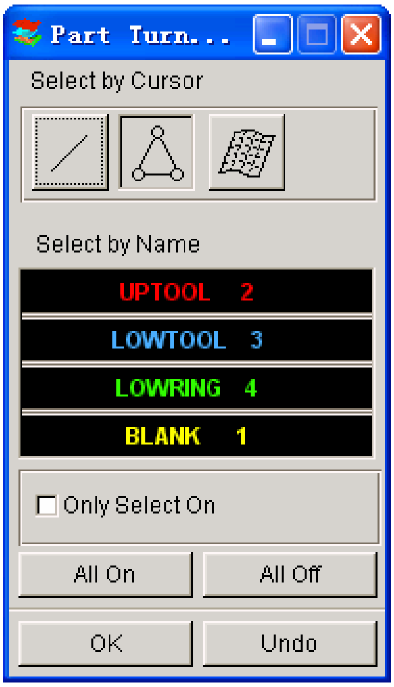


Figure 4.4.1 Turn Part On/Off

- Only one part will be turned on if the ONLY SELECTED ON button is toggled on.

- **ALL ON**

  All parts will be shown.

- **ALL OFF**

  All parts will be turned off.

4.5 CURRENT PART

This function allows the user to change the current part. All lines, surfaces, and elements that the user creates are automatically included in the current part. The name of the current part is displayed in its part color in the lower right corner of the screen in the DISPLAY OPTIONS window, and A SELECT PART dialog window is displayed. The user can also click on this field to change the current part. See Figure 4.5.1.

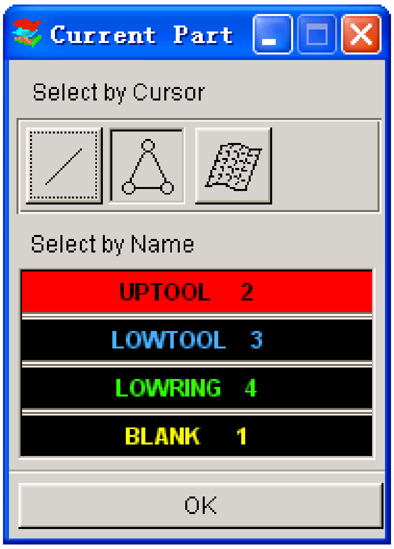


Figure 4.5.1 Current Part

- In Figure 4.5.1, the user can select a part by picking a location, line, element, or surface that is included in the part. The user can also select a part by name.

4.6 SEPARATE PART

This function allows the user to quickly separate parts having common nodes. Once the parts are separated, each common node becomes several nodes, one for each part, at the same location.

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

4.7 TRANSPARENT

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

4.8 SUMMARY

This function enables the user to display statistics regarding the geometry, material, and interface information of the selected parts. Figure 4.8.1 is an example of a part summary table.

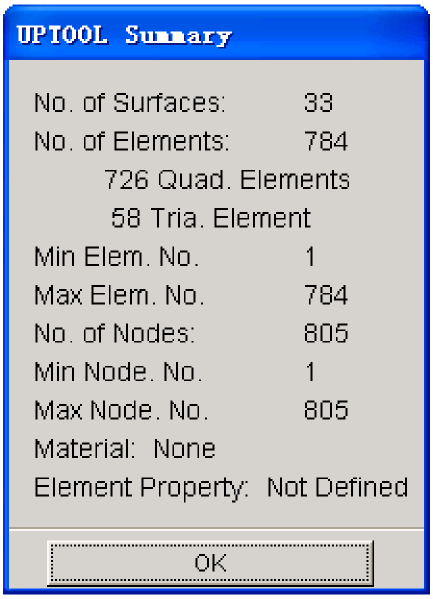


Figure 4.8.1 Part Summary

CHAPTER 5
PREPROCESS

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

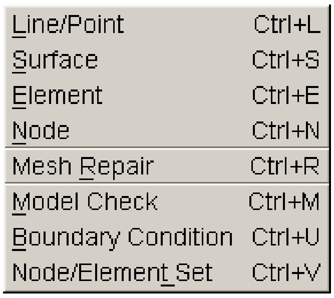


Figure 5.1 Preprocess menu

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

Many dialog and submenu windows work in the same or similar manner as windows that have appeared in previous chapters. References to 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.

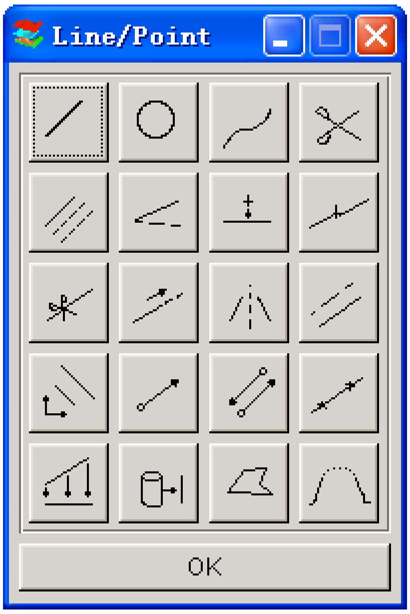
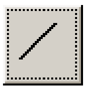


Figure 5.1.1 Line/Point Options

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

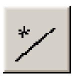
5.1.1 CREATE LINE 

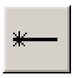
This function allows the user to draw a line. In eta/DYNAFORM, a line is formed by a sequence of points and is displayed by the set of straight-line segments between adjacent points in the sequence. To create a line, the user needs to define the location of the points. The INPUT COORDINATE dialog window, Figure 5.1.2, provides various ways to define the location of the points.

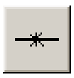
1. To create a point of the line at an existing node or point.

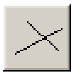
   - Select NODE or POINT option.

   - Select the location of the node or point. If the POINT option is selected, the following buttons will determine the creation of the point.

      At a point of an existing line near the cursor.

      At an endpoint of a line.

      At the midpoint of a line.

 At the intersection of other lines.

2. To create a point by its coordinates.

- Select the LCS or ABS option for the coordinates in the current local coordinate system or global system. See Section 2.5, LOCAL COORDINATE SYSTEM.

- Select the XYZ or DXYZ option.

  If XYZ is selected, the typed values of U, V and W are the coordinates.

  If DXYZ is selected, the typed values of DU, DV and DW are increments of the previously created point.

- Type the three values in the corresponding field.

- Select APPLY INPUT VALUE to confirm the values and create the point.

3. To cancel the last created point

- Select REJECT once.

4. To finish the definition of the line

- Select OK or click on the middle mouse button.

5. To close the dialog window

- Select CANCEL or click on the right mouse button. The selected points will be ignored when the CANCEL button is selected. It is the equivalent of ABORT.

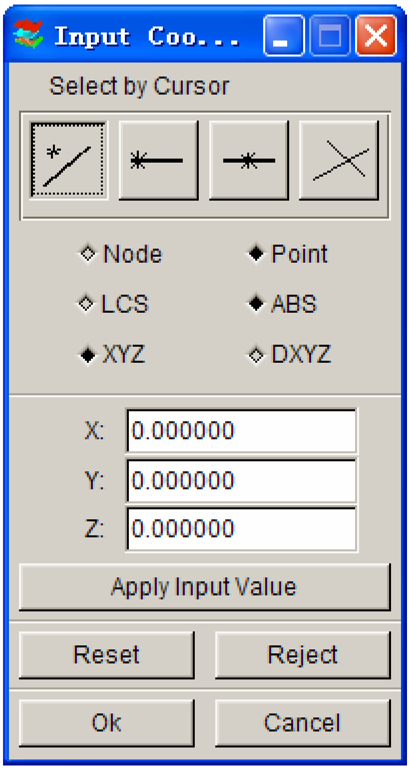
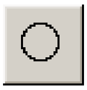


Figure 5.1.2 Input Coordinate

5.1.2 CREATE ARC 

This function allows the user to generate circular arcs. There are three types:

1. CENTER AND RADIUS

- Define a LCS. See Section 2.5, LOCAL COORDINATE SYSTEM. The center of the arc will be at the origin, and the arc will be in the UV plane of the LCS.

- Enter the values in the ARC PARAMETER dialog window. See Figure 5.1.3.

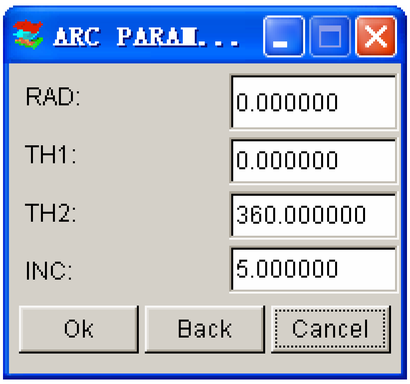


Figure 5.1.3 Arc Parameters

RAD: Radius of arc

TH1: Beginning angle from U-axis

TH2: Ending angle from U-axis

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

2. TANGENT TO 2L

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

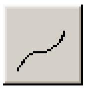
   - Select two intersecting lines. See Section 4.5.1, PART/ADD LINE, for more information on selecting lines.

   - Enter the radius in the RADIUS OF ARC field.

3. Create by THROUGH 3 PTS.

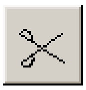
   This function allows the user to create an arc through three non-collinear points/nodes.

   - Select three points in Figure 5.1.2, and refer to Section 5.1.1. When the third point is selected, the arc and its center will be generated.

5.1.3 CREATE SPLINE 

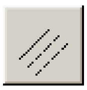
This function allows the user to draw a spline curve through multiple points, nodes or through any combination of points or nodes. A minimum of three points or nodes is required.

The user selects the points or nodes in the INPUT COORDINATE dialog window. Selecting DONE will generate the spline line.

5.1.4 DELETE LINE 

This function allows the user to delete lines.

- Select lines to delete. See Section 4.3.1, PART/ADD LINE, for more information.

5.1.5 COPY OR TRANSFORM LINE 

This function allows the user to generate a set of duplicate lines. The user can simultaneously translate and rotate the copied lines. The user must complete the options in the COPY LINES dialog window. See Figure 5.1.4.

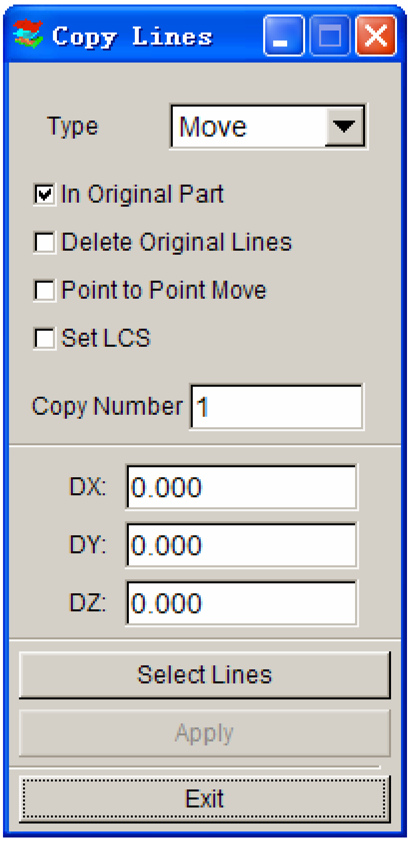


Figure 5.1.4 Copy/Transform Line

1. Select lines. See Section 4.3.1, PART/ADD LINE, for more information.

2. Choose a type of transformation, MOVE or ROTATE.

3. Enter the number of copies.

4. Select or deselect IN ORIGINAL PART. If IN ORIGINAL PART is not selected, the transformed lines will be in the current part.

5. Select DELETE ORIGINAL LINES to delete the line(s) after the transformation.

6. Select or deselect POINT TO POINT MOVE. POINT TO POINT MOVE is discussed later in the chapter. This option is only active for the translation.

7. Select or deselect SET LCS. SET LCS is discussed later in the chapter.

8. Type in the increments DX, DY, DZ for the translation or type in the angle increment DX for rotation about the W-axis.

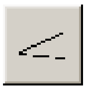
9. Select APPLY.

- If POINT TO POINT MOVE is selected, the INPUT COORDINATE window, Figure 5.1.2, will be displayed for selecting two points. The translation will be based on the differences of three coordinates of selected points.

- If SET LCS is selected, the LCS dialog window will be displayed to define a new system. See Section 2.5, LOCAL COORDINATE SYSTEM. The transformation will be done under the defined system

- If SET LCS is not selected, the LCS window will not be shown. The transformation will be done under the current coordinate system.

10. Select OK to close the COPY LINE dialog window.

5.1.6 MODIFY LINE 

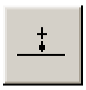
This function allows the user to modify a line by moving its points.

- Select a line in the window. See Figure 4.3.2 of Section 4.3.1, PART/ADD, for more information.

- Select a point on the line to modify.

- Input the new location for the selected point in the INPUT COORDINATE dialog window.

- Select DONE in the INPUT COORDINATE dialog window.

5.1.7 ADD POINT 

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

There are two ways to add a point:

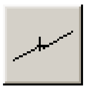
1. BETWEEN 2 POINTS
   This option creates a specified number of new points between two selected endpoints.

   - Select two points by cursor on a line.

   - Enter the desired number of points in the next dialog window.

2. CURSOR LOCATION

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

5.1.8 COMBINE LINE 

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

- Select Combine Line function the program will pop-up a dialog box for user to select the gap value between lines as Figure 5.1.5. If the user doesn't toggle on the line gap option, all select lines will be combined.

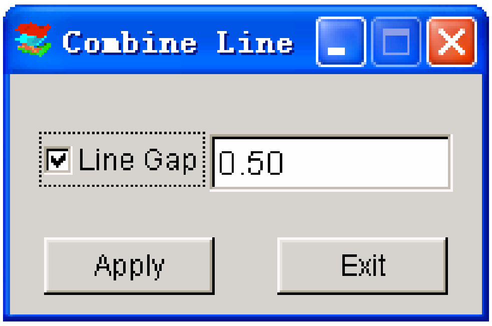
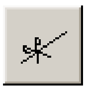


Figure 5.1.5 Combine Line

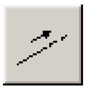
- Select Apply to select lines and click OK to combine lines.
- The user can click Apply to continue this function or click Exit to quit this function.

5.1.9 SPLIT LINE 

The user can split a line into two lines. The first dialog window is used for selecting a line; the second dialog window is used for the split location.

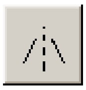
1. Select a line using one of the functions in the SELECT LINE dialog. See Section 4.3.1, PART/ADD LINE, for more information.

2. Select the location on which to split the line using one of the functions of the INPUT COORDINATE dialog. See CREATE LINE, Section 5.1.1, for more information.

5.1.10 EXTEND LINE 

This function allows the user to extend a line by a specified distance from the selected end of the line.

- Select a line by mouse pick.
- Select the end of the line to extend.
- Enter a real number for the extended length in the next dialog window.

5.1.11 MIRROR LINE 

This function allows the user to mirror a set of lines with respect to a given plane. The MIRROR LINES dialog window appears. See Figure 5.1.6.

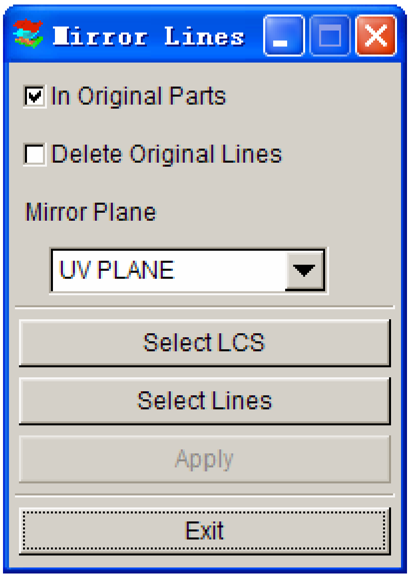
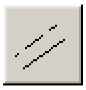


Figure 5.16 Mirror Line

- Select line(s). See Section 4.3.1, PART/ADD LINE, for more information.
- Select or deselect the option boxes.
  IN ORIGINAL PARTS
  The created lines are put in the original part if the box is toggled on. If the box is not toggled on, the created lines are put in the current part.

  DELETE ORIGINAL LINES
  The original lines are deleted if the box is toggled on..

- Select LCS. See Section 2.5, LOCAL COORDINATE SYSTEM, for more information.
- Select a coordinate plane as the mirror plane.
- Select APPLY.

5.1.12 OFFSET LINE 

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

- Select a line in the display window.
- Define a coordinate system to determine the U-V plane.
- Enter a positive number in the next window for offsetting distance.
- The DYNAFORM Question window is displayed. See Figure 5.1.7. There is an arrow in the display window indicating the offset direction.

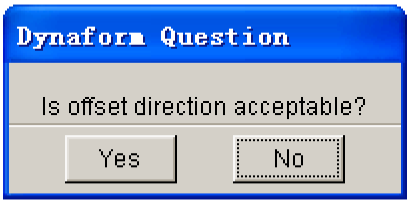
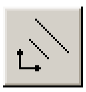


Figure 5.1.7 eta/DYNAFORM Question window

Select YES to accept the direction.

Select NO to turn to the opposite direction.

5.1.13 SCALE LINE 

This function allows the user to scale the selected lines.

- Select a line. See Section 4.3.1, PART/ADD LINE, for more information.
- Define a coordinate system for scaling. See Section 2.5, LOCAL COORDINATE SYSTEM, for more information.

- Enter the desired scale factors in the fields in the SCALE FACTOR dialog window. See Figure 5.1.8.

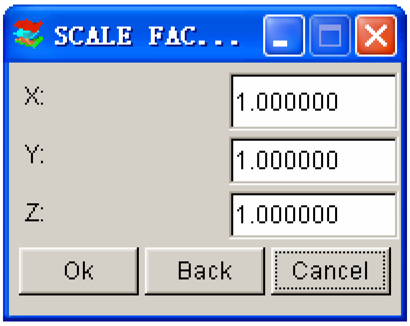
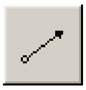


Figure 5.1.8 Scale Factor

5.1.14 SHOW LINE 

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

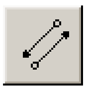
There are two ways to show a line:

1. LINE

   Select line by cursor.

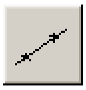
2. KEY IN LINE NO.

   Enter the line number in the dialog window; then select OK.

5.1.15 REVERSE LINE DIRECTION 

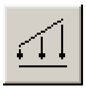
Each line has a direction defined from the starting point to the endpoint of the line. This function allows the user to reverse the line direction.

A dialog window is displayed, and the user can select lines by cursor. Once the lines are selected, the direction of the selected lines is automatically reversed.

5.1.16 RESPACE LINE 

This function allows the user to redistribute the points on the selected line.

- Select the line by cursor.
- Enter any integer greater than two in the next dialog window to define the number of points on the line.

5.1.17 PROJECT LINE 

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

- Select a surface. See Section 4.3.2, PART/ADD ELEMENT, for more information.
- The user must define a coordinate system for the projected line. The projected direction is along the W-axis.

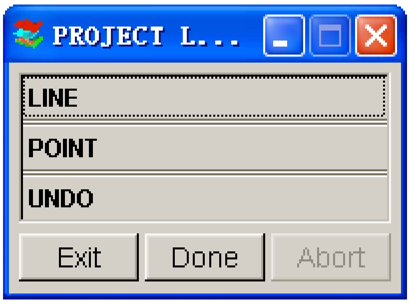
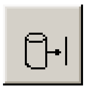


Figure 5.1.9 Project Line

- PROJECT LINE/POINT is displayed. See Figure 5.1.9.
- The selected lines or points will be projected on the selected line along the given W-axis. If there is no projection found for some lines or points, an error message will be given in the prompt area.

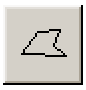
NO PROJECTION FOUND FOR XX POINTS

XX indicates the number of points that have no projection.

5.1.18 SECTION THROUGH LINE 

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

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

5.1.19 F E. BOUNDARY LINE 

This function allows the user to generate a line around the boundary of a finite element mesh. See Figure 5.1.10.

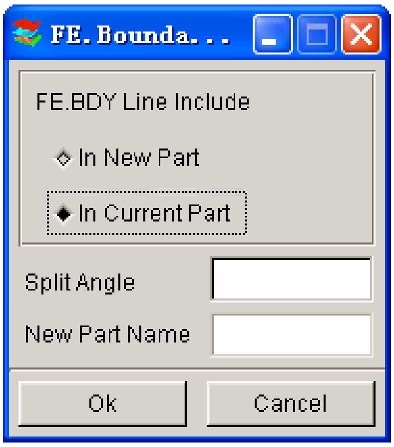


Figure 5.1.10 FE.Boundary Line

IN NEW PART

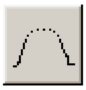
This option includes the boundary line in a new part. The new name is entered in the field next to NEW PART NAME. If IN NEW PART is toggled off, the NEW PART NAME field is disabled.

IN CURRENT PART

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

SPLIT ANGLE

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

5.1.20 BRIDGE LINE 

This function generates a 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 window as shown in Figure 5.1.11 and prompt the user to select two lines.

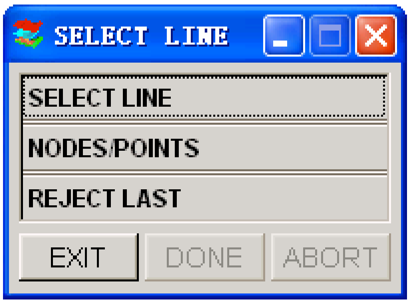


Figure 5.1.11 Select Line dialog window

There are three options in the dialog window.

SELECT LINE

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

NODES/POINTS

This option allows the user to define the line from selecting nodes or points on the screen. The program will display an INPUT COORDINATES dialog window 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 5.1.12 shows the original lines and the generated bridge line.

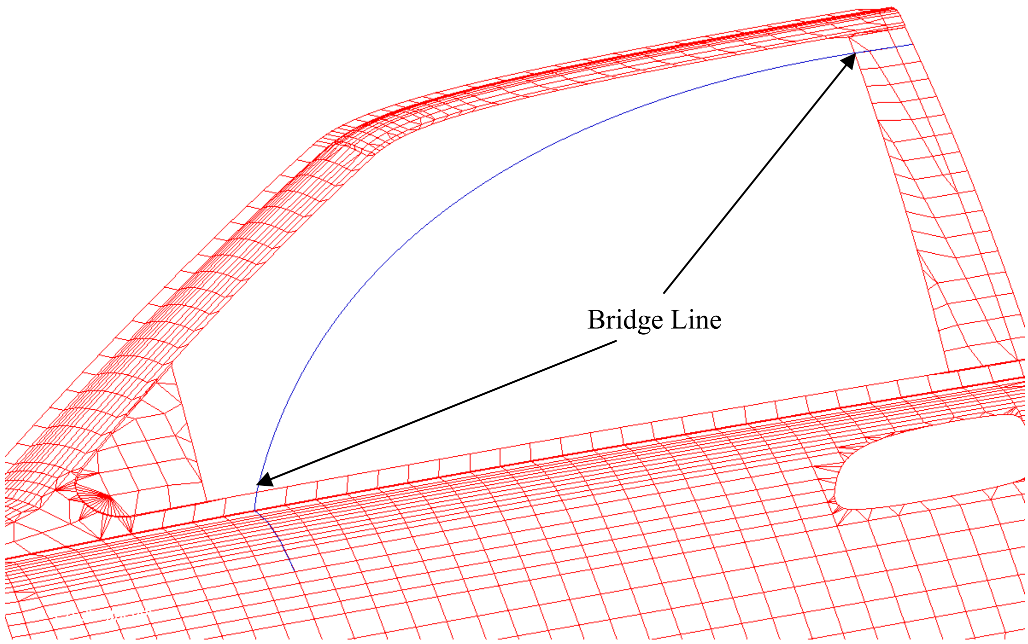


Figure 5.1.12 Bridge Line

5.2 SURFACE (CTRL+S)

The functions in this menu allow the user to create and modify surfaces in eta/DYNAFORM. The options are shown in Figure 5.2.

Figure 5.2 Surface menu

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

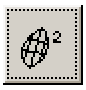
- **U-V LINE**

  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 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 submenu window display. See Figure 5.2.1.

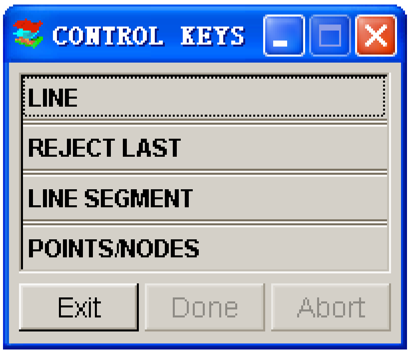
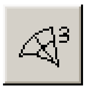


Figure 5.2.1 Control Keys

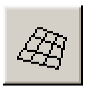
- LINE allows the user to select lines (default).

- LINE SEGMENT - This option allows the user to select several line segments to form a line. After the line segments are complete, DONE is selected.

- 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.

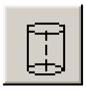
5.2.2 CREATE 3L 

This function allows the user to create a surface from three selected lines. The procedure is similar to CREATE 2L in SURFACE.

5.2.3 CREATE 4L 

This function allows the user to create a surface from four lines. The procedure is similar to CREATE 2L.

Note: Select a line in a clockwise or counterclockwise direction to define the surface normal direction. The direction of surface normal dictates the direction of 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 window. 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 window. See Figure 5.2.2. Select OK, and the surface will be generated after the window is closed.

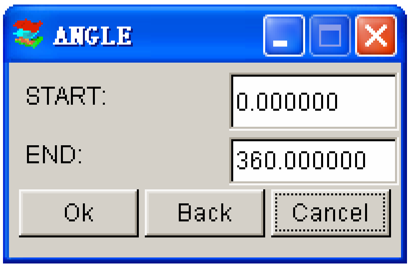
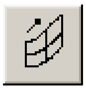


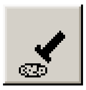
Figure 5.2.2 Revolution Angle

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:
  1. NORMAL SWEEP
     The section line rotates along the normal direction of the direction line while sweeping.
  2. RIGID SWEEP
     The section line does not rotate while sweeping.

- The procedure of selecting lines is similar to 5.2.1.

5.2.6 SHOW SURFACE 

This function allows the user to highlight the selected surface and UV lines. There are several ways to show surface. See Figure 5.2.3.

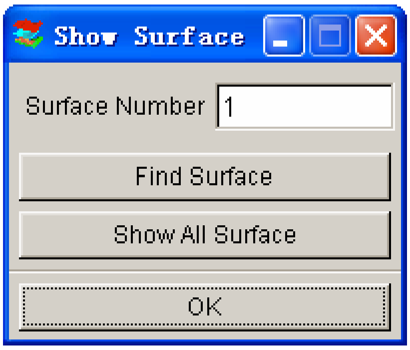
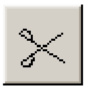


Figure 5.2.3 Show Surface

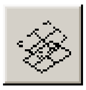
1. Enter the surface number in the field; then select FIND SURFACE. The surface ID number will be shown.

2. Select SHOW ALL SURFACES. The surface ID of the displayed surfaces will be shown.

3. Select the surface in the graphics directly (default).

5.2.7 DELETE SURFACE 

This function allows the user to delete the selected surface from the database. See Section 4.3.3, PART/ADD SURFACE, for selecting surfaces.

5.2.8 TRANSFORM SURFACE 

This function allows the user to change the location of the selected surfaces with the MOVE or ROTATE operations.

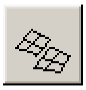
- Define a new coordinate system by using the options in LCS dialog window.
- Select MOVE or ROTATE.

  If MOVE is selected, the user can enter increments of U, V and W in the next dialog window

  If ROTATE is selected, the user can enter increments of the angle in the next dialog window. This will rotate the surface about the W-axis.

- Select the desired surface by using the options in the SELECT SURFACE dialog window.

- The user also has the options to select AGAIN to repeat, REVERSE OPERATION to reverse the process, or SELECT SURFACE to select another surface.

5.2.9 COPY SURFACE 

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

- Select a surface by using options in the SELECT SURFACE dialog window.

- Define a new coordinate system by using options in the LCS dialog window.

- Enter the number of copies along the copy direction.

- Select either MOVE or ROTATE.

- Enter the increments of U, V, W or angle.

- Select YES or NO in the DYNAFORM Question window to determine where the surfaces are included. See Figure 5.2.4.

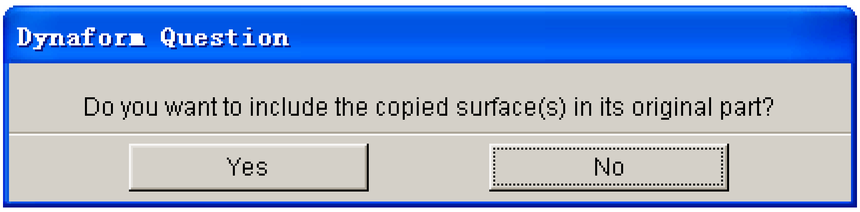
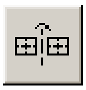


Figure 5.2.4 Copy Question

5.2.10 MIRROR SURFACE 

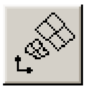
This function allows the user to mirror selected surfaces with respect to a given plane.

- Select surface(s) by using the options in the SELECT SURFACE dialog window.

- Define a local coordinate system.

- Select a coordinate plane as the mirror plane.

- Select APPLY and a DYNAFORM Question window appears. Select YES/NO to mirror surfaces to the ORIGINAL/CURRENT part.

5.2.11 SCALE SURFACE 

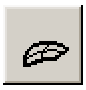
This function allows the user to scale selected surfaces by user-defined scale factors.

- Select a surface by using the options in the SELECT SURFACE dialog window.

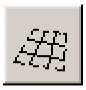
- Define a local coordinate system.

- Enter a scale factor in the direction of U, V and W.

- After the surface is scaled, the user also has the option to select AGAIN to repeat, REVERSE OPERATION to reverse the process, or SELECT SURFACE to select a surface.

5.2.12 CREATE BOUNDARY LINE 

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

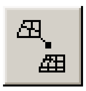
5.2.13 CREATE SECTION LINE 

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

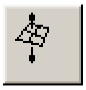
- Select the surface.

- eta/DYNAFORM places edge numbers on the surface and prompts the user to enter the numbers of the section lines along edges 1 and 2 in the dialog window.

- eta/DYNAFORM creates a set of section lines along edges 1 and 2. These lines are included in the current part.

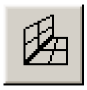
5.2.14 RESPACE U-V LINE 

This function allows the user to change the density of U-V lines in a selected surface. The procedure is similar to CREATE SECTION LINE.

5.2.15 REVERSE NORMAL 

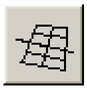
This function allows the user to reverse the normal direction of the surface. The surface normal affects light source shading on some workstations. REVERSE NORMAL also controls the normal direction of the plate elements during meshing.

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

5.2.16 SURFACE INTERSECT 

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

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

5.2.17 SURFACE SPLIT 

This function is used to split a selected surface into two surfaces at a defined line.

After the user selects a surface, there are six options in the submenu window to define the split line.

1. TWO BOUNDARY POINTS

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

2. SECTION U-V LINE

   The user can select a U-V line on the selected surface by cursor. The surface will split at this U-V line.

3. SURFACE INTERSECTION

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

4. SPLINE CURVE

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

5. LINEAR SEGMENT

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

6. RESPACE U-V LINES

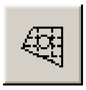
   The user is prompted to enter two values to define a new set of U-V lines.

7. CURVE IN DATABASE

   The function allows the user to pick up the existing curve in database to define the split line.

8. OVERLAP SURFACE

   The function allows user to pick up a surface overlapping the selected surface and smaller than the selected surface. The first selected surface will split along the second surface's boundary.

5.2.18 TRIM SURFACE 

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

1. Select SELECT SURFACE to copy.

2. Select an existing closed curve as the trim line.

3. If there is no existing line, choose EXIT, and select DEFINE CURVE.

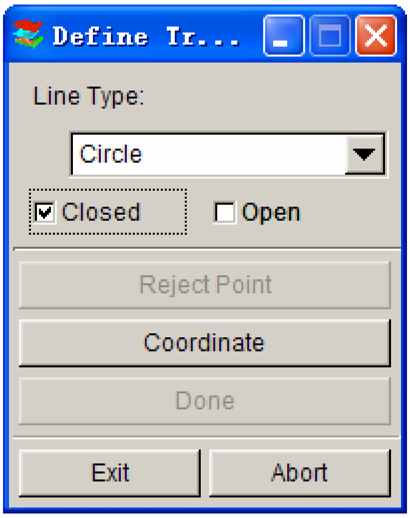


Figure 5.2.5 Define Trim Line

4. DEFINE TRIM LINE is displayed. See Figure 5.2.5

   - 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 a 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 an INTERSECTION LINE, no point needs to be selected. The user selects another surface to define the intersection line.

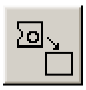
  During the point selection, the user can select REJECT POINT to deselect the last selected point.

- After the curve is defined, select DONE

5. After the curve is defined or selected, select EXIT. The window, Figure 5.2.6, appears.

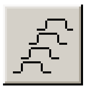
Figure 5.2.6 DYNAFORM Question window

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

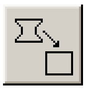
5.2.19 REMOVE HOLE 

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

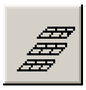
1. Select the REMOVE HOLES icon from the PREPROCESS/SURFACE menu.
2. Select the surface to be altered
3. Select the type of hole, either outer boundary hole or inner hole
4. Select a method for removing the holes, either individually or all at once. NOTE: For outer boundary holes, you must define two points, one on either side. DYNAFORM will then fill in the hole.

5.2.20 SKIN SURFACE 

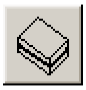
This function allows the user to create a skin surface from a selected series of section lines. The direction of the lines must be consistent. The order of the lines must be selected from one end to the other.

5.2.21 UNTRIM SURFACE 

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

5.2.22 DUPLICATE SURFACE CHECK 

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

5.2.23 GENERATE MIDDLE SURFACE 

This function automatically generates the middle surfaces of sheet-metal parts.

The following example creates middle surface of different thickness. Once this function is selected, the SELECT SURFACE window appears and enables the user to select a seed surface at the part's thickness. See Figure 5.6.7.

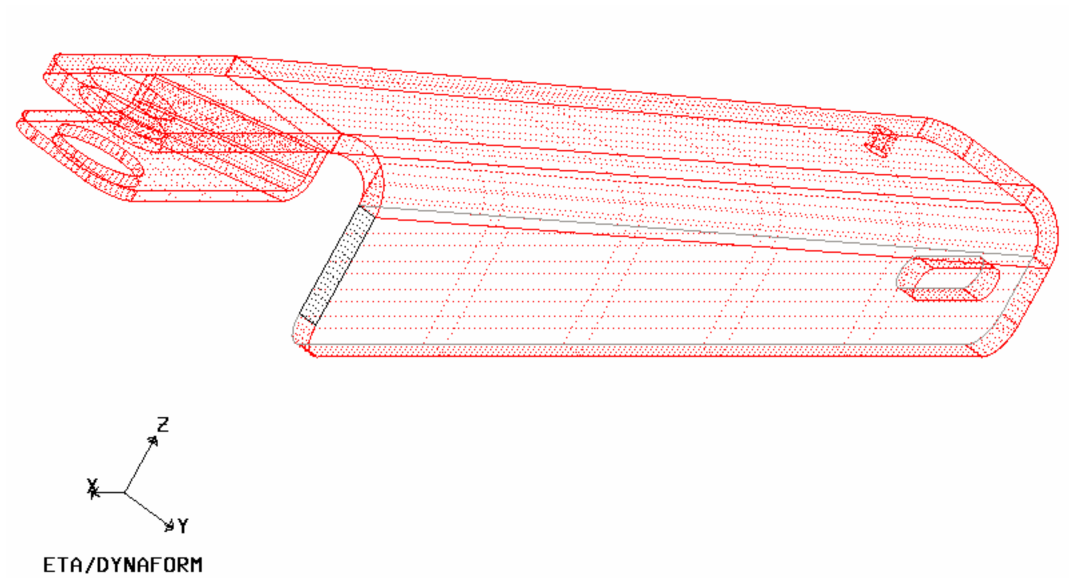


Figure 5.2.7 Select Seed Thickness Surface

1. After selecting the thickness surface, the THICKNESS window appears with the thickness calculated from the select seed surface. The data field allows the user to enter an average of the thickness of the part. After accepting the given thickness value, the program will automatically group the surfaces through the thickness and highlight the corresponding surfaces. Refer to Figure 5.2.8. At the same time, the OPTION window appears (Figure 5.2.9).

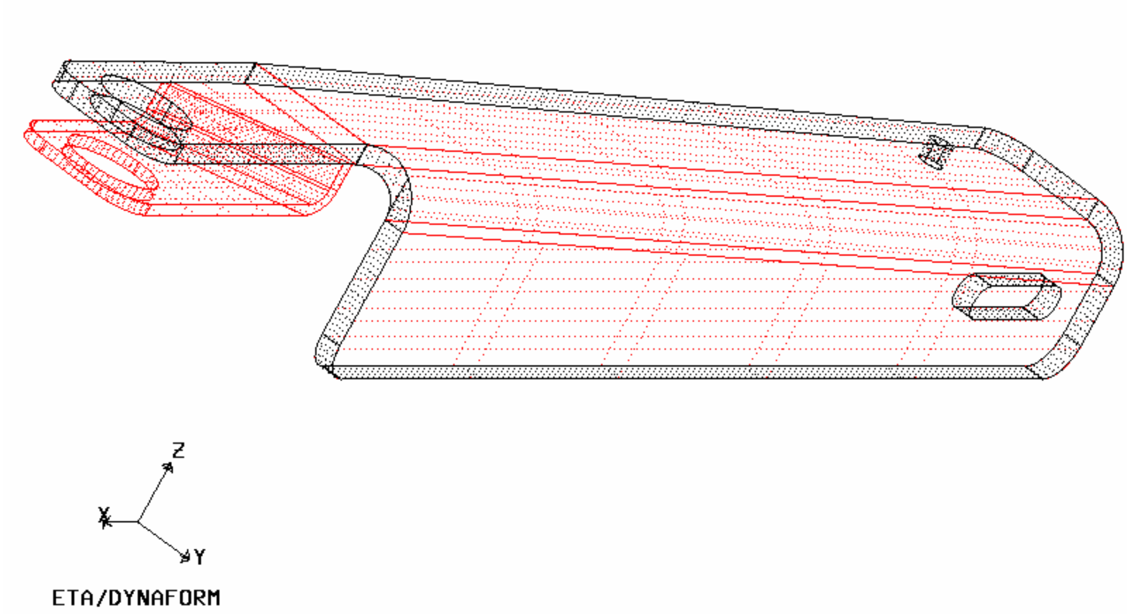


Figure 5.2.8 Thickness Surfaces

Figure 5.2.9 Select Options

2. ADD THICK SURFACES and REMOVE THICK SURFACES are used to add or remove surfaces from the thickness group if the program does not produce perfect grouping. In this example, it is necessary to add some surfaces. See 5.2.10.

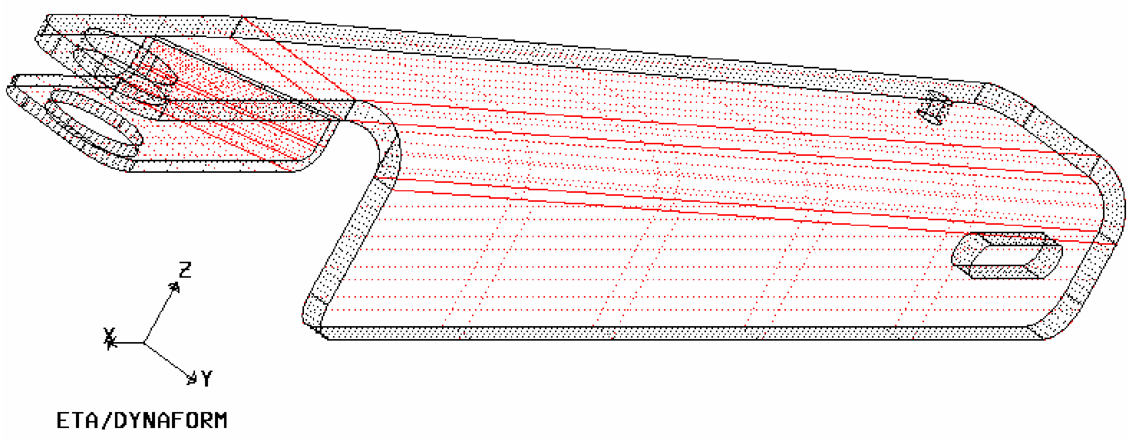


Figure 5.2.10 Add Thick Surfaces

3. After the thickness grouping is done, click GROUP TOP SURFACE (Figure 5.2.9) to group the top surfaces of the part (Figure 5.2.11). The grouped surfaces are highlighted and the SELECT OPTION window appears (Figure 5.2.12).

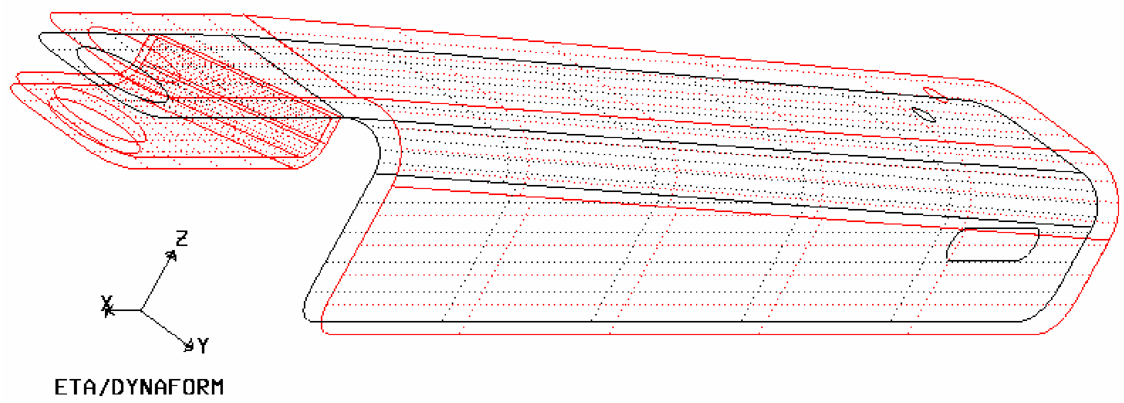


Figure 5.2.11 Grouped Top Surfaces

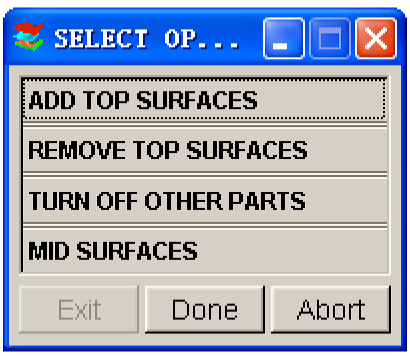


Figure 5.2.12 Select Options

4. The program groups the surfaces for the top faces. If the result is not perfect, Select ADD TOP SURFACE or REMOVE TOP SURFACE to adjust the result. In this instance, three surfaces are added to the top surface group. The result is seen in Figure 5.2.13.

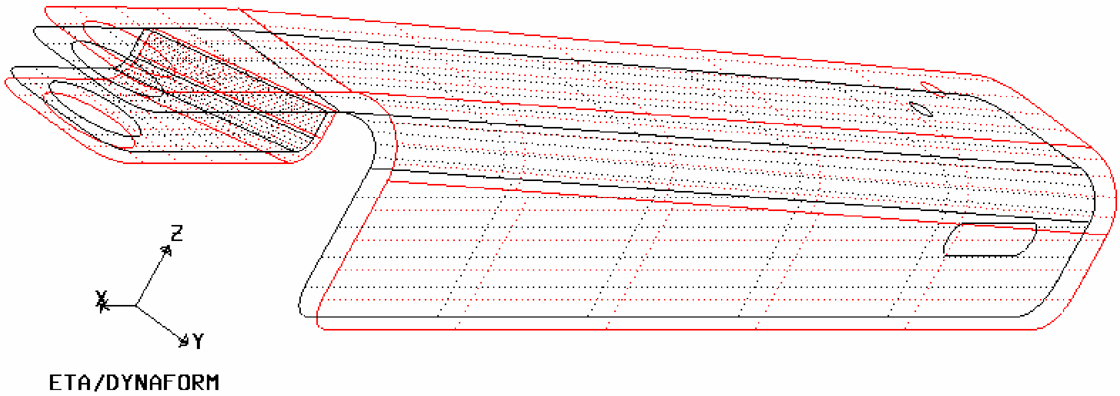


Figure 5.2.13 Grouped and Added Top Surfaces

5. Click MID SURFACE in Figure 5.2.12 to generate the middle surfaces of the part.
6. Middle surfaces of the part are generated from the offset of the top surfaces. They are included in a new part MIDSURF1. Refer to the yellow part in Figure 5.2.14.

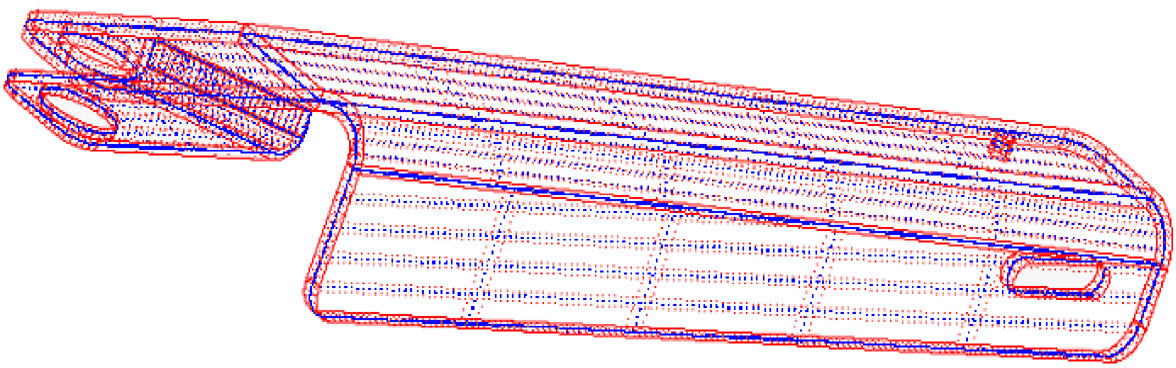
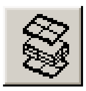


Figure 5.2.14 Middle Surfaces of the Part

5.2.24 GROUP SURFACE 

This function is used to group the top and bottom surfaces of sheet metal parts.

The GROUP SURFACE procedure is similar to the GENERATE MIDDLE SURFACE function. The following steps are used to group top and bottom surface.

1. Once the function is selected, the SELECT SURFACE window appears to prompt the user to select a seed surface for the part thickness. Select a surface at the part thickness. Refer to Figure 5.2.7.

2. After selecting the thickness surface, the THICKNESS window appears with the thickness calculated from the select seed surface. The data field allows the user to enter the average thickness of the part.

3. After accepting the given thickness value, the program will automatically group the surfaces through the thickness, and the corresponding surfaces will be highlighted. Refer to Figure 5.2.8. At the same time, the OPTION window, Figure 5.2.9, is displayed.

4. ADD THICK SURFACE and REMOVE THICK SURFACES functions are used to add or remove surfaces from the thickness group in case the program does not produce prefect grouping. In this example it is necessary to add surfaces as in Figure 5.2.10.

5. After the thickness grouping is done, click GROUP TOP SURFACE, Figure 5.2.9. The program will group the top surfaces of the part. Refer to Figure 5.2.11. The grouped surfaces are highlighted, and at the same time the SELECT OPTION window, Figure 5.2.15, is displayed.

Figure 5.2.15 Group Options

6. This program groups the surfaces for the top face. If the result is not perfect, use ADD TOP SURFACE or REMOVE TOP SURFACE to adjust the result. In this case, three surfaces must be added to the top surface group. The result is shown in Figure 5.2.13

7. Click GROUP BOTTOM SURFACE to group the bottom surfaces of the part (Figure 5.2.15). At this point, Figure 5.2.16 and Figure 5.2.17 are displayed.

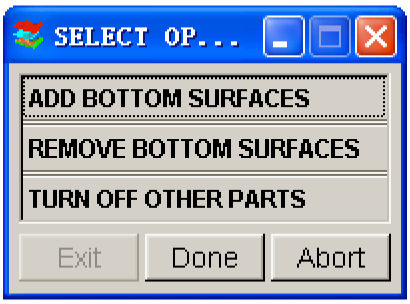


Figure 5.2.16 Group Bottom Surfaces Options

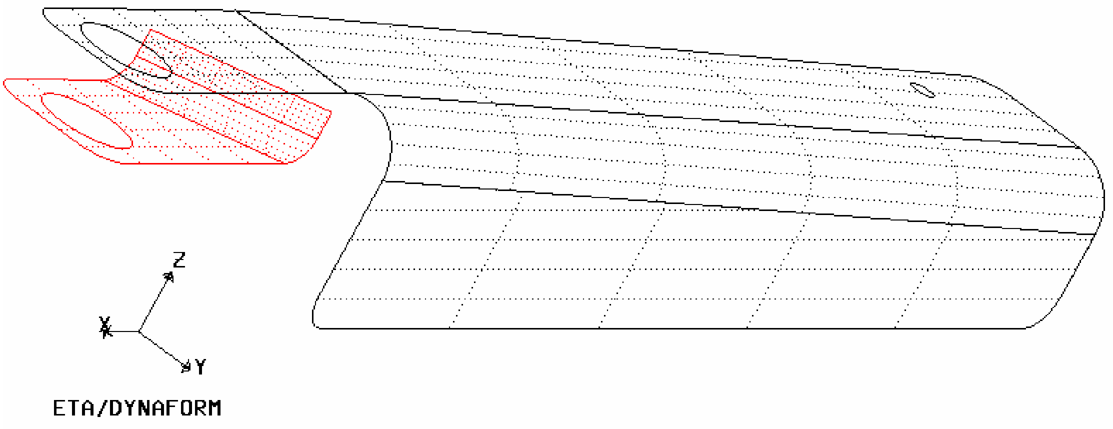


Figure 5.2.17 Group Bottom Surfaces

8. The program groups the surfaces for the bottom face. If the result is not perfect, use ADD BOTTOM SURFACES or REMOVE BOTTOM SURFACES to adjust the result. In this instance, it is necessary to add three surfaces to the bottom surface group. The result is shown in Figure 5.2.17.

9. Click DONE on Figure 5.2.16 to finish grouping. The result is seen in Figure 5.2.18.

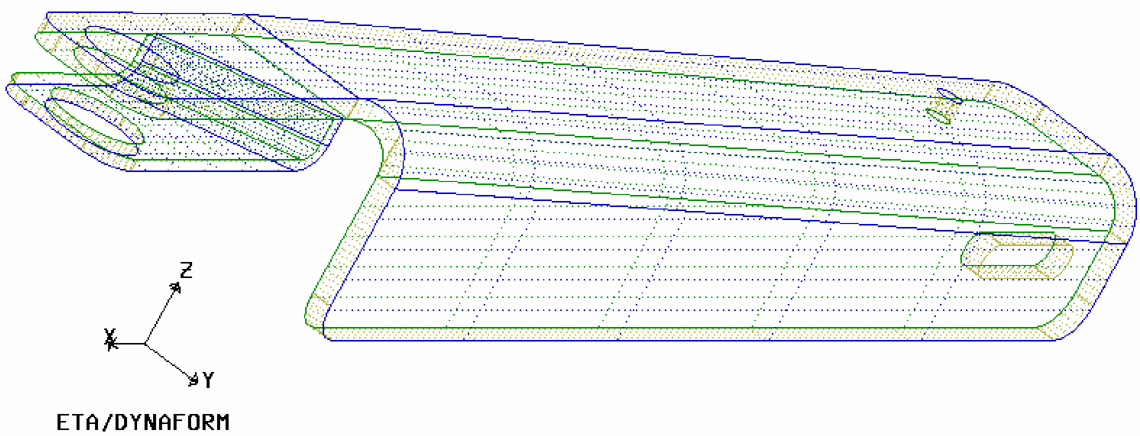


Figure 5.2.18 Result of Grouping

The thickness surfaces are put in the part called **ThkSrf1**, the top surfaces are put in the part called **TopSrf1**, and the bottom surfaces are put in the part called **BotSrf1**. The number “1” will be incremented by 1 for each subsequent grouping operation.

5.3 ELEMENT (CTRL+E)

The ELEMENT menu contains the functions to build elements. The functions are shown in Figure 5.3.1.

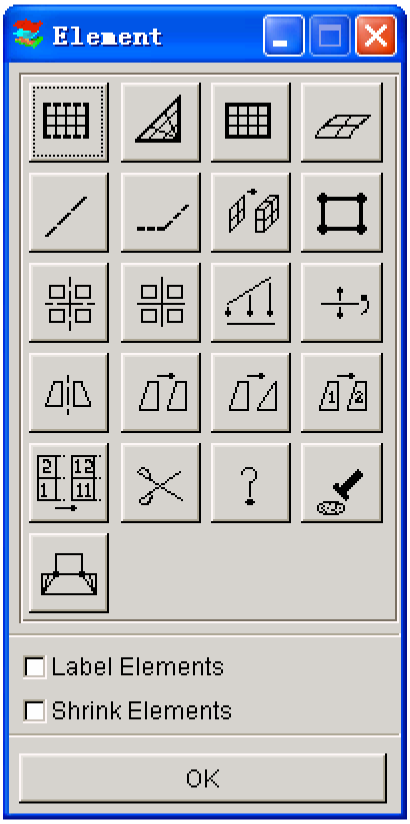


Figure 5.3.1 Element Options menu

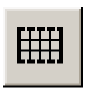
- **LABEL ELEMENT ON**

  Toggles the element label ON/OFF

- **SHRINK ELEMENT ON**

  Creates a plot with elements reduced in size by 20%

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

5.3.1 2 LINE MESH 

The created elements will be included in the current part.

- Select two lines in the submenu window. See Section 5.2.1 for available options.

- Enter the integers in N1, N2, N3, and N4 fields in the next dialog window.

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, and N4 < 2N2

If N3 and N4 are omitted, N1 will be 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 DYNAFORM Question window (Figure 5.3.2).

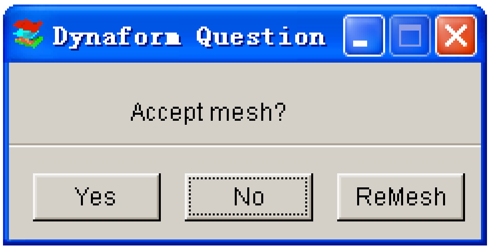


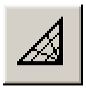
Figure 5.3.2 DYNAFORM Question window

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

Select NO not to accept 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 three lines are selected, the submenu SELECT OPTION window is displayed. See Figure 5.3.3.

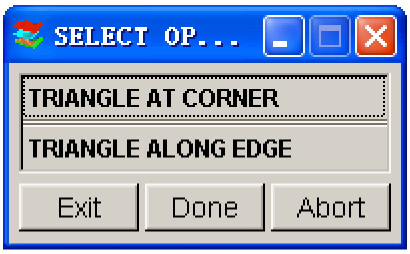


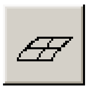
Figure 5.3.3 Select Option

- **TRIANGLE AT CORNER**
  The user needs to input two numbers, N1 and N2, in the next dialog window. 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 next dialog window. There will be N elements along each of the three lines. All triangular elements will be along the third selected line.

5.3.3 4 LINE MESH 

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

5.3.4 SURFACE MESH 

This function allows the user to create auto-mesh on the selected surface(s). The user must complete the definition in Figure 5.3.4.

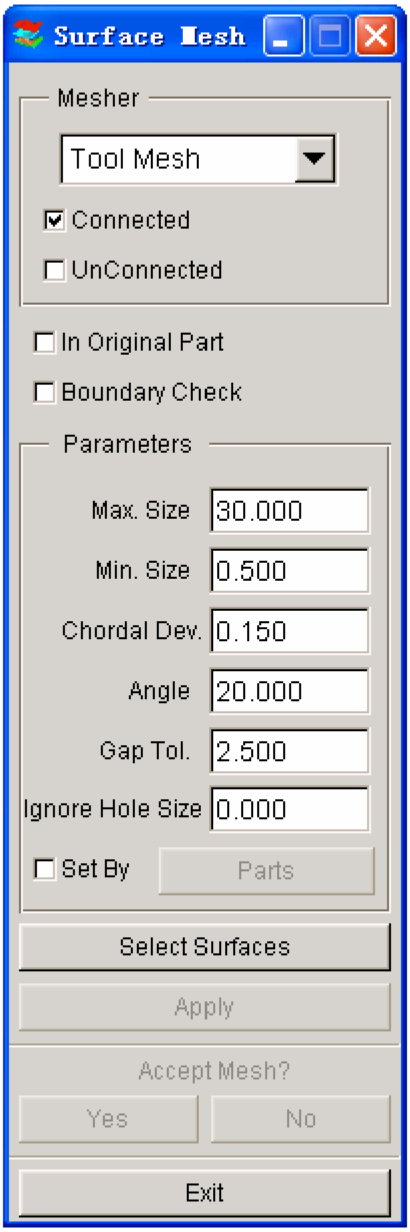


Figure 5.3.4 Surface Mesh

- Select the SELECT SURFACES button. See Section 4.3.3, PART/ADD SURFACE, for more information.

- Select a type of tool: CONNECTED or DISCONNECTED MESH, PART MESHER, or TRIANGLE MESHER.

  CONNECTED TOOL MESH will connect the adjacent surfaces if the gaps are within the GAP TOL.

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

  PART MESHER is the new topology mesher.

  TRIANGLE MESHER will only generate triangle elements.

- Select or deselect IN ORIGINAL PART. This option includes the created elements in original part or in current part.

- Select or deselect BOUNDARY CHECK. This option checks and displays the model boundary after building the topological structures.

- Enter the numbers of MAXIMUM SIZE, MINIMUM SIZE, ANGLE and CHORDAL DEVIATION in the data field.

  MAXIMUM SIZE
  Controls the maximum size

  MINIMUM SIZE
  Controls the minimum size

  ANGLE
  Controls the inclination of the adjacent elements

  CHORDAL
  Controls the number of elements at the radius

  IGNORE HOLE SIZE
  Fills the interior hole of the surface with mesh if the hole size is less than the given value

- Select APPLY TO MESH

If TOOL MESH is selected:

eta/DYNAFORM will create the 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 given in the PROMPT window.

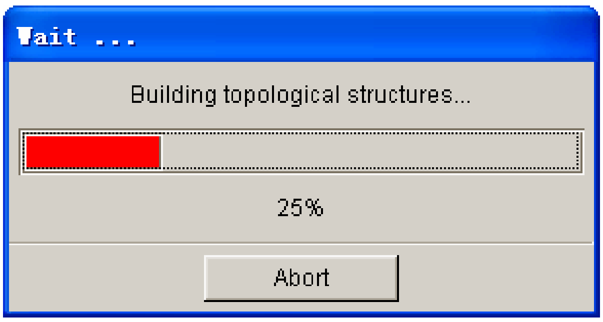


Figure 5.3.5 Building Topological Structures

- After building the topological structures, the DYNAFORM Question window appears. See Figure 5.3.6. Select YES to accept the boundary and continue to the next step. Select NO to display the BOUNDARY CHECK window. See Figure 5.3.7.

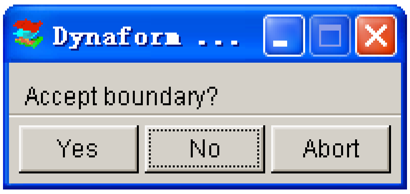


Figure 5.3.6 DYNAFORM Question

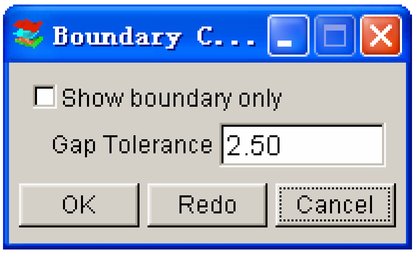


Figure 5.3.7 Boundary Check

Select/deselect SHOW BOUNDARY ONLY to display surfaces with model boundary. Select OK to accept the Gap Tolerance. Select REDO to enter the new GAP TOL to rebuild the topological structures.

- After the user selects or accepts the gap tolerance, eta/DYNAFORM begins to mesh surface boundary. See Figure 5.3.8. For mesh creation, see Figure 5.3.9.

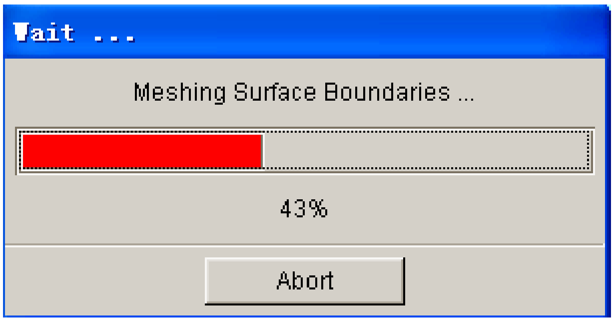


Figure 5.3.8 Meshing Surface Boundaries

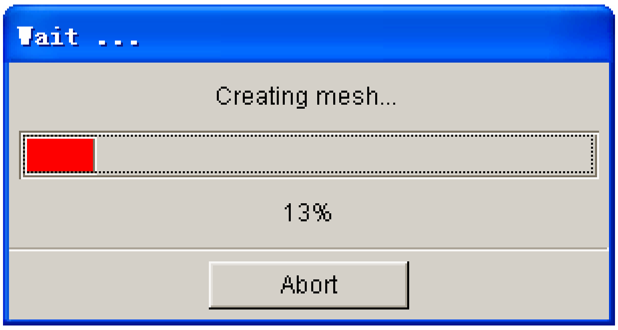


Figure 5.3.9 Creating Mesh

- The ACCEPT MESH CONFIRMATION (YES and NO) window will be activated in the SURFACE MESH window. 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. Refer to Figure 5.3.10 for the list of control parameters.

- CHECK SURFACE
  Check for duplicate surfaces while meshing.

- 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 delete free nodes.

- IGNORE HOLE SIZE
  Fill an interior hole in the surface with mesh if the hole size is less than the given value.

- MESH QUALITY
  Refer to the parameters to control mesh quality in Figure 5.3.11.

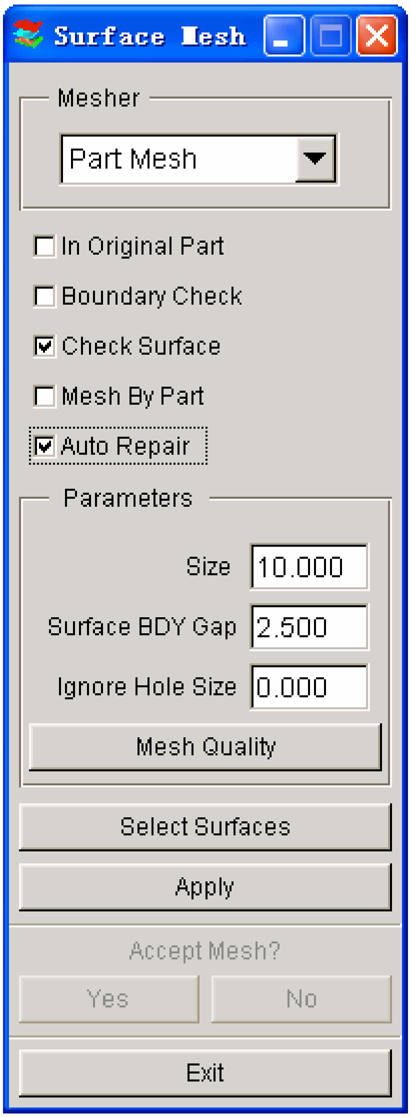


Figure 5.3.10 Part Mesher

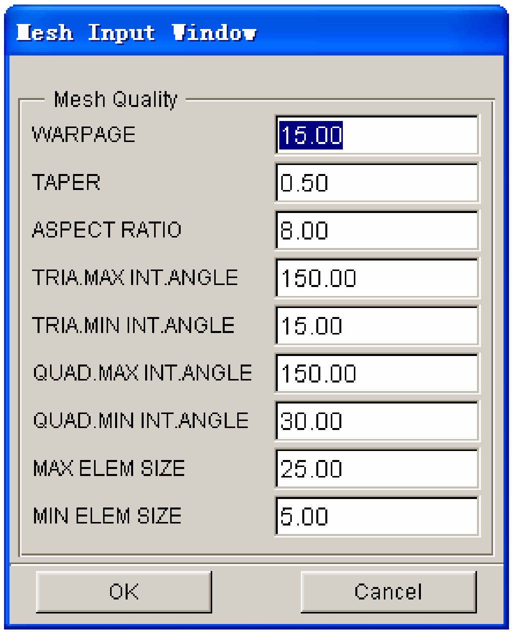


Figure 5.3.11 Mesh Quality

If TRIANGLE MESHER is selected:

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

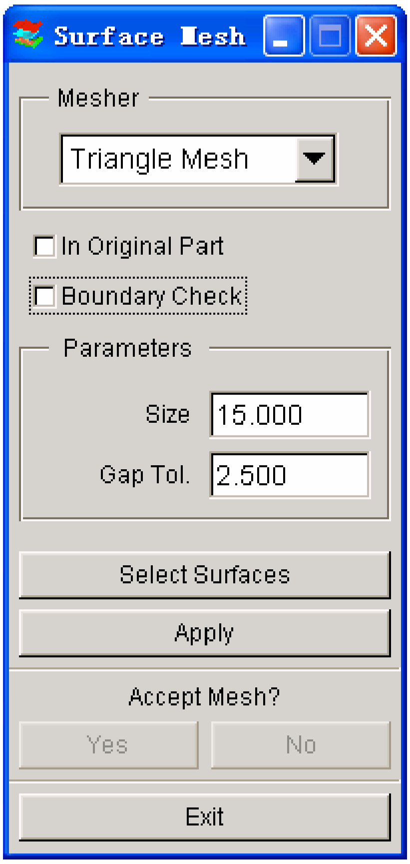
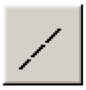


Figure 5.3.12 Triangle Mesher

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, and then press OK. See Figure 5.3.13.

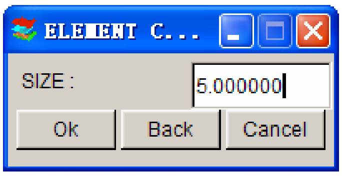


Figure 5.3.13 Element size

Note: Mesh pattern can be controlled by Size or Number. The option is defined in Option/Mesh Control/Line Mesh Method.

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

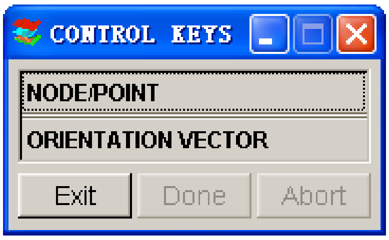


Figure 5.3.14 t Control Keys

- If NODE/POINT is selected, a new dialog window will be displayed as in Figure 5.3.15. This window prompts the user to select the appropriate NODES/POINTS and also provides an option to define the orientation vector.

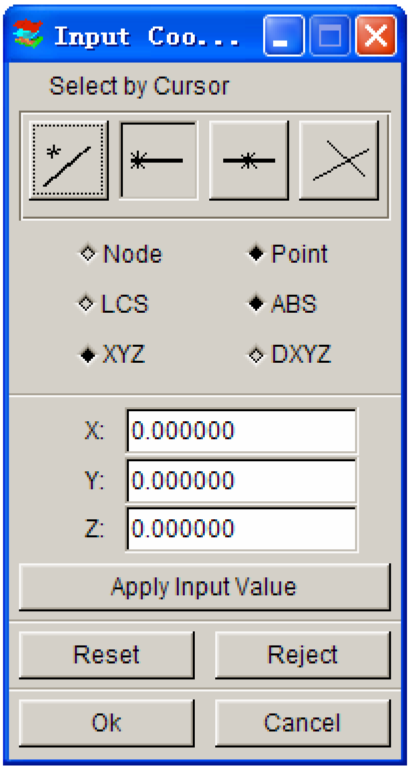


Figure 5.3.15 Select Node/Point

- If the user selects ORIENTATION VECTOR, a new dialog window will be displayed as in Figure 5.3.16. This window prompts the user to define the orientation vector.

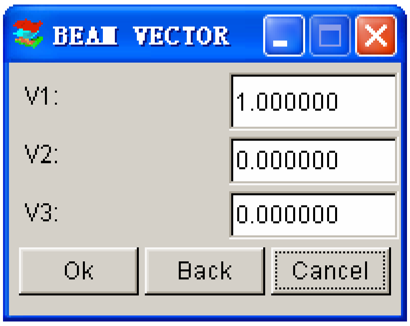
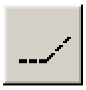


Figure 5.3.16 Beam Vector

- Select YES/NO/REMESH from the mesh confirmation window to accept/recject/remesh the result.

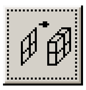
5.3.6 2 LINE POINT MESH 

This function is designed to help the user quickly remove the draw bead mesh. It allows the user to generate a mesh between two lines based on the points of the selected lines.

- Select two lines.

- Enter the number of layers between the two lines.

- Confirm acceptance of the generated mesh.

5.3.7 DRAG MESH 

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

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

1. ONE LINE DRAG

    - Select the elements for the drag pattern by using the options in the SELECT ELEMENT dialog window. See Figure 4.3.3 of Section 4.3.2, PART/ADD ELEMENT.

- Select a line. If the number of points is *n*, *n-1* elements will be generated for each pattern element.

- The user can accept, reject, or remesh.

2. NORMAL DRAG

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

- The user can accept, reject, or remesh.

Note: Plate element normal should be consistent in the drag pattern prior to the execution of NORMAL DRAG.

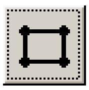
3. 3 OR 4 LINE DRAG

- Select the plate element pattern by using the options in the SELECT ELEMENT dialog window.

- Select three or four control lines to form a volume with the drag pattern. The user must select the control lines in clockwise or counterclockwise order.

- Enter the number of layers.

- The user can accept, reject, or remesh.

5.3.8 CREATE ELEMENT 

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

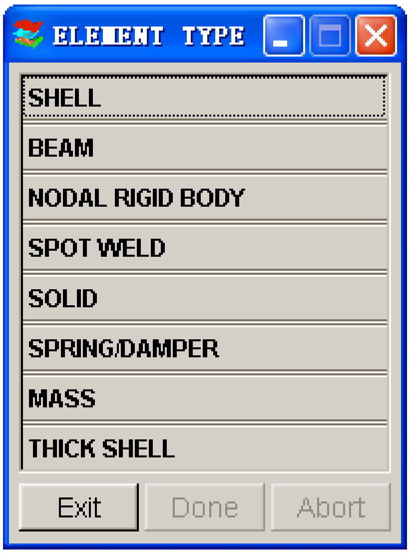


Figure 5.3.17 Element Type

1. BEAM

- Select two points or nodes by cursor in the screen.

- Define the material axis by selecting a third node/point or by defining the orientation vector.

  BY NODE

  BY POINT

  BY VECTOR

  Up to three values may be entered in the POP-UP dialog window 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 window. 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 window.

4. SHELL
This function allows the user to create quadrilateral and triangular elements in the current part. The INPUT COORDINATE dialog window 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 window.

- **TETRAHEDRON**

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

- **PENTAHEDRON**

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

- **HEXAHEDRON**

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

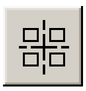
6. SPRING/DUMPER (2 NODE ELEMENTS)
   The element is created after two nodes/points are selected.

7. MASS (1 NODE ELEMENT)

- Enter the value of the mass in the field of the next dialog window.

- Select nodes/points as the mass element by using options in the INPUT COORDINATE dialog window.

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

5.3.9 COARSE ELEMENT 

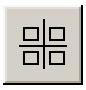
This function allows the user to coarsen two or four QUAD elements into one QUAD element, two TRIA elements into one QUAD element, or a QUAD/TRIA combination into one QUAD element.

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

- Select two adjacent plate elements, and select OK in the SELECT ELEMENT dialog window, or click the middle mouse button. If 4 adjoined 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 window. Each of the selected pairs above will be combined into one element.

5.3.10 SPLIT ELEMENT 

This function divides any SHELL elements into multiple elements based on the options described below. The options are shown in Figure 5.3.18.

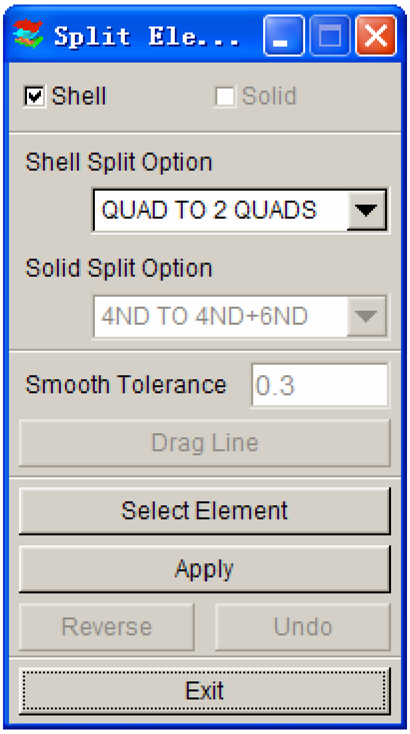


Figure 5.3.18 Split Element

1. SHELL SPLIT OPTIONS

   - Click the arrow next to SHELL SPLIT OPTION to access the drop-down menu of options as in Figure 5.3.19.

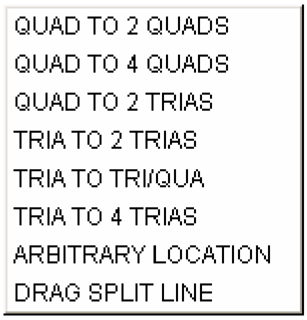


Figure 5.3.19 Shell Split Options

QUAD TO 2 QUADS
Split a quadrilateral into two quadrilaterals.

QUAD TO 4 QUADS

Split a quadrilateral into four quadrilaterals.

QUAD TO 2 TRIAS

Split a quadrilateral 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

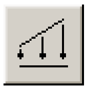
First select an element, and then select two nodes/points arbitrarily on the boundary of the element.

DRAG SPLIT LINE

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

- Select SELECT SPLIT OPTION for splitting.
- Click SELECT ELEMENTS to open the SELECT ELEMENT dialog window.
- Select the elements, and then select DONE to go back to the SPLIT ELEMENT dialog window.

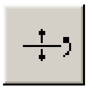
- Select APPLY to split the selected elements. After the elements are split, the user can select UNDO to cancel the split or REVERSE to change the split direction.

5.3.11 PROJECT ELEMENT 

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

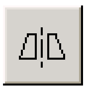
- Select the target part by using options in the SELECT PART dialog window.
- Define a coordinate system by using options in the LCS dialog window.
- Select a source part in order to project the element on the source part onto the target part.

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

5.3.12 REVERSE ELEMENT 

This function allows the user to reverse the element normal for selected elements except the mass.

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

5.3.13 MIRROR ELEMENT 

This function allows the user to generate a mirror image of selected elements. The available options are shown in Figure 5.3.20

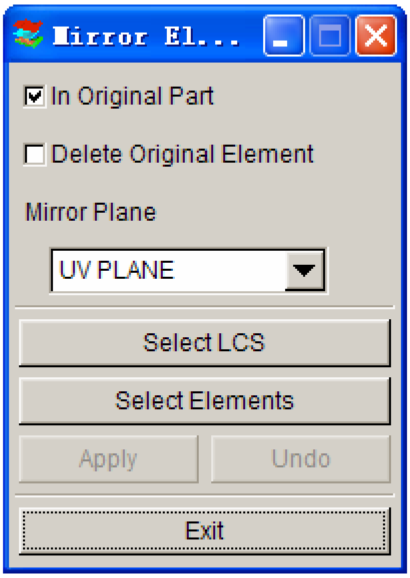
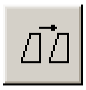


Figure 5.3.20 Mirror Element

For more information on mirror elements, see Section 5.1.19, MIRROR LINES.

5.3.14 COPY ELEMENT



This function allows the user to generate a set of duplicate elements. The user may simultaneously translate, rotate, offset (in the normal direction), or map the copied elements. The options are shown in Figure 5.3.21. See Section 5.1.5 for COPY and TRANSFORM LINE and Section 9.2.7 for MATING TOOLS FOR OFFSETTING.

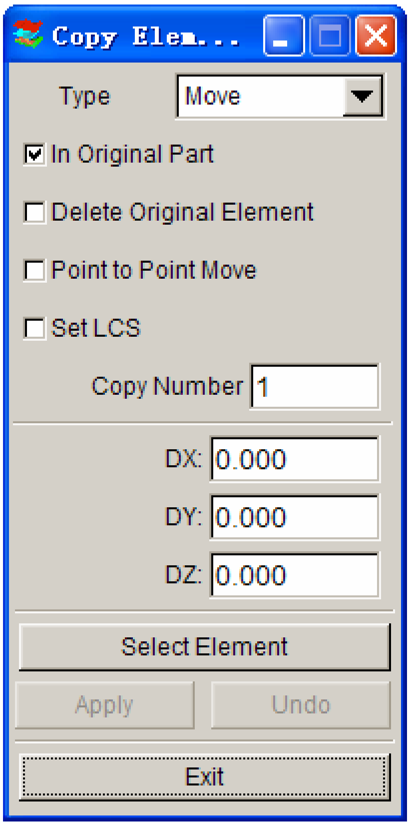
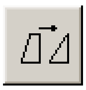
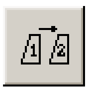


Figure 5.3.20 Copy Element

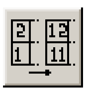
For the MAPPING option, the user defines only two coordinate systems. The selected elements are copied from the place in the first coordinate system to the same place in the second coordinate system.

5.3.15 MODIFY ELEMENT 

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

5.3.16 CHANGE ELEMENT NUMBER 

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

5.3.17 RENUMBER ELEMENT 

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

SEQUENCE

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

BY PART

A DYNAFORM Question window appears, Figure 5.3.22.

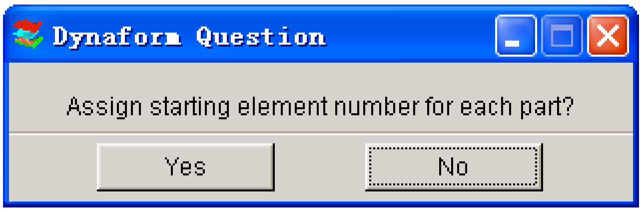


Figure 5.3.22 Starting Element Number Question

1. YES
   eta/DYNAFORM prompts

   ENTER STARTING ELEMENT NUMBER FOR PART: xxx

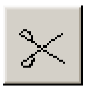
   and displays a dialog box for the user to enter the starting element number of this part.

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

   ELEM x TO xx ASSIGNED, NEXT ELEM NO: yyyy

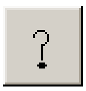
   This procedure will be repeated for every part in the database.

2. NO
   Two INPUT dialog windows will appear. The first is for entering the starting element number, and the second is for entering the element increment between parts (default is 1000).

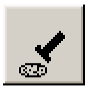
5.3.18 DELETE ELEMENT 

This function allows the user to delete elements from the model.

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

5.3.19 IDENTIFY ELEMENT 

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

5.3.20 FIND ELEMENT 

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

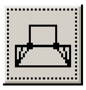
- Once the element number is entered, the program will highlight the element on the screen and label its node numbers. The node numbers will also be displayed in the PROMPT window.

- If the element is not displayed on the screen, the prompt will read:

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

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

  ELEMENT xxxx NOT FOUND IN DATABASE

5.3.21 AUTO REPAIR 

This function automatically repairs gaps and improves element quality in the mesh by merging nearby nodes and modifying elements. Caution should be used when using this function as it can change the geometry of the tooling. Several user-defined controls are given for identifying and repairing the imperfect mesh. After selecting elements that need to be repaired, the REPAIR PARAMETERS window appears as shown in Figure 5.3.23.

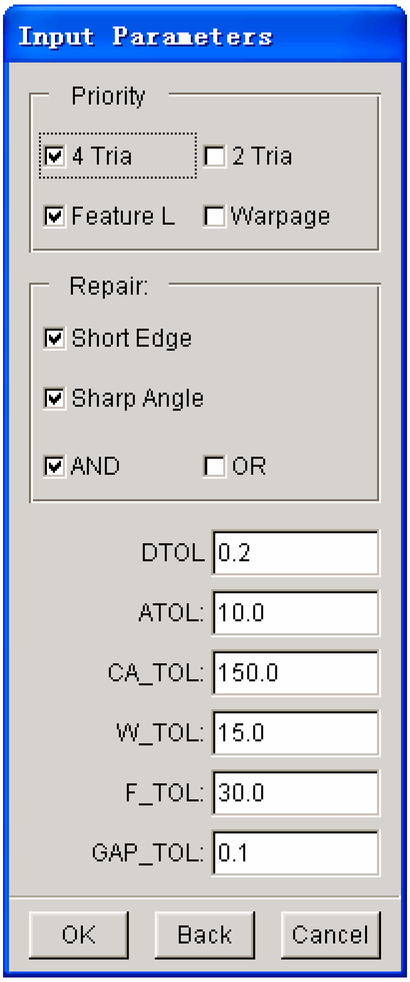


Figure 5.3.23 Auto Repair

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

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

PARAMETERS FOR CHECKING BAD ELEMENTS
These parameters are the criteria for repairing bad elements: MINIMUM ELEMENT SIZE (DTOL), SHARP ANGLE TOLERANCE (ATOL), LARGE ANGLE TOLERANCE (CA_TOL), WARPAGE TOLERANCE (W_TOL), FEATURE LINE ANGLE TOLERANCE (F_TOL), and GAP TOLERANCE (GAP_TOL).

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

5.4 NODE (CTRL+N)

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

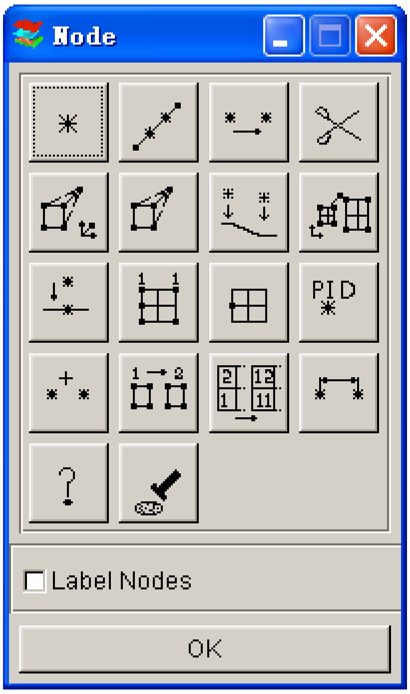
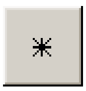


Figure 5.4.1 Node Options

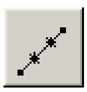
- **LABEL NODES ON**

  Toggles the node labels ON/OFF.

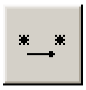
A detailed description of these functions is given in the following sections.

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 detail on how to select points or enter coordinates. The user can create nodes by using options in the INPUT COORDINATE dialog window.

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 window. Enter the number of added nodes in the dialog window and the nodes will be generated.

5.4.3 COPY NODE 

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

- Select nodes by using options in the SELECT NODE dialog window. See Figure 5.4.2.

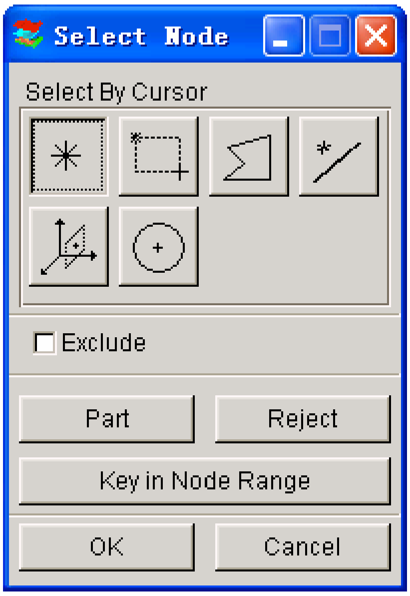
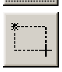
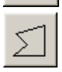


Figure 5.4.2 Select Node

There are several ways to select nodes:

 By cursor.

 By drag window: Select two points in the screen to form a window.

 By multi-point region: Select several points on the screen to form a multi-point region.

 Near a line: Select a line and enter the tolerance.

 Near a plane: Select a plane and enter the location and tolerance.

 All the nodes are in the circle will be selected.

BY PART NAME
Select the PART option and select the part that includes the nodes.

BY NODE RANGE
Select KEY IN NODE RANGE. A dialog window will be displayed for entering the starting

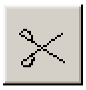
number, ending number, and increment.

- Select OK after node selection.
- Define a coordinate system by using options in the LCS dialog window.
- Enter the number of copies in the next dialog window.

There are two ways to copy nodes:

1. MOVE
   Enter displacements in the next dialog window.

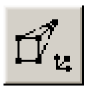
2. ROTATE
   Enter the angle increment in the next dialog window. The nodes will rotate along the W-axis.

5.4.4 DELETE UNREFERENCED NODES 

This function allows the user to delete 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 deleted automatically.

SELECTED FREE NODES
Select free nodes by using options in the SELECT NODE dialog window.

5.4.5 TRANSFORM NODE 

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

- Define a new coordinate system by using options in the LCS dialog window.

- Select the MOVE or ROTATE option and enter the value of displacement or angle increment**.**

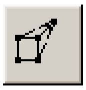
- Select nodes by using the options in the SELECT NODE dialog window.

- There are three choices after selected nodes are transformed.

1. AGAIN
   Repeats the last translation.

2. REVERSE OPERATION
   Performs the translation in the opposite direction.

3. RESELECT NODE
   Selects a new set of nodes for another copy operation.

5.4.6 MOVE NODE 

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

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

  MULTIPLE NODES FOUND

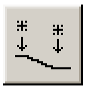
  SELECT ELEMENT FOR NODE

  If the selected element is not connected with the node, eta/DYNAFORM prompts:

  SELECTED ELEM IS NOT CONNECTED TO DUPL NODE

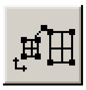
  SELECT ELEMENT FOR NODE

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

5.4.7 ALIGN 

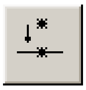
This function allows the user to align the selected nodes along the selected lines.

- Select nodes to be aligned.
- Select a line to align the selected nodes
- Nodes are moved to the selected line according to the shortest distance to the line.

5.4.8 SCALE NODE 

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

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

5.4.9 PROJECT NODE 

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

1. ON F. E. MESH

   - Select a set of elements by using options in the SELECT ELEMENT dialog window.
   - Create a new local coordinate system to define the projection vector. The nodes are projected along the W-axis.
   - Select nodes to project.
   - 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 in the LCS dialog window.
   - A DYNAFORM Question window is displayed. See Figure 5.4.3.

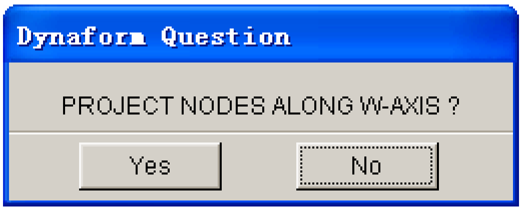


Figure 5.4.3 Project Node Question

   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 window.

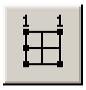
   Selecting NO prompts the user to create a new coordinate system. .

3. ON SURFACE

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

4. ON SHELL ELEMENT (NORMAL)

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

5.4.10 CHECK DUPLICATE NODES 

This function allows the user to renumber any duplicate node ID numbers found in the database. The user cannot create duplicate node numbers from within the program. However, the user may import files containing duplicate nodes from other programs. eta/DYNAFORM will renumber the duplicate nodes to the higher number from the current maximum node number.

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

CHECKING nnnn NODES FOR DUPLICATE NODES

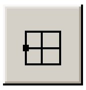
DUPLICATE NODE xxxx CHANGED TO yyyy

mmmm DUPLICATE NODES FOUND

If no duplicate nodes are found, the following messages will be given in the PROMPT window and in the DYNAFORM.msg message file located in the work directory:

CHECKING nnnn NODES FOR DUPLICATE NODES

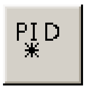
NO DUPLICATE NODE FOUND

5.4.11 CHECK COINCIDENT NODES 

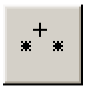
This option allows the user to check and merge coincident nodes that are currently displayed. Coincident nodes are defined as having a distance between each other that is less than or equal to a specified tolerance.
eta/DYNAFORM merges all coincident nodes into the one with the lowest ID number and sets all others as free nodes.

- Select an option in the dialog window.
- Type the tolerance in the dialog window. eta/DYNAFORM displays the number of coincident nodes in the PROMPT window and displays the DYNAFORM QUESTION window to confirm that the user wants to merge the coincident nodes.

- Answer YES to merge the coincident nodes.

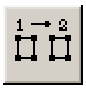
5.4.12 PART CONNECT 

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

5.4.13 COMPACT NODE 

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

- Delete nodes. See Section 5.4.4.
- Enter a number as the starting number for the remaining nodes.

5.4.14 CHANGE NODE NUMBER 

This function allows the user to change the ID numbers of the existing nodes.

1. NODE
   Select a node by cursor, or select the KEY IN option (step 2).
2. KEY IN NODE NO.
   Enter the node number in the next dialog window.
3. After a node is selected, eta/DYNAFORM prompts:

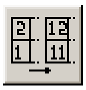
   ENTER NEW NODE NO FOR xxx

   If the entered number already exists in the database, a message will be given in the PROMPT window:

   NODE NUMBER xxx ALREADY EXISTS, REQUEST DENIED

   ENTER NEW NODE NUMBER FOR xxx

4. UNDO LAST
   Reject the last operation.

5.4.15 RENUMBER NODES 

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

- Another DYNAFORM QUESTION window appears. See Figure 5.4.4.

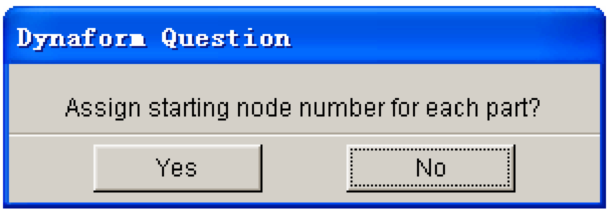


Figure 5.4.4 Starting Node Number Question

1. YES
   eta/DYNAFORM prompts:

   ENTER STARTING NODE NO. FOR PART? xxxx

   Once a number is entered, the prompt will read:

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

   ENTER STARTING NODE NUMBER FOR PART? xxxx

   These prompts will continue for every part in the database. If the user enters a starting number that already exists, eta/DYNAFORM displays a message that the number has already been assigned, and returns the user to the beginning of this step.

   NODE NUMBER HAS BEEN ASSIGNED

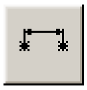
2. NO
   eta/DYNAFORM prompts:

   ENTER STARTING NODE NUMBER

   Once a number has been entered, the prompt will read:

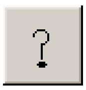
   ENTER NODE NUMBER INC. BETWEEN PARTS

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

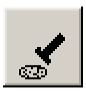
5.4.16 DISTANCE BETWEEN NODES/POINTS 

This function allows the user to calculate the distance between two nodes, two points or a node and a point.

The nodes/points are selected by using options in the INPUT COORDINATE dialog window. The distance is calculated automatically and is shown in the PROMPT window.

5.4.17 IDENTIFY NODE/POINT 

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

5.4.18 FIND NODE 

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

5.5 MESH REPAIR (CTRL + R)

This menu provides a convenient way of repairing mesh. It collects the most commonly used functions from the NODE, ELEMENT and MODEL CHECK menus to streamline the mesh repair operation. Figure 5.5.1 shows all the functions in the MESH REPAIR menu. 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, AUTO REPAIR AND SHRINK and LABEL ELEMENT. A detailed description of these functions is included in their respective sections in the user's manual.

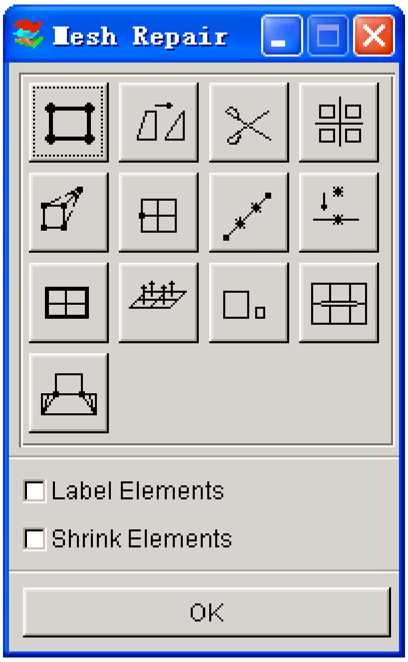


Figure 5.5.1 Mesh Repair menu

5.6 MODEL CHECK (CTRL+M)

The functions in the MODEL CHECK menu are used to validate the model to meet the specific forming simulation criteria. The functions in this menu are shown in Figure 5.6.1.

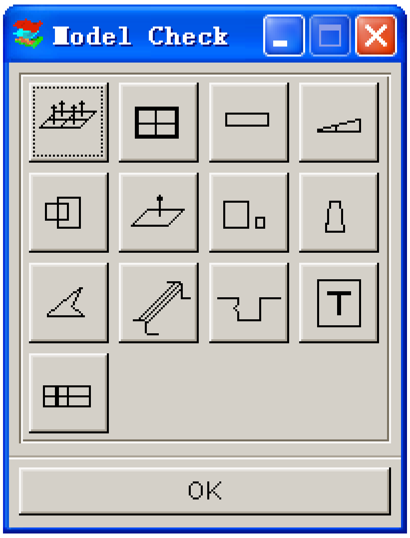
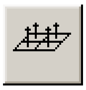


Figure 5.6.1 Model Check

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

5.6.1 AUTO PLATE NORMAL 

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

- There are two options to select elements: ALL ACTIVE PARTS and CURSOR PICK PART. The program will display the current normal direction of the selected element.
- A DYNAFORM QUESTION window appears. See Figure 5.6.2

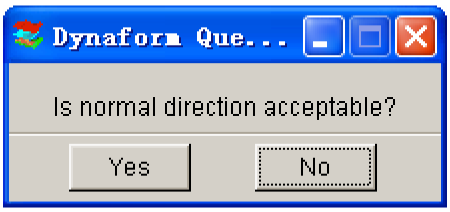
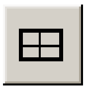


Figure 5.6.2 DYNAFORM Question

Select YES, and the normal of all selected elements will be made consistent with the referenced element.

Select NO, and the normal direction of the reference element will be reversed. The normal of all selected elements will be made consistent with the reference direction.

Note: It is recommended that the user select one part at a time. Parts with branches may not obtain consistent normal direction.

5.6.2 BOUNDARY DISPLAY 

This function allows the user to check the boundary of displayed part(s). It is used to check the proper connection between 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. See Figure 5.6.3.

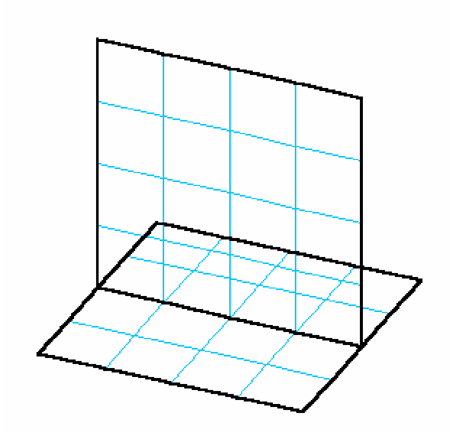
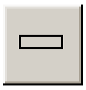


Figure 5.6.3 Surface Boundaries

Note: The boundary will remain highlighted until the user clicks the CLEAR icon.

5.6.3 ASPECT RATIO CHECK 

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

1. The user may accept the default value of 8.0 or enter any positive real number that is a valid value for the aspect ratio. eta/DYNAFORM runs a check on the values for the aspect ratio. Any elements that exceed the defined aspect ratio are highlighted. The messages echoed in the PROMPT window are:

   xxx ELEMENTS FAILED CHECK ASPECT RATIO= xxx

   or

   ALL ELEMENTS PASS CHECK

   The number of failed elements and maximum aspect ratio will be displayed in the PROMPT window.

2. If there are failed elements, a DYNAFORM QUESTION window appears. See Figure 5.6.4.

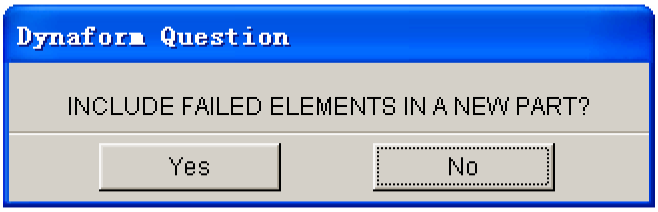


Figure 5.6.4 DYNAFORM Question Window

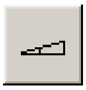
Select YES and the CREATE PART dialog window will appear.

Select NO to skip to the next function.

3. eta/DYNAFORM prompts:

   SELECT ELEMENT FOR ASPECT RATIO

   The user can select any highlighted element, and its aspect ratio will be shown in the PROMPT window.

5.6.4 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 the option to adjust the default criteria as necessary.

1. Enter the interior angle in the next dialog window (DEFAULT = 1 DEGREES). If the elements pass the interior angle check, eta/DYNAFORM displays the following message and returns the user to the MODEL CHECK menu:

   ALL ELEMENTS PASS CHECK!

   If some of the elements fail the angle check, eta/DYNAFORM displays the following message with the option to create a new part.

   xxxx ELEMENTS FAILED CHECK

2. At same time, a DYNAFORM QUESTION window appears as in Figure 5.6.5. Select PUT IN NEW PART, and the CREATE PART dialog window appears. Select DELETE, and all failed elements are deleted. Select CANCEL, and the function exits.

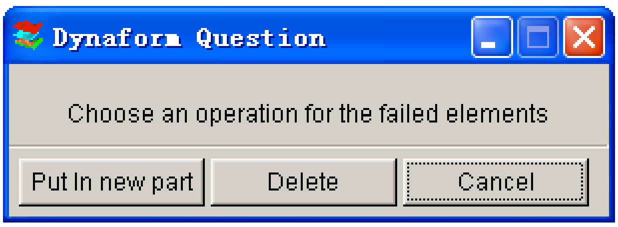
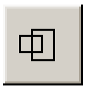
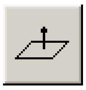


Figure 5.6.5 DYNAFORM Question

5.6.5 OVERLAP ELEMENT 

This function allows the user to check for overlapping elements that share two or more edges.

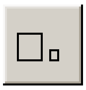
If there are failed elements, the user can include these elements in a new part.

5.6.6 PLATE NORMAL 

This function is used to draw a boundary line between elements with opposite normal directions.

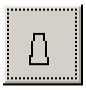
If there are no opposing plate normals, eta/DYNAFORM prompts:

NORMAL CHECK COMPLETED, NORMAL IS CONSISTENT

5.6.7 ELEMENT SIZE 

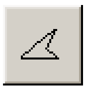
This function is used to check the minimum length of the edges of plate, solid, or beam elements. eta/DYNAFORM highlights all elements with an edge shorter than the user-defined length criteria. The default is 1.0. The user may change the default value.

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

5.6.8 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 next dialog window (default is 0.5).

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

5.6.9 CHECK WARPAGE 

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

1. Enter the warpage criteria (default is 5.0 degrees).
2. If there are any failed elements, a DYNAFORM QUESTION window appears as in Figure 5.6.6.

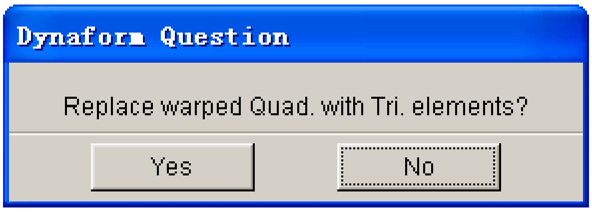


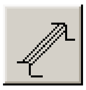
Figure 5.6.6 DYNAFORM Question Window

Select YES to replace the warped quadrilateral with a triangular element.

Select NO to reject the operation.

3. If the warped elements are not replaced, eta/DYNAFORM prompts to include the failed elements in the new part or moves to the next step.

4. The user can select any element show its warpage value in the PROMPT window.

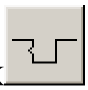
5.6.10 FEATURE LINE 

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

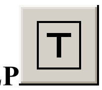
1. Enter angle criteria (default is 20 degrees).

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 prompts to generate the plate element.

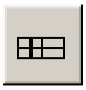
3. If the user selects YES, eta/DYNAFORM will prompt to include the element in a new part.

5.6.11 DIE LOCK 

This function is used to verify that the construction of the die is correct by determining that there is no interference from the die to prevent the punch from traveling to its home position. The program checks for an undercut of the elements defining the tool. The user has the option to add failed elements to a new part.

5.6.12 TIME STEP 

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

5.6.13 SECTION CUT 

There are several functions in the sub menu as shown in Figure 5.6.7.

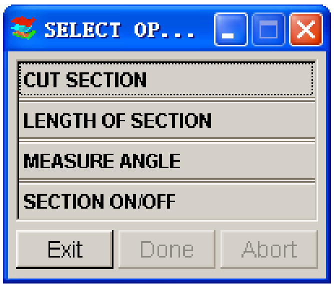


Figure 5.6.7 Section Cut

1. CUT SECTION
   This function allows the user to generate section lines that are intersections of the displayed F.E. mesh with a set of parallel planes.

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

2. LENGTH OF SECTION
   This function allows the user to measure the arc lengths of a line.
   There are three options in this function:

   - **CURSOR LOCATION**
     Select two points on a line. eta/DYNAFORM calculates the arc length between two points and prompts:

     LINE SEGMENT LENGTH IS: xxx

   - **LINE**
     Select the desired line by cursor. The entire length of this section line will be calculated and given in the PROMPT window.
   - **REJECT LAST**
     Reject the last selection.

3. MEASURE ANGLE
   This function allows the user to measure the angle between two segments on a line.

- Select the desired line by cursor.
- Select two points on the line to define the linear regression line 1 in red. The user can reset the selection by using REJECT LAST POINT.
- Select another two points on the line to define the regression line 2 in red. The user can reset the selection by using REJECT LAST POINT.
- eta/DYNAFORM calculates the angle between the two regression lines. The result is given in the PROMPT window.

4. SECTION ON/OFF
   This function toggles all parts OFF except the section part. If toggled again, the parts that were initially displayed will be re-displayed.

5.7 BOUNDARY CONDITIONS (CTRL+U)

The functions of the BOUNDARY CONDITIONS MENU allow the user to create and verify constraints and loads on finite element models. The functions in the BOUNDARY CONDITIONS MENU are shown in Figure 5.7.1.

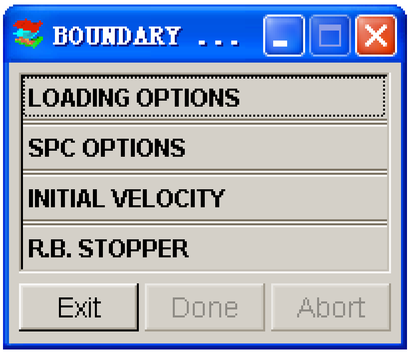


Figure 5.7.1 Boundary Conditions

5.7.1 LOADING OPTIONS

The LOAD SET dialog window displays as in Figure 5.7.2.

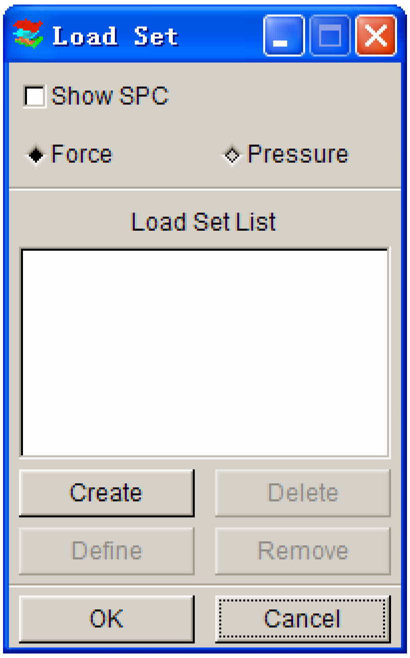


Figure 5.7.2 Load Set

SHOW SPC
Toggle the SPC (single point constraint) display ON or OFF. See SPC OPTIONS.

1. CREATE FORCE at selected nodes.

- Toggle FORCE on.
- Select CREATE.
- Enter the load set number in the next dialog window.
- Define node force as shown in Figure 5.7.3.

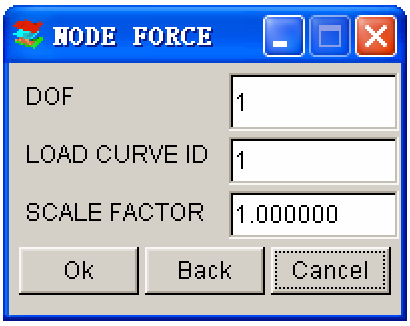


Figure 5.7.3 Define Node Force

DOF
Degree of freedom. The meaning of these numbers is as follows:

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 load value versus time. Refer to Section 10.9.1, UTILITIES/LOAD CURVE/CREATE CURVE for load curve description.

SCALE FACTOR
Load the curve scale factor.

- Select OK in Figure 5.7.3 to select the nodes to load. Refer to Figure 5.4.2 for selecting nodes.

- Select CANCEL in Figure 5.7.3 to exit.

2. CREATE PRESSURE on selected elements.

- Toggle PRESSURE on.

- Select CREATE.

- Enter the load set number in the next dialog window.

- Define element pressure. See Figure 5.7.4.

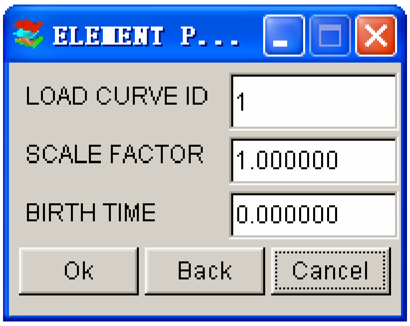


Figure 5.7.4 Define Element Pressure

LOAD CURVE ID
See CREATE FORCE.

SCALE FACTOR
See CREATE FORCE.

BIRTH TIME
Time imposed motion/loading is active.

- Select OK in Figure 5.7.4 to select the element to load.

- Select CANCEL in Figure 5.7.4 to exit.

The following three functions are used for both loading node force and element pressure.

DELETE
A DYNAFORM QUESTION dialog window appears. See Figure 5.7.5.

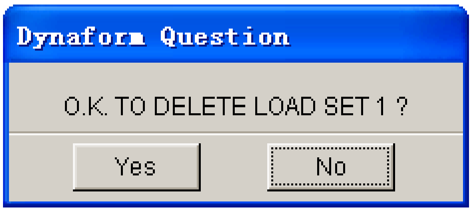


Figure 5.7.5 DYNAFORM Question Window

Select YES to delete the last selected load set.

Select NO to reject the choice.

DEFINE
Add more nodes or elements to a selected load set.

REMOVE
Remove some selected nodes/elements from a load set.

5.7.2 SPC OPTION

This function allows the user to create and/or modify the SPC options. The SPC SET dialog window appears as in Figure 5.7.6.

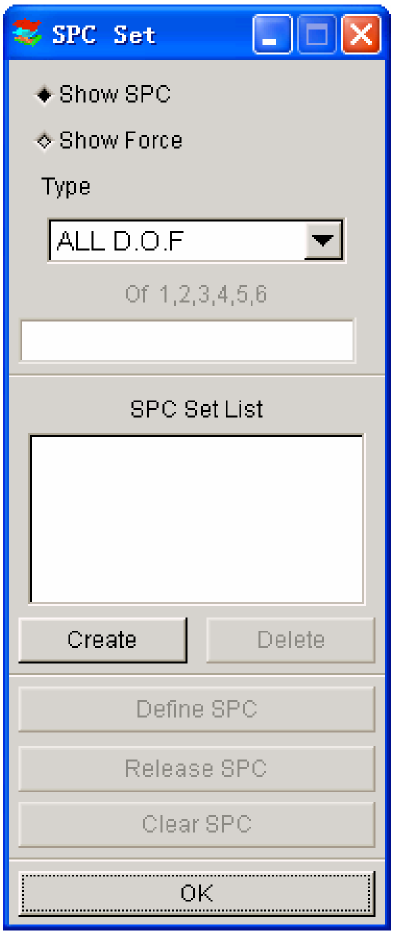


Figure 5.7.6 SPC option

SHOW SPC ON
Toggle SPC ON/OFF to show the current SPC set.

SHOW FORCE ON
Toggle defined force ON/OFF to show the current load force.

1. CREATE SPC SET

- Select an SPC type. There are seven types of degree of freedom in LS-DYNA:

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

The following menu (Figure 5.7.7) provides the ways to define one or several constraints.

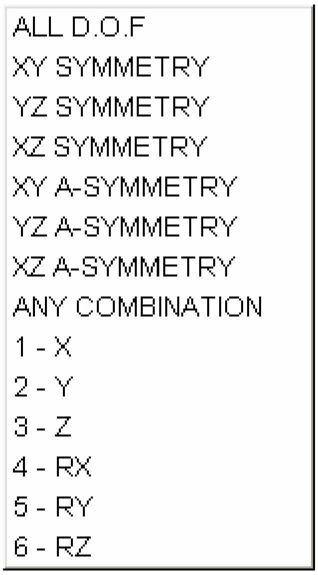


Figure 5.7.7 SPC types

- Select CREATE.
- Enter the SPC Set ID Number in the next window.
- Select nodes to constrain.

2. DELETE SPC SET

- Select an SPC set from the list.
- Select DELETE.
- Confirm the deletion in the DYNAFORM QUESTION window (Figure 5.7.8).

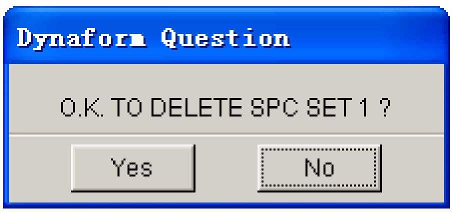


Figure 5.7.8 DYNAFORM Question Window

3. DEFINE SPC

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

4. RELEASE SPC
Remove some nodes from the selected SPC set.

5. CLEAR SPC
Remove all nodes from a selected SPC set. A DYNAFORM QUESTION window is displayed as in Figure 5.7.9.

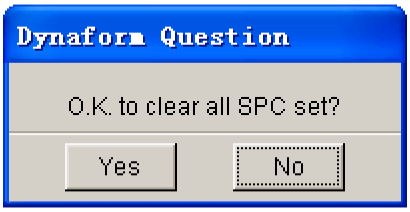


Figure 5.7.9 DYNAFORM Question Window

Select YES to delete all SPC.

Select NO to reject the choice.

5.7.3 INITIAL VELOCITY

This function allows the user to define and assign linear or angular velocities to selected nodes. See Figure 5.7.10 for available options.

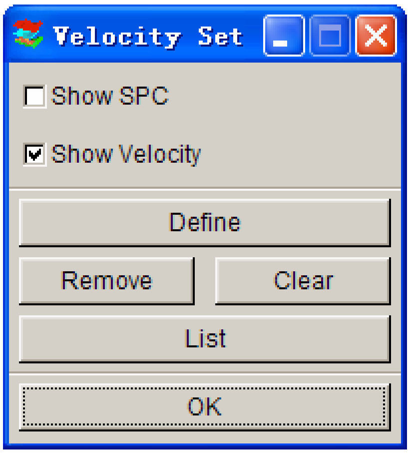


Figure 5.7.10 Initial Velocity Set

SHOW SPC
See SPC OPTION.

SHOW VELOCITY
Toggle a defined velocity set ON/OFF.

DEFINE
Create a velocity set.

- Select DEFINE.
- Enter the initial velocity in the INITIAL VELOCITY dialog window as in Figure 5.7.11.

- Select nodes.

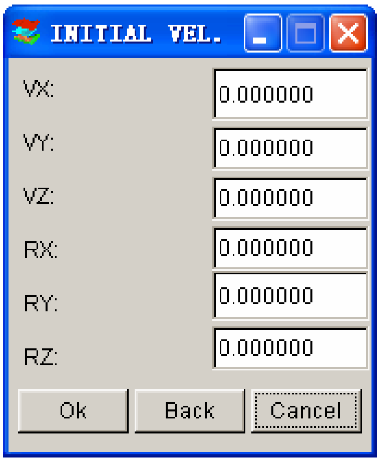


Figure 5.7.11 Velocity Define Table

REMOVE
Remove defined initial velocity from the selected nodes set.

CLEAR
Remove all defined initial velocity from the database. A DYNAFORM QUESTION window appears as in Figure 5.7.12.

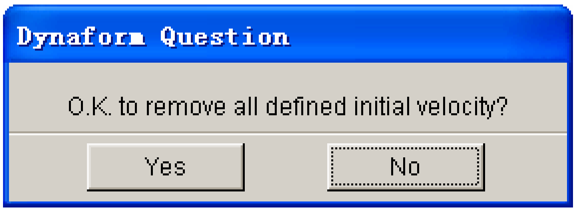


Figure 5.7.12 DYNAFORM Question Window

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 PROMPT window.

5.7.4 RIGID BODY STOPPER

This function allows the user to define the RIGID BODY STOPPER. The RIGID BODY STOPPER provides a convenient way of controlling the motion of rigid tooling in the forming process. For a detailed description, refer to the LS-DYNA User's Manual. The available options are shown in Figure 5.7.13.

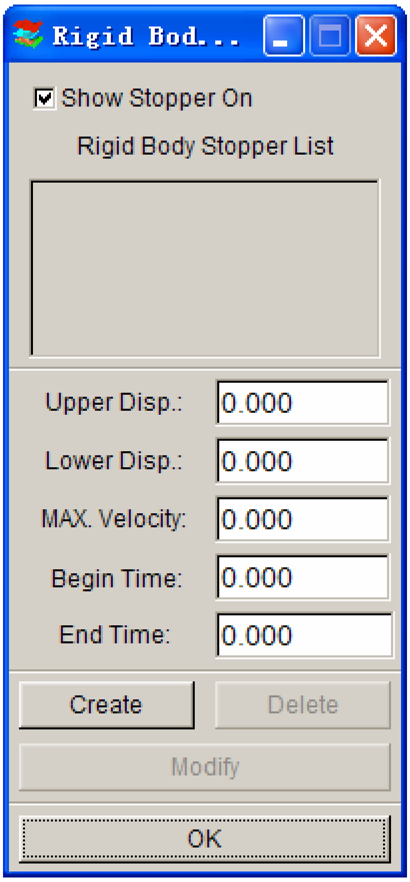


Figure 5.7.13 Rigid Body Stopper

- Enter the value in the 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 delete the defined stopper as in Figure 5.7.14.

Figure 5.7.14 DYNAFORM Question Window

- 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.

5.8 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 5.8.1.

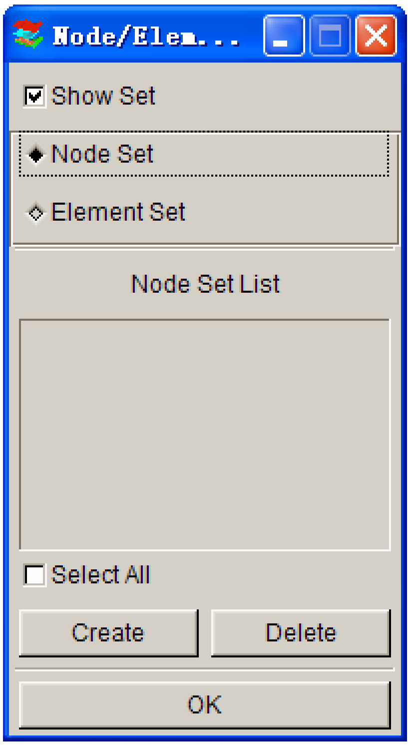


Figure 5.8.1 Node/Element Set Menu

- Toggle on NODE SET or ELEMENT SET.
- Select CREATE.
- Select nodes or elements in the corresponding dialog window.
- Select DELETE to delete a set that is highlighted in the list.

CHAPTER 6
DFE (Die Face Engineering)

The DYNAFORM-DFE module, Die Face Engineering, has been developed to generate Die Face (Addendum and Binder) in the early stages of die design. The functions of the DYNAFORM-DFE menu are shown in Figure 6.1.

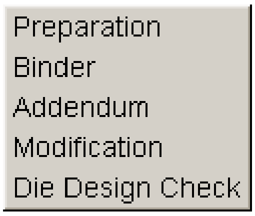


Figure 6.1 DFE menu

Features of the DFE module include the following highlights.

- Unfolding flange
- Automatic filleting for sharp edges
- Automatic filling of holes, boundary fill and outer boundary smooth
- 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
- Creating a Drawbar
- 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 in order to begin the die face design process. The user may import part geometry or part mesh using DFE/PREPARATION/IMPORT functions. Refer to Section 3.6, IMPORT, for a detailed description of importing geometry file and mesh file.

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

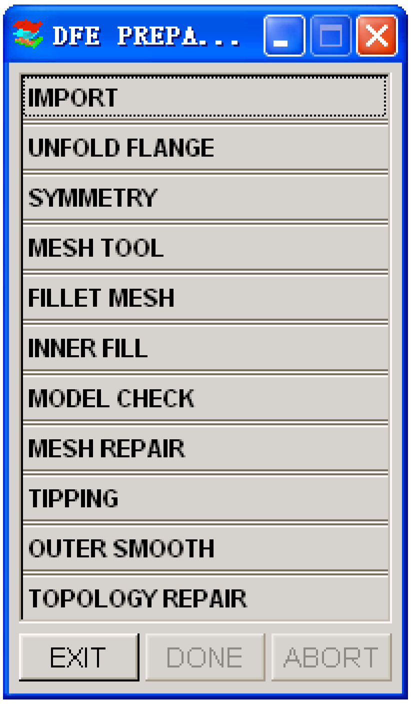


Figure 6.1.1 Preparation menu

6.1.1 UNFOLD FLANGE

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

- Once this function is selected, DYNAFORM opens the SELECTED SURFACE window to select flange surfaces to be unfolded. Refer to Figure 6.1.2 for typical surfaces with the defined flange highlighted.

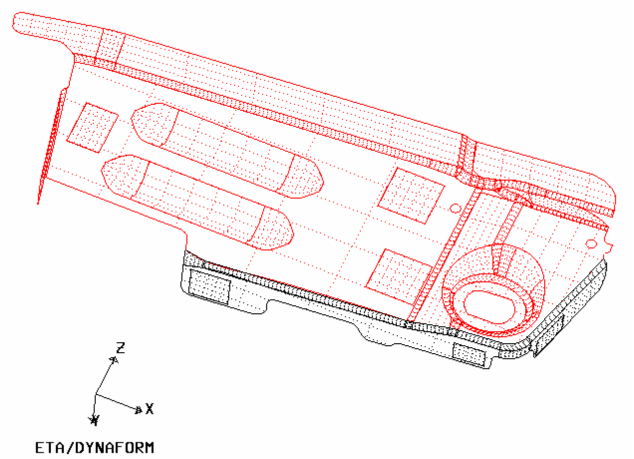


Figure 6.1.2 Select Flange Surface

- After selecting surfaces, DFE will find and highlight the unfold line and the boundary line. The bold line is the unfold line, and the other is the boundary line of the base surface and unfolded surface as shown in Figure 6.1.4. At the same time, the program displays an option window as shown in Figure 6.1.3 to select the option for the next operation.

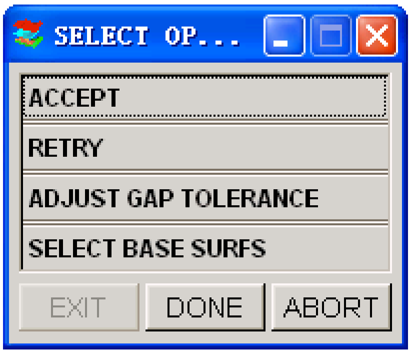


Figure 6.1.3 Select Options

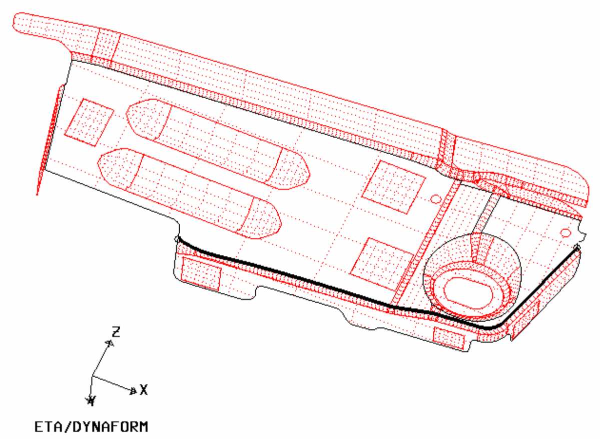


Figure 6.1.4 Unfold Line and Boundary Line

- 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, a dialog window as shown in Figure 6.1.5 will be displayed to allow the user to change the gap tolerance between surfaces.

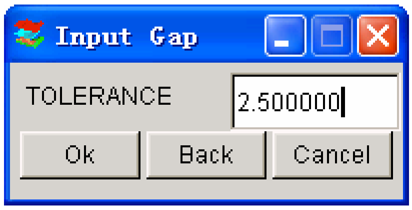


Figure 6.1.5 Gap Tolerance

Default gap tolerance is 2.5, which is adequate in most cases. 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. Base surface is the surface that connects the flange surfaces along the unfold line.

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

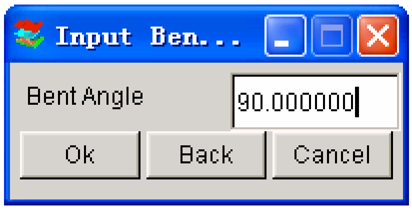


Figure 6.1.6 Define Bent Angle

- **OK** accepts the entered bent angle. The unfolded surfaces are shown in Figure 6.1.7 in yellow.

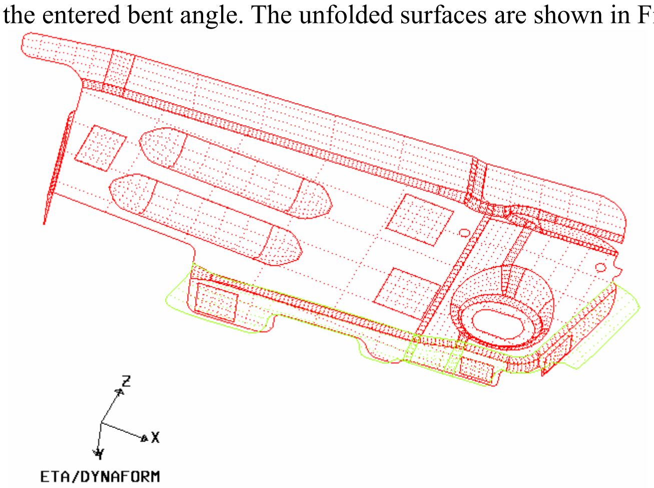


Figure 6.1.7 Unfolded Flange

The unfolded surfaces are included in a new part called UNFOLDED.

- Select an option from the CONTROL KEYS window as shown in Figure 6.1.8.

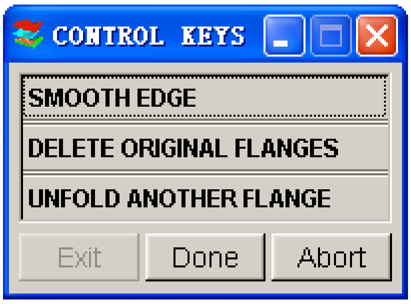


Figure 6.1.8 Control Keys Window

SMOOTH EDGE

This function allows the user to smooth the edge of unfolded surfaces. Eta/DYNAFORM will prompt the user to select the starting point and end point on the edge of the unfolded flange. The smoothed edge will be highlighted as shown in Figure 6.1.9. The program will display another CONTROL KEYS window as shown in Figure 6.1.10.

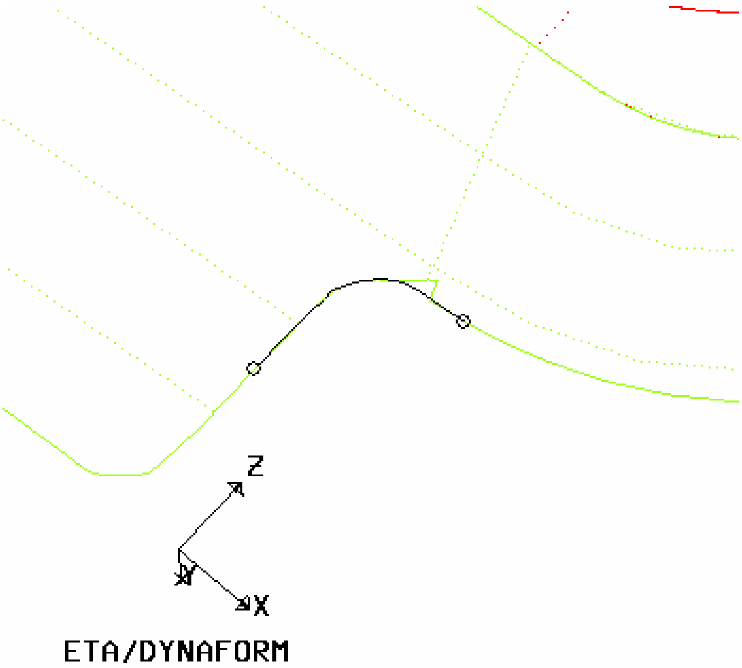


Figure 6.1.9 Smooth Edge

Figure 6.1.10 Control Keys window

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 deleted.

UNFOLD ANOTHER FLANGE

Continues to unfold another flange from the first step.

6.1.2 SYMMETRY

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

The DIE needs to be defined in the database in order to define the symmetry plane. If there is no part defined for the die, a DYNAFORM QUESTION window as shown in Figure 6.1.11 will appear.

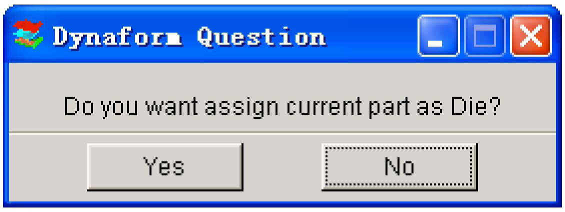


Figure 6.1.11 DYNAFORM QUESTION window

Select **YES**, and the current part will be assigned to the die and the function will continue. Selecting **NO** will exit the function. See Figure 6.1.12.

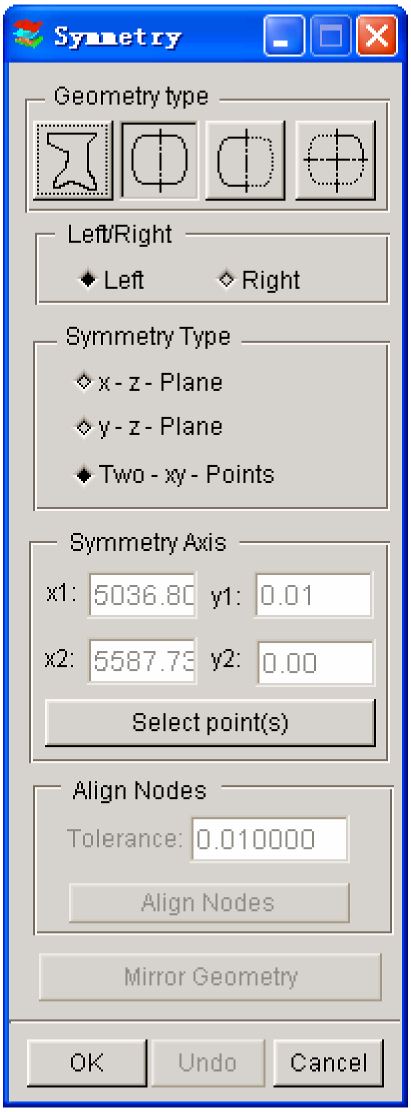
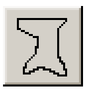


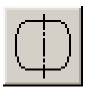
Figure 6.1.12 Symmetry window

Eta/DYNAFORM52 supports the following four types of part geometry.



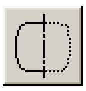
NO SYMMETRY (Default)

This option is selected as the default. If the function is toggled **ON**, other functions in Figure 6.1.12 will be disabled.



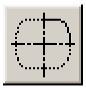
SYMMETRY

The part is defined as symmetrical.



HALF SYMMETRY INPUT

Only one half of the part geometry is available. The program will mirror the half part about the symmetry plane to form the complete part.

 **QUARTER SYMMETRY INPUT**

Only one quarter of the part geometry is available. The program will mirror the quarter part about both symmetry planes to form the complete part.

There are three types of symmetry that can be defined by the SELECT POINTS button.

X-Z PLANE

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

Y-Z PLANE

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

TWO-XY-POINT

The symmetry plane is normal to the XY plane of the Global System and through the selected two points. The user must select two points. The first point defines the location of the symmetry plane, and the second point determines the direction of the symmetry axis.

If the nodes at the plane of symmetry are not perfectly collinear along the symmetry plane, use **ALIGN NODES** function to correct the problem. When the part is defined as Half Symmetry or Quarter Symmetry, DYNAFORM will activate the ALIGN NODES function. The user can define the tolerance in the dialog window to align the nodes along the line of symmetry. The program will show the result of the align nodes operation and prompt the user for acceptance.

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). The other symmetry axis is vertical to the first plane and through the second symmetry axis.

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

Follow the below steps to design the symmetrical die.

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

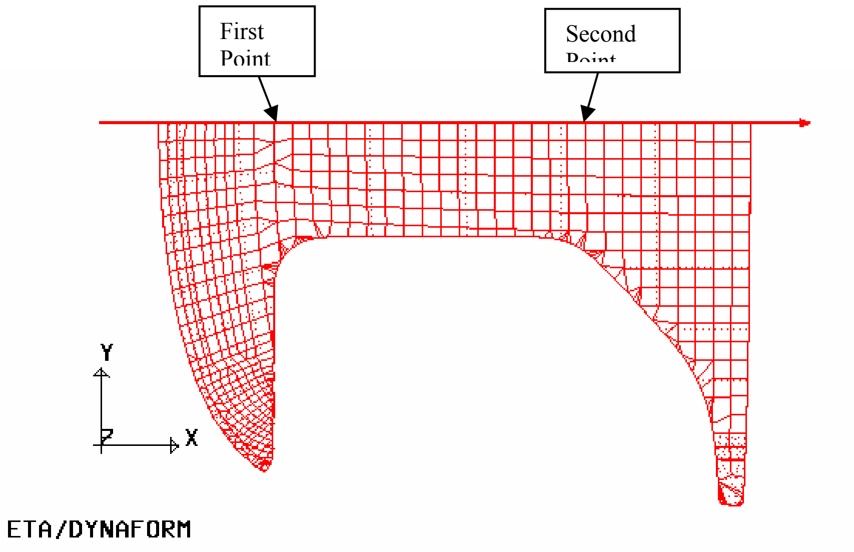


Figure 6.1.13 Half Symmetry Part With X-Z –Plane

5. Click **MIRROR GEOMETRY SYMMETRY**, then select the part as shown in Figure 6.1.13. The result is shown in Figure 6.1.14.

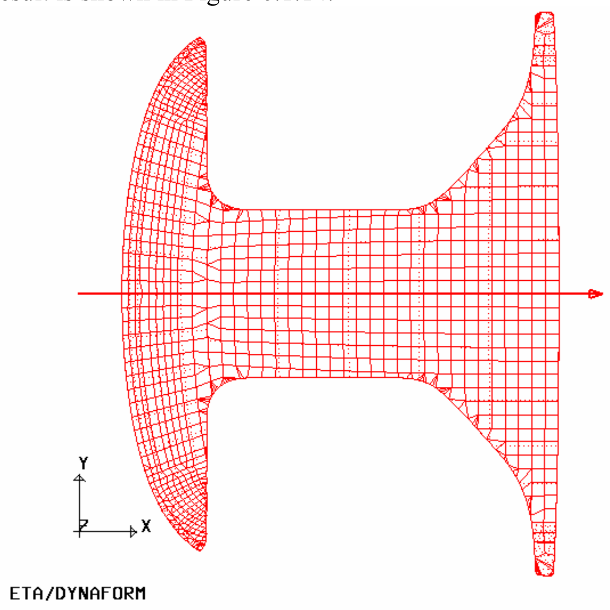


Figure 6.1.14 The Result of Mirroring

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

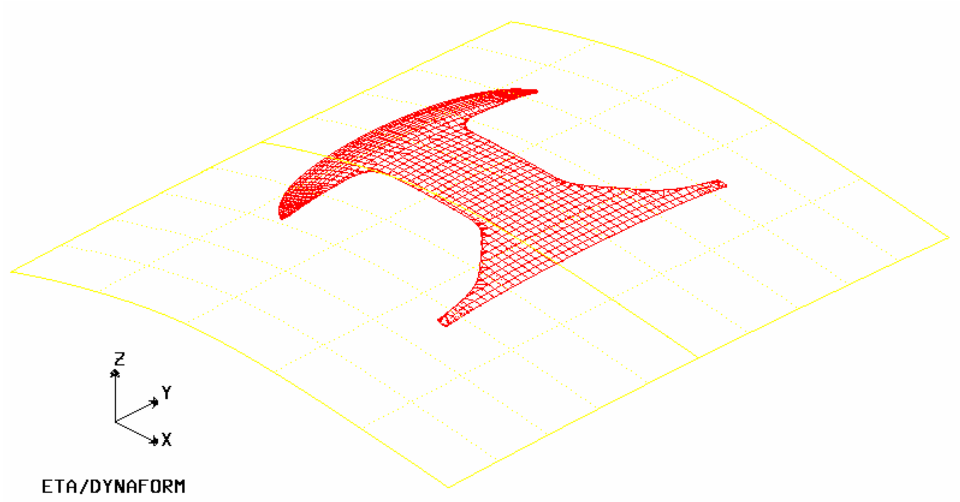


Figure 6.1.15 Binder With Symmetry Plane

6.1.3 TOOL MESH

This function is the same as the function PREPROCESS/ELEMENT/SURFACE MESH as described in the Section 5.3.4, SURFACE MESH.

6.1.4 FILLET MESH

The FILLET MESH function is used to generate fillet elements at the sharp corners of a part. This function requires a relatively clean mesh without overlap elements or interior gaps. The element normal must be consistent in the mesh. The FILLET MESH dialog window, Figure 6.1.16, provides options to find and fillet shape corners.

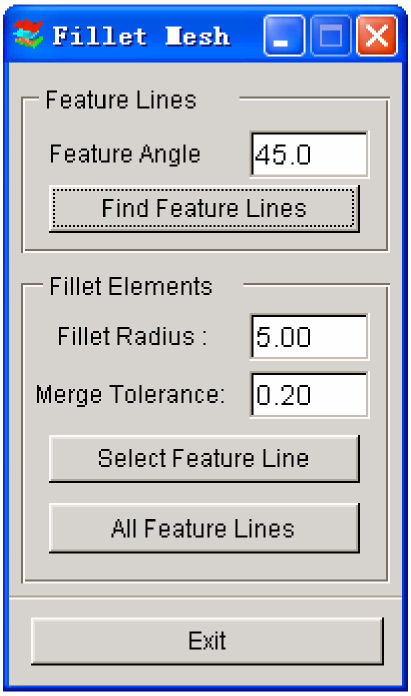


Figure 6.1.16 Element Fillet dialog window

The following two steps are required to generate the fillet mesh.

- **FEATURE ANGLE**

  1. Enter the angle criteria (the default is 45 degrees).
  2. Click the **FIND FEATURE LINES** button to find the feature lines where the angle is larger than or equal to the angle criteria. The program will highlight all features lines that meet the criteria.

- **FILLET ELEMENT**

  1. **FILLET RADIUS** defines the fillet radius for all feature edges.
  2. **MERGE TOLERANCE** defines the gap tolerance between nodes on the fillet radius. Nearby nodes will be merged if the distance is less than the defined Merge Tolerance.
  3. If **SELECT FEATURE LINE** is clicked, the program will display the SELECT NODE window for the user to select two or three nodes on the feature line. If two nodes are selected on the two-face feature line, the program will display a dialog window as shown in Figure 6.1.19. If three nodes on the three-face feature line are selected, the program will display a dialog window as shown in Figure 6.1.20a.

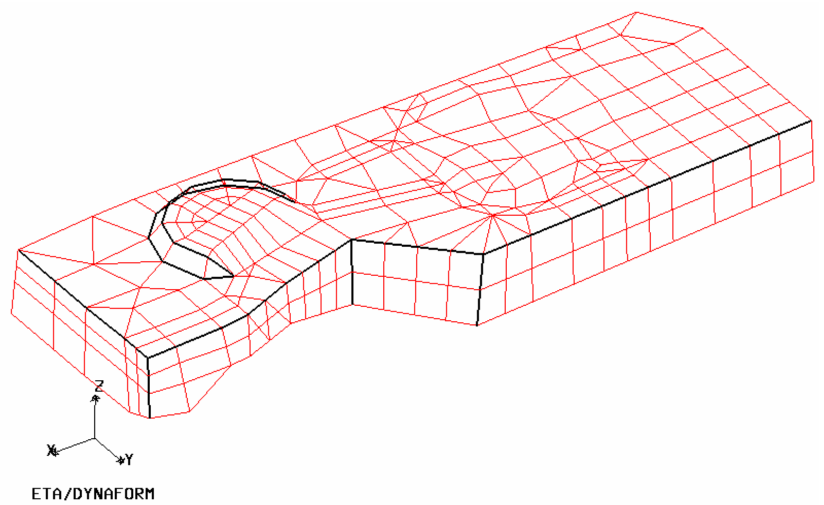


Figure 6.1.17 Find Feature Lines

  4. **ALL FEATURE LINES**
     When **ALL FEATURE LINES** is clicked, all feature lines will be filleted with the global radius. Press **OK** on the PARAMETERS window to generate the fillet mesh. The Fillet function supports different radii and automatically smooths the elements.

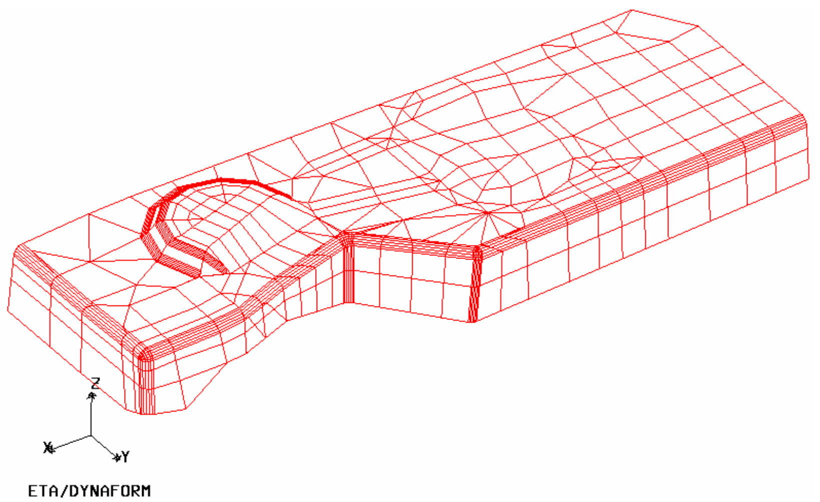


Figure 6.1.18 Fillet with All Feature Lines

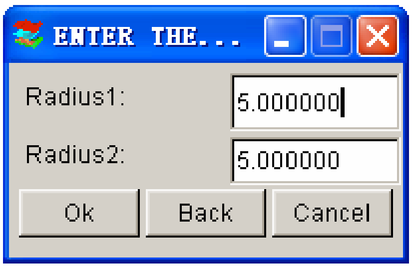


Figure 6.1.19 a Element Fillet

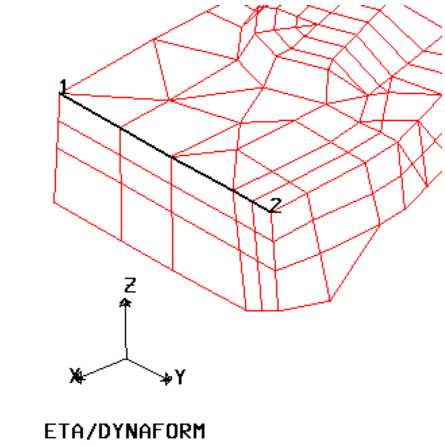


Figure 6.1.19 b Select feature line

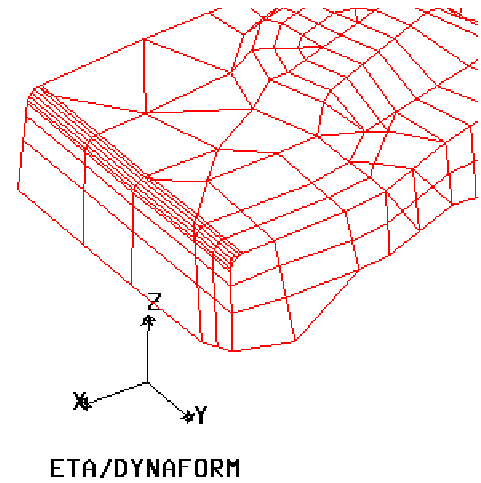


Figure 6.1.19c Element Fillet Result

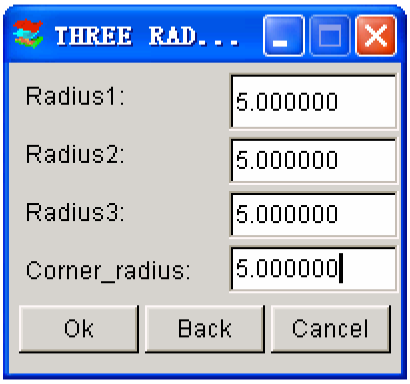


Figure 6.1.20a Element Fillet

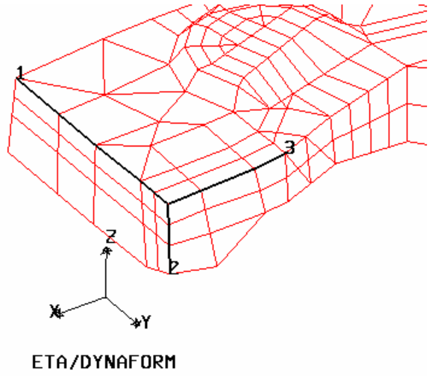


Figure 6.1.20b Select feature line

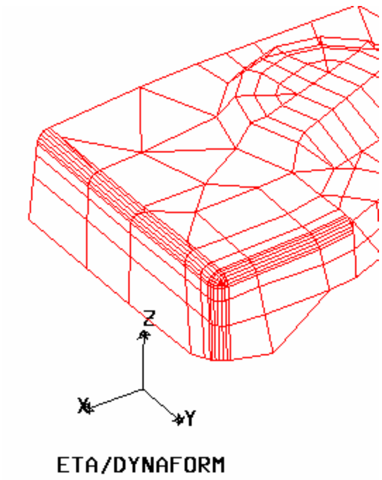


Figure 6.1 20c Element Fillet Result

6.1.5 INNER FILL

The functions in the INNER FILL are used for filling interior holes and certain missing geometry on the outer boundary of a product geometry, such as windows, wheel house, tail light lamp, shipping holes, access holes, bolt holes, etc. There are four functions in this menu as shown in Figure 6.1.21.

1. CLOSED CURVE (or close area)
2. OPEN CURVE (or open area)
3. POLYGON
4. AUTO FILL

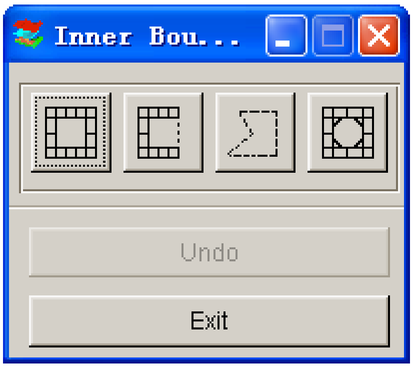


Figure 6.1.21 Inner Fill Options

6.1.5.1 CLOSED CURVE (Fills in a closed area)

This function fills an inner hole or missing area in a part. The program highlights the inner boundary as shown in Figure 6.1.22.

Figure 6.1.22 Boundary lines of Inner holes

At the same time, DYNAFORM displays a dialog window as shown in Figure 6.1.23 to prompt the user to select a node on an inner boundary to fill. After selecting a node on the boundary, the interior hole will be filled automatically with elements as shown in Figure 6.1.24.

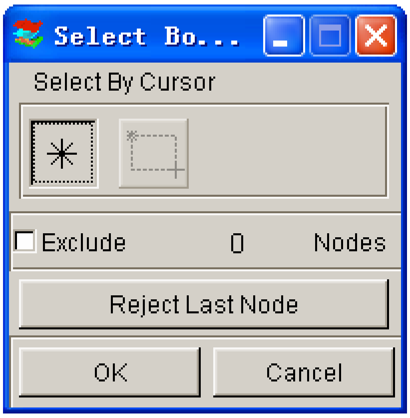


Figure 6.1.23 Select a Node on Inner Boundary

Figure 6.1.24 Filled Mesh with Closed Curve

6.1.5.2 OPEN CURVE (Fills in an open area)

This function allows the user to fill in elements in a cut out area. Figure 6.1.25 shows the result of filled mesh using a spline curve to close the opening.

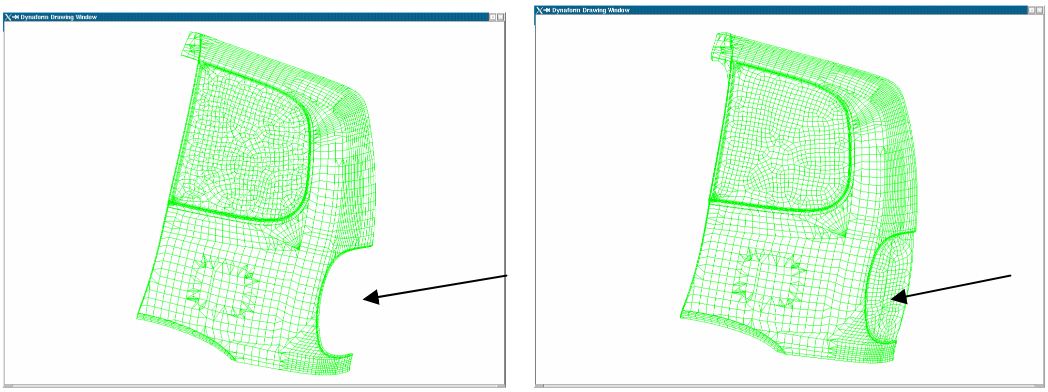


Figure 6.1.25 Filled mesh with Open Curve

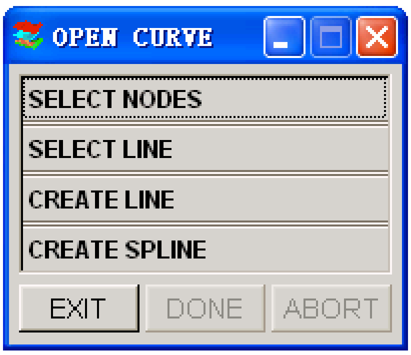


Figure 6.1.26 Open Curve Options

1. Select the **OPEN CURVE** option, and the program will prompt the user to select elements near the expected fill area.
2. Select **OK** to accept the selected elements, and a window will be displayed, as shown in Figure 6.1.26, to prompt the user to select one of the following methods to fill the area.

   SELECT NODES

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

   SELECT LINE

   Allows the user to select an existing line as the outer boundary to close the open curve for filling. After a line is selected, the program displays the SELECT NODES window and prompts the user.

   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 window will be displayed.

   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

   Allows the user to create a spline curve as the outer boundary for fill mesh. After the spline is created, the program will display the SELECT NODES window and prompt the user with the following message.

   SELECT TWO NODES TO SPLIT THE LINE, OR PICK OK TO USE THE ENTIRE LINE
   After selecting two nodes, the spline segment between the two points will be used

3. After the curve is defined by any of the above methods to close the region, the program will fill in a set of elements within the region.

6.1.5.3 POLYGON

This function allows the user to select a set of nodes as the boundary edge to create a mesh to fill the boundary. Once this function is selected, the program will display the SELECT NODE window and prompt the user with the following message.

SELECT A SET OF NODES CONSECUTIVELY AS BOUNDARY EDGE
After a set of nodes is selected, a set of elements is created to fill the inner boundary.

6.1.5.4 AUTO FILL

Once selected, this function automatically finds and fills all inner holes in the part with elements.

Eta/DYNAFORM will create elements as well as surface in all filled areas. The user may choose whether or not to display the surfaces. This option is controlled in the DYNAFORM configuration file: DYNAFORMDEFAULT, located in the program installation directory.

Note: Click UNDO on the OPTION window as in Figure 6.1.21. The filled mesh and surface will be removed.

6.1.6 MODEL CHECK AND MESH REPAIR

All functions in the MODEL CHECK menu and MESH REPAIR menu are the same as in Sections 5.5 and 5.6. Refer to the previous sections in this manual for a detailed description of these functions.

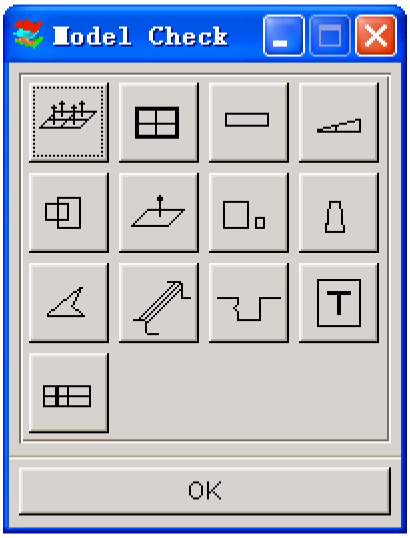


Figure 6.1.27 Model Check window

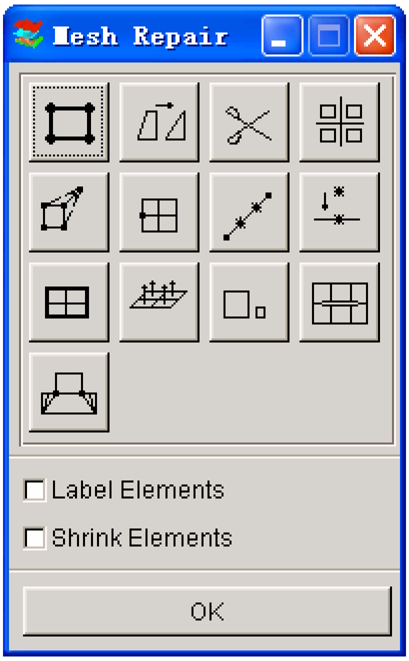


Figure 6.1.28 Mesh Repair window

6.1.7 TIPPING

In most stamping simulations, the global Z direction is used as the stamping direction. Some of the part design might not be compatible with DYNAFORM's setting. The tipping function is used to orient the part in the stamping direction. It allows the user to translate and/or rotate the part about any axis. The options in the tipping function are shown in Figure 6.1.29.

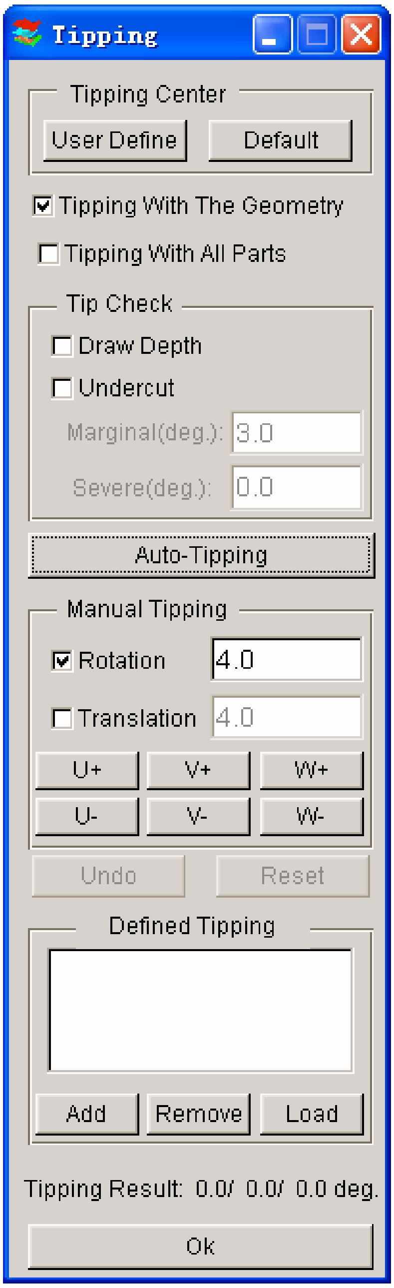


Figure 6.1.29 Tipping

Die must be defined before tipping. The program will display a DYNAFORM QUESTION window, Figure 6.1.30, to prompt the user.

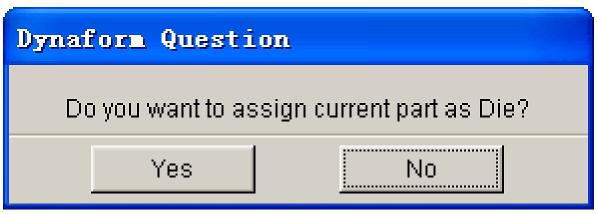


Figure 6.1.30 DYNAFORM QUESTION window

YES will assign the current part to the die, and **NO** will exit the function.

6.1.7.1 Tipping Center

This function is used to define the Tipping Center.

- The default center is located on the mass center of the die.
- User Define used to define a new center according the request. Click the Button USER DEFINE, it opens a node/point select dialog. User can select node or point as tipping center or input the coordinate at input box and click Apply Input Value to define the tipping center.

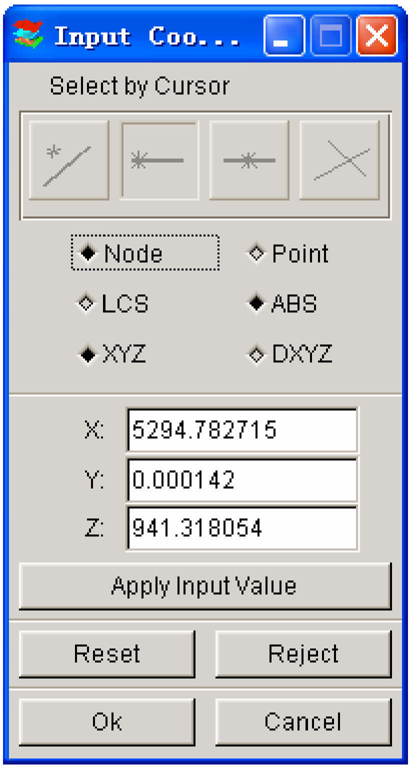


Figure 6.1.31 tipping center select window

6.1.7.2 TIPPING WITH THE GEOMETRY (TOGGLE)

If this option is toggled **ON**, DYNAFORM will tip all the parts defined as DIE. Geometry (surface, line) in the DIE will be rotated and translated with mesh of die tool.

6.1.7.3 TIPPING WITH All PARTS (TOGGLE)

If this option is toggled **ON**, DYNAFORM will tip all the parts in the project. Geometry (surface, line) and Mesh in the project will be rotated and translated with mesh of die tool.

6.1.7.4 TIPPING CHECK

After the Auto and Manual Tipping operation, it is recommended to use the UNDER CUT and DRAW DEPTH functions to check the tipping result.

- DRAW DEPTH (toggle)
  A contour plot shows the draw depth relative to the binder on the DIE. Different draw depth ranges are displayed with different colors on the DIE. Draw Depth is estimated with reference to the Point of the First Contact. When the Die/Punch is closed, the first point on the blank in contact with the Punch is locked. The rest of the part will be stretched from this point.
- UNDERCUT (toggle)

  A contour plot will show the region in the DIE where under cut occurs. A red region indicates severe under cut of which the backdraft is less than zero degrees. A blue region indicates a marginal area with an approximately 1~3 degree draft angle. A green region indicates a safe region of which the drawing angle is greater than 3 degrees. User can set the thresholds of the Marginal and Severe degree.

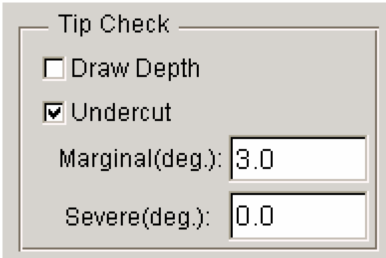
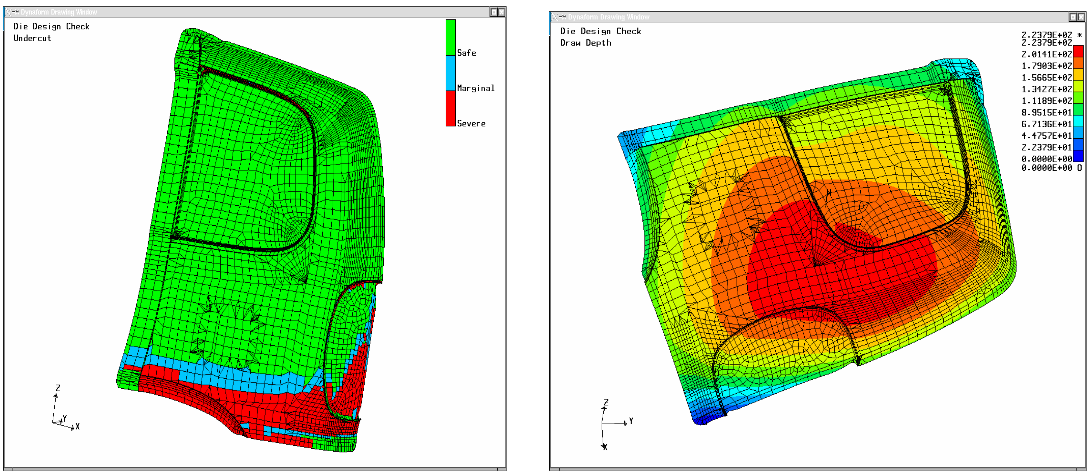


Figure 6.1.32 Tip Check options

6.1.7.5 AUTO-TIPPING

This function automatically converts the die to stamping position by averaging all normal direction vectors of elements to minimize the undercut and draw depth. In addition to the automatic tipping function, the user can tip the part around the U-,V-,W-axis to a more useful draw direction. See Figure 6.1.33.



Before Tipping　　After Tipping

Figure 6.1.33 Tipping Check

Note: eta/DYNAFORM will automatically calculate the Tipping center. There is a default tipping Local Coordinate System at the tip center. The UVW direction is consistent with XYZ of the Global Coordinate System. The W-axis is always the drawing direction. The functions on the Tipping page rotate the part in a way so that the W-axis will be the drawing direction.

6.1.7.6 MANUAL TIPPING

Auto Tipping might not be satisfactory due to an unbalanced mesh pattern or part shape. A manual tipping function is provided for the user to further tip the DIE. The DIE can be rotated or translated on the U-, V-, or W-axis. Using the U+/-, V+/-, W+/- buttons performs the tipping operation according to the increment defined in the dialog window. The tipping check will be visible if the **UNDER CUT** or **DRAW DEPTH** button is toggled **ON.**

- ROTATE

  Select **ROTATE**, and enter a value for rotate angle in the data window. Click on the U+(-), V+(-), and W+(-) buttons that represent rotate axis, and the model will be rotated by that angle about the selected axis.

- TRANSLATION

  If **TRANSLATION** is selected, the U+(-), V+(-), W+(-) buttons represent translation direction. The value in the TRANSLATION DATA window is the incremental displacement along the selected axis.

6.1.7.7 UNDO AND RESET TIPPING OPERATION

Undo will cancel the last tipping operation
Reset will cancel all tipping operations.

6.1.7.8 DEFINED TIPPING

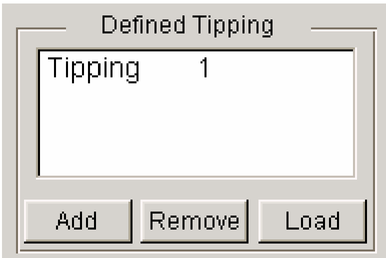


Figure 6.1.34 Defined Tipping

This function in tipping menu is used to record and restore the tipping operation.
Add is used to record the Tipping result;
Remove is used to remove one record;
Load is used to restore the tipping result to the select record.

6.1.8 OUTER SMOOTH

The functions in the OUTER SMOOTH menu are used to smooth the outer boundary of a part. There are four functions in this menu as shown in Figure 6.1.35.

1. ROLLER
2. MORPHING
3. CORNER SMOOTH
4. EXPAND

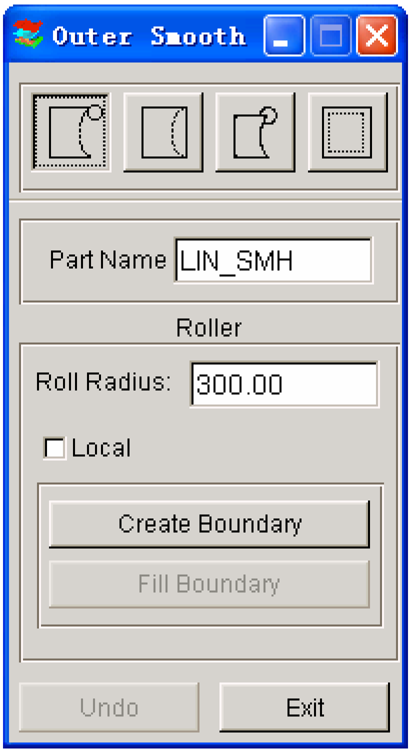


Figure 6.1.35 Outer Smooth Options

6.1.8.1 ROLLER

This function allows the user to roll a cylinder with a defined radius along the boundary and fill the gap region between the part boundary and the cylinder.

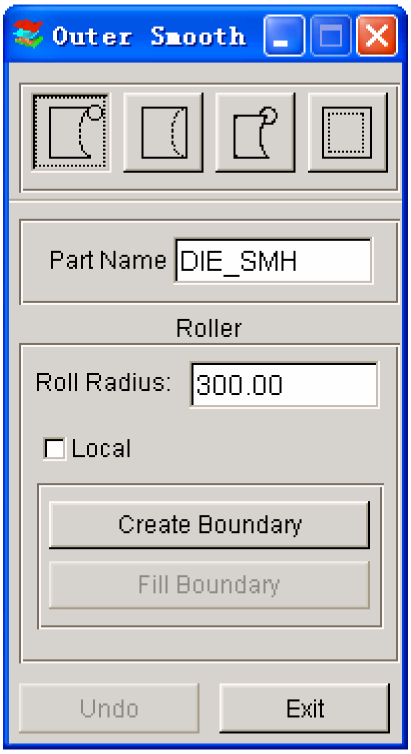


Figure 6.1.36 Roller

Once this function is selected, a dialog window as shown in Figure 6.1.36 will be displayed. The default radius of the cylinder is 300 in the ROLL RADIUS window.

1. After the desired radius is entered, click the **CREATE BOUNDARY** button to show the new boundary line with the cylinder rolled along the part boundary. It the result is acceptable, click the **FILL BOUNDARY** button to fill in the mesh in gap region. A new part will be created to include the filled mesh and will be automatically added to the DIE.

2. **UNDO** will remove the filled mesh in the gap region and allow the user to go back to the original part. Figure 6.1.37 shows the result of a typical roller operation.

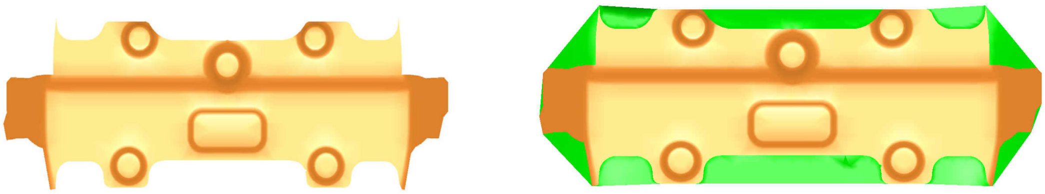


Figure 6.1.37 Roller Example

3. Click the **CLOSE** button, and the function exits.

If toggles on the **Local** option, it allows user to roll a cylinder with a defined radius along the selected boundary and fill the gap region between the part boundary and the cylinder path line. So it can smooth the local boundary freely.

Click the **Roller** button and toggle on the Local option. The default radius of the cylinder is 300 in the ROLL RADIUS window. The user can input the desired **Part Name** and the **Roll Radius.**

1. After the desired radius is entered, click the **CREATE BOUNDARY** button to show the highlighted part boundary line. In message windows the program prompts the user to select first point on boundary. Select a desired point on boundary and then the second point. Then the program will prompt the user to press middle button to apply this operation.

2. Press middle button and a smoother virtual boundary will be highlight than original boundary. If the result is acceptable, click the **FILL BOUNDARY** button to fill in the meshes in gap region. The program will create a new part with the given name and include the filled meshes in it. Moreover, the part will be automatically added to the DIE. Fig. 6.1.38 shows the result of a typical roller operation with Local option on.

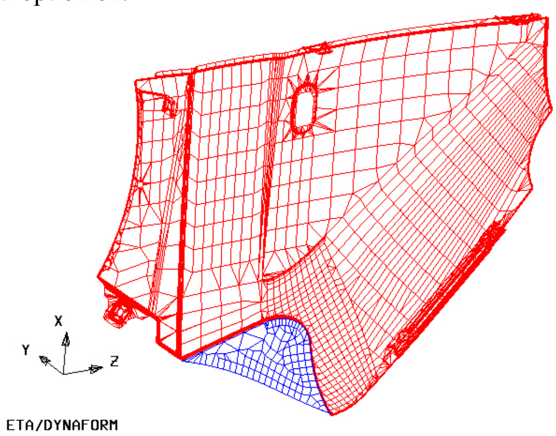


Fig. 6.1.38 Outer Smooth/Roller with Local option

6.1.8.2 MORPHING

This function allows the user to modify the part mesh locally by morphing the boundary line of the part. Figure 6.1.39 shows the dialog window of the MORPHING function.

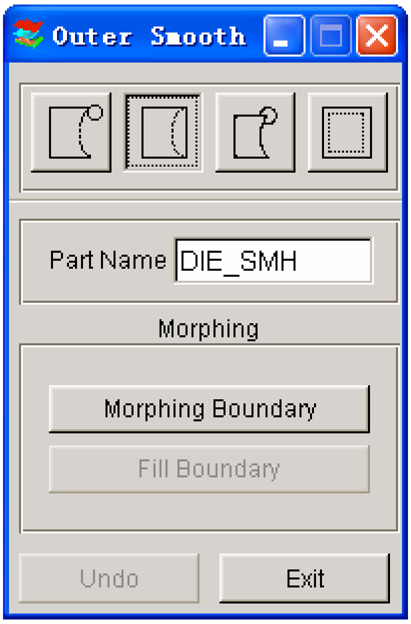


Figure 6.1.39 Morphing

1. Click the **MORPHING BOUNDARY** button, and the boundary of the Die will be highlighted.
   The program will prompt the user to select two points on the part boundary to define the morphing region. Figure 6.1.40 shows the selected region on the part boundary.

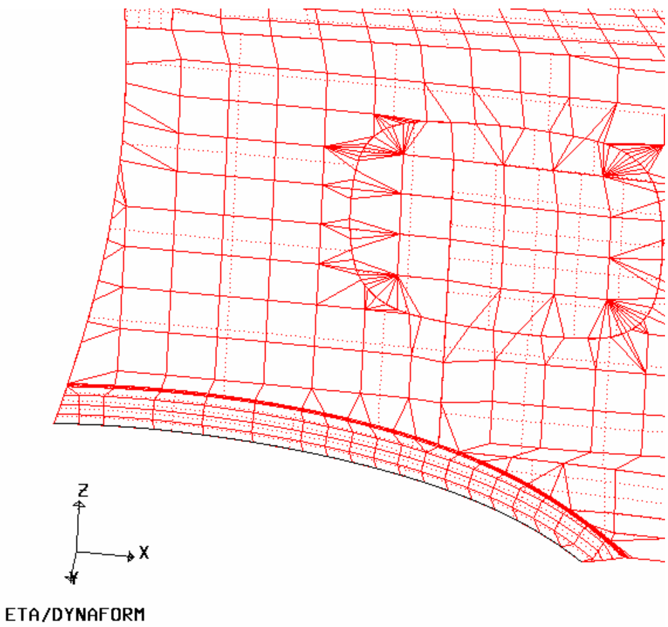


Figure 6.1.40 Select Morphing Boundary

2. After two points are selected, a dialog window will be displayed as in Figure 6.1.36. There are three options in this window to allow the user to morph and smooth the boundary line. Figure 6.1.41 shows the smoothed boundary.

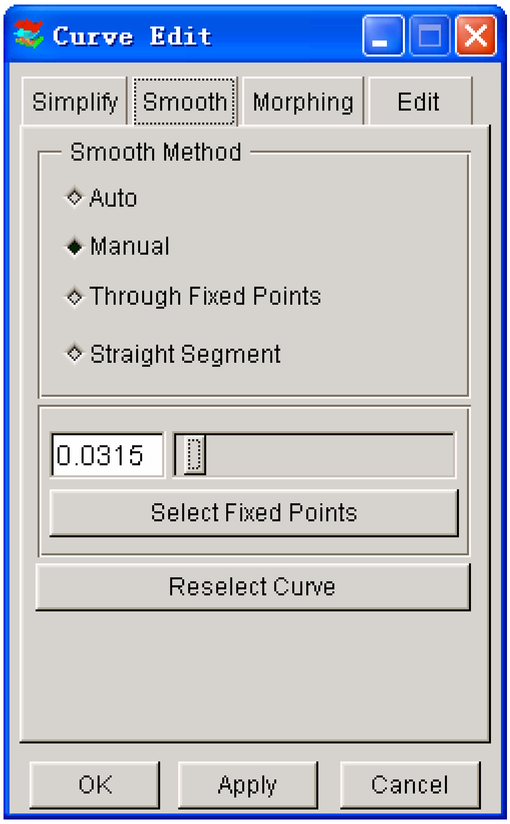


Figure 6.1.41 Curve Edit

3. Click **FILL BOUNDARY** to fill in the mesh between the original boundary and the smoothed boundary. The result is shown in Figure 6.1.42.

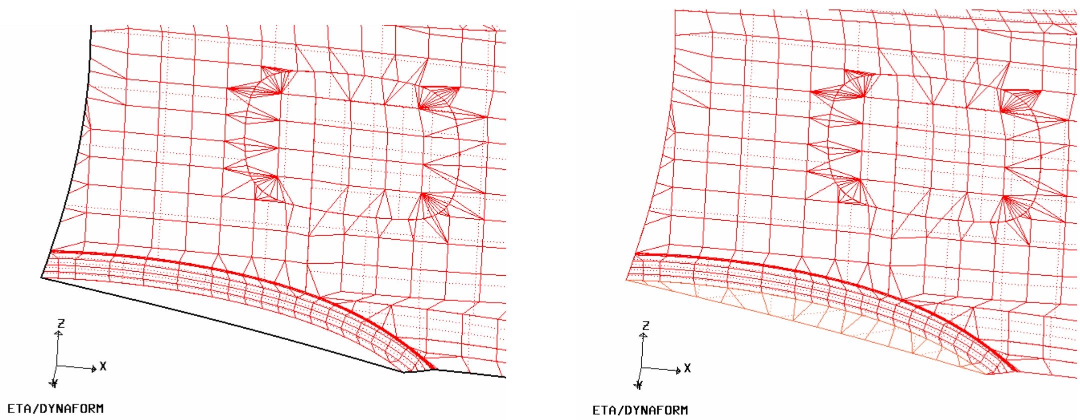


Figure 6.1.42 Morphing Result

4. **UNDO** will remove the mesh from the gap region, allow the user to go back to the original boundary and put the newly created part in IGS_SMH. **PART NAME** enables the user to name the part; it will be automatically assigned to Tool.

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

6.1.8.3 CORNER SMOOTH

This function allows the user to smooth out the sharp angle of a part with a defined radius. The program will generate a set of elements in a circular shape with the defined radius around the selected point.

When this function is selected, the program displays a window as shown in Figure 6.1.43. The default corner radius is 30 as displayed in the CORNER RADIUS window.

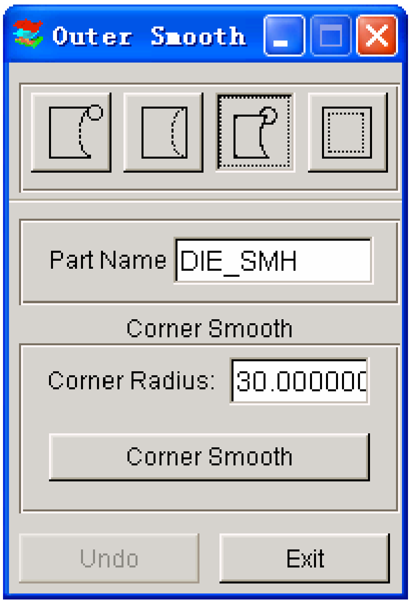


Figure 6.1.43 Corner Smooth

1. After entering the desired corner radius, click the **CORNER SMOOTH** button, and the program will highlight the boundary line of the part as shown in Figure 6.1.44.

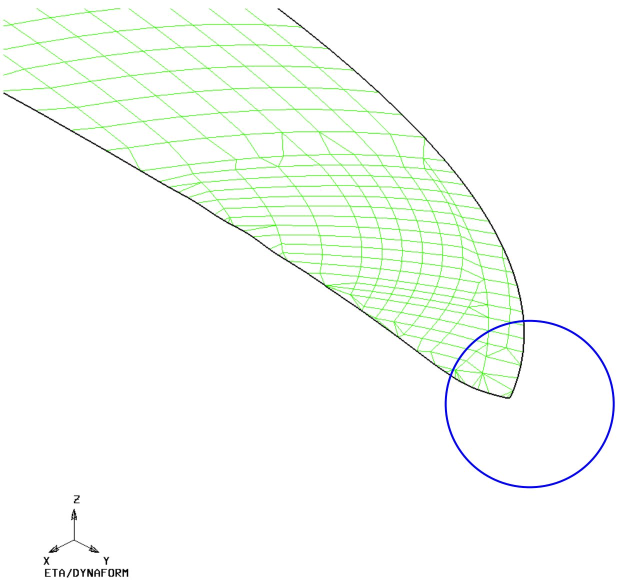


Figure 6.1.44 Sharp Corner

2. The program prompts the user to select a node on the boundary for corner smooth operation:
   SELECT A NODE FOR CORNER SMOOTH
3. After a node is selected at a sharp corner, the program will generate a set of elements around the part boundary to form a circular shape around the corner. Figure 6.1.45 shows the result of a corner smooth operation.

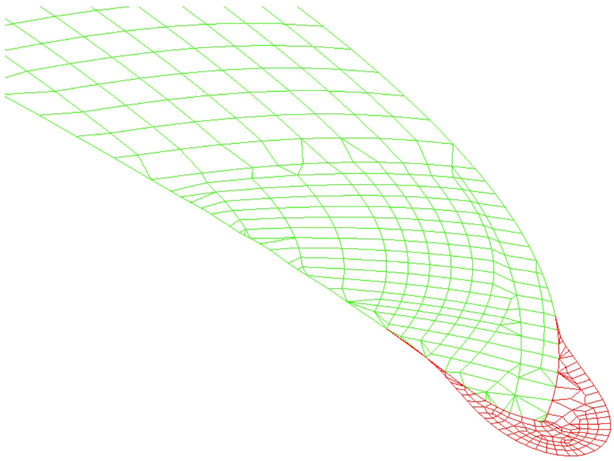


Figure 6.1.45 Corner Smooth

4. **UNDO** will remove the mesh from the corner smooth to allow the user to go back to the original boundary.
5. Click the **CLOSE** button to exit the function.

6.1.8.4 EXPAND

The Expand function will add one or two rows of elements to the part boundary to round off the outer edge with formable radii of approximately 3mm. When this function is selected, a dialog window as shown in Figure 6.1.46 will be displayed. The value in the extension field is the default extension value for the current part. The user can enter a desired extension value.

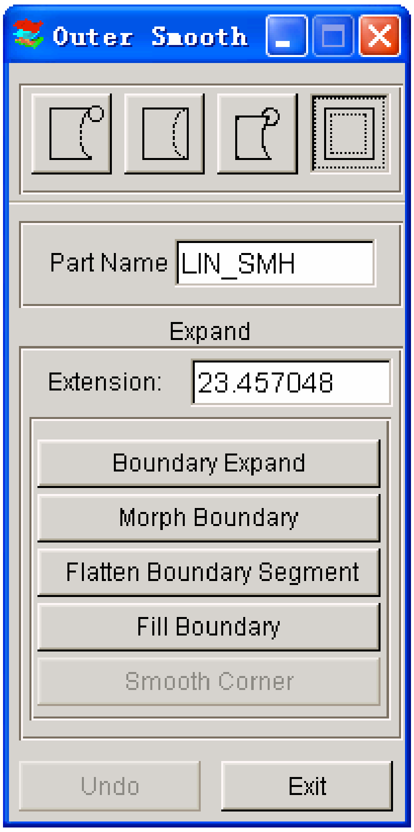


Figure 6.1.46 Boundary Expand

1. Click the **BOUNDARY EXPAND** button, and the extension boundary is highlighted as shown in the left picture of Figure 6.1.47. The default extension value is calculated from the average element size. The user can enter another extension value and click **BOUNDARY EXPAND** again.

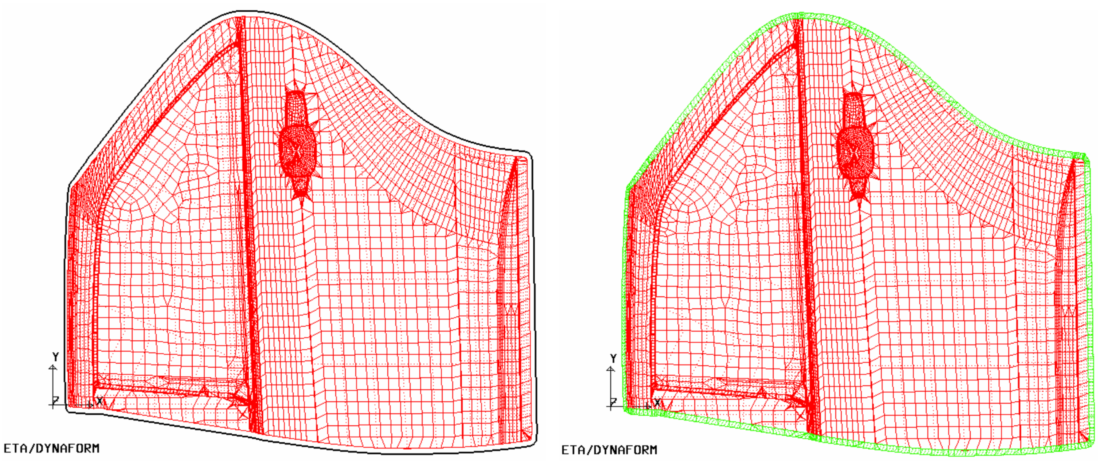


Figure 6.1.47 An Example of Boundary Expand

2. Select the **MORPH BOUNDARY** button to morph the boundary line. Refer to the description of the **MORPHING LINE** function in Section 6.4.1.

3. The **FLATTEN BOUNDARY SEGMENT** function allows user to select two points on the boundary line to flatten the line segment between the selected points.

4. When the morphed boundary is accepted, select **FILL BOUNDARY** to generate the mesh between the original boundary and the extension boundary. See the right picture in Figure 6.1.38 for the filled mesh.

5. **SMOOTH CORNER** allows the user to select a sharp corner node to smooth out the corner.

Note: The sharp angle is determined by two adjacent boundary elements when the angle between two boundary edges is greater than 30 degrees.

6. Click **UNDO** to remove the filling mesh and exit the function.

6.2 BINDER

The functions in the BINDER menu are used to generate various types of binders. Figure 6.2 shows the types of binder available in the BINDER menu.

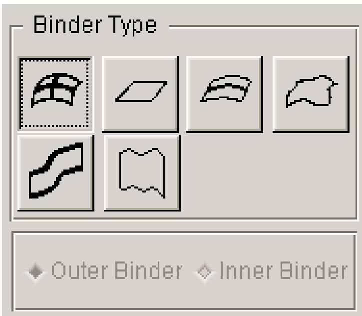


Figure 6.2 Binder Types

Outer Binder or Inner Binder is associated with Outer or Inner Addendum, and there is only one outer binder / addendum pair exists in project and several inner binder / addendum pairs in project if necessary.

1. TWO-LINE BINDER

   Uses two orthogonal cross lines to define the binder shape.

2. FLAT BINDER

   Uses a flat rectangular surface to define the binder.

3. CONICAL BINDER

   Uses a conical surface with two radii to define the binder shape.

4. BOUNDARY LINE BINDER
   Uses the part boundary line or user-selected/created line to define the binder.

5. FLANGE BINDER
   Uses the flanges on the part to define the binder shape.
6. FREE FORM BINDER

   Uses the binder section lines from clicking locations (digitizing) on the screen to define the binder.

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

6.2.1 TWO-LINE BINDER

The TWO-LINE BINDER function is used to create a binder with curvatures in both the X and Y directions. The dialog window is shown in Figure 6.2.1.

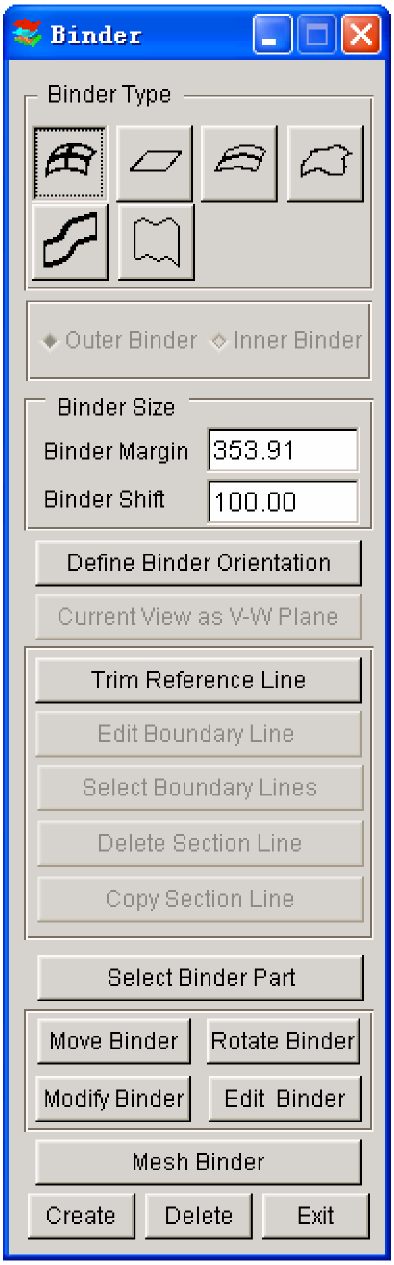


Figure 6.2.1 Two-line Binder

BINDER MARGIN and BINDER SHIFT parameters are used to control the binder size and position relative to the part.

- **BINDER MARGIN**

  BINDER MARGIN allows user to expand the binder from the original binder. The original binder is calculated by the part boundary. The value of the BINDER MARGIN is the distance between the boundary of the original binder and the expanded binder.

- **BINDER SHIFT**

  BINDER SHIFT is used to adjust the vertical distance between the part and the binder. If the value of the BINDER SHIFT is positive, the binder will be shifted along the (-) Z axis direction. A negative value will shift the binder along the (+) Z axis. BINDER SHIFT directly affects the draw depth of a part.

- **DEFINE BINDER ORIENTATION**

  DEFINE BINDER ORIENTATION controls the U-V direction of the binder surface. TRIM REFERENCE LINE adjusts the length of the Reference line, which controls the binder shape.

  When the button is clicked, the program displays the part in the X-Y view. The program will highlight the Point of First Contact with a white circle and prompt a message in the MESSAGE window.

  CLICK AND HOLD THE LEFT MOUSE BUTTON AT THE CROSS HAIR CENTER. MOVE THE MOUSE TO ROTATE. RELEASE THE MOUSE BUTTON WHEN DONE

  The program draws two orthogonal cross lines on the binder as shown in Figure 6.2.2. The line with an arrow head indicates that it is the U-axis of the binder surface. Click, hold and move the mouse to obtain a desirable orientation of the U-V axes.

- **TRIM REFERENCE LINE**

  The created binder is affected by the reference point, the orientation of the U-V axes, the length of the U-V axes, and the part geometry. If some of the details on the part are not acceptable for the binder shape, the user can trim the reference line to adjust the curvature of the binder. Refer to Figure 6.2.3, Figure 6.2.4 and Figure 6.2.5 for the binder created by different lengths of U-V axes.

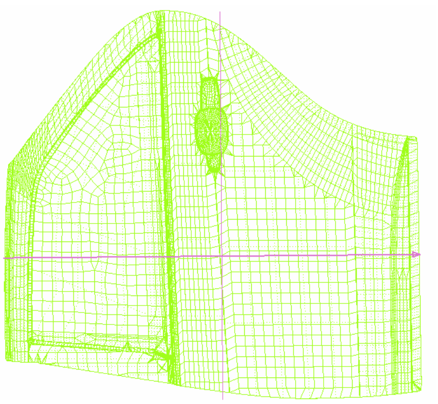


Figure 6.2.2 Two Control Lines

Figure 6.2.3 Two Line binder (before trimming)

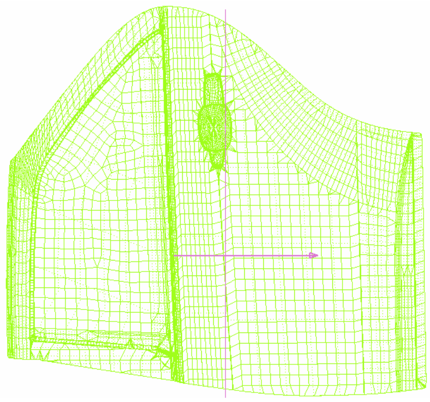


Figure 6.2.4 Trim Reference Line

Figure 6.2.5 Two Line Binder From Trimmed Reference Line

To create a binder follows these steps.

1. Enter the desired **BINDER MARGIN** and **BINDER SHIFT.**
2. Define the proper orientation of the U-V direction of the binder surface.
3. Adjust the binder shape by using the **TRIM REFERENCE LINE**.
4. Click the **CREAT**E button at the bottom of the window. The program will create the binder surface and put it in a new part called C_BINDER that will be automatically assigned to TOOL (Binder). Refer to Section 9.1 for detailed explanation on the TOOL.
5. If the binder has already been created in C_BINDER part, DYNAFORM will display a QUESTION window as in Figure 6.2.6.

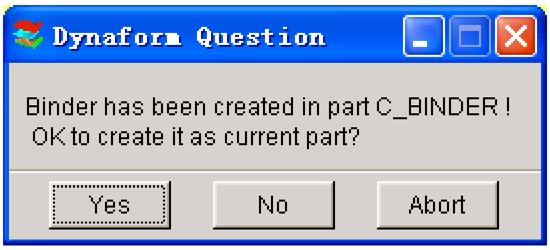


Figure 6.2.6 DYNAFORM QUESTION window

YES - Recreate the binder with the current definition.

NO - Keep the old binder.

- **SELECT BINDER PART**
  When there is more than one binder on the database and the user wants to modify the specified binder, the user can click the SELECT BINDER PART button and the Select Binder window as shown in Figure 6.2.7 will pop up and prompt the user to select the desired binder part:

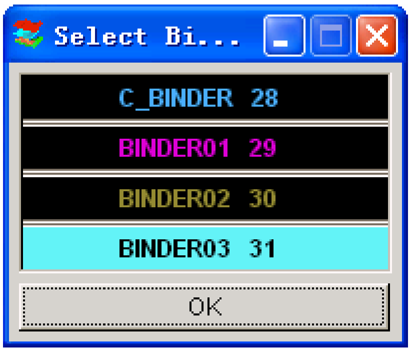


Figure 7.2.7 Select Binder window

Once one binder is created, the editing functions are activated for the user. The editing functions are used to modify the binder as shown in the lower part of Figure 6.2.8.

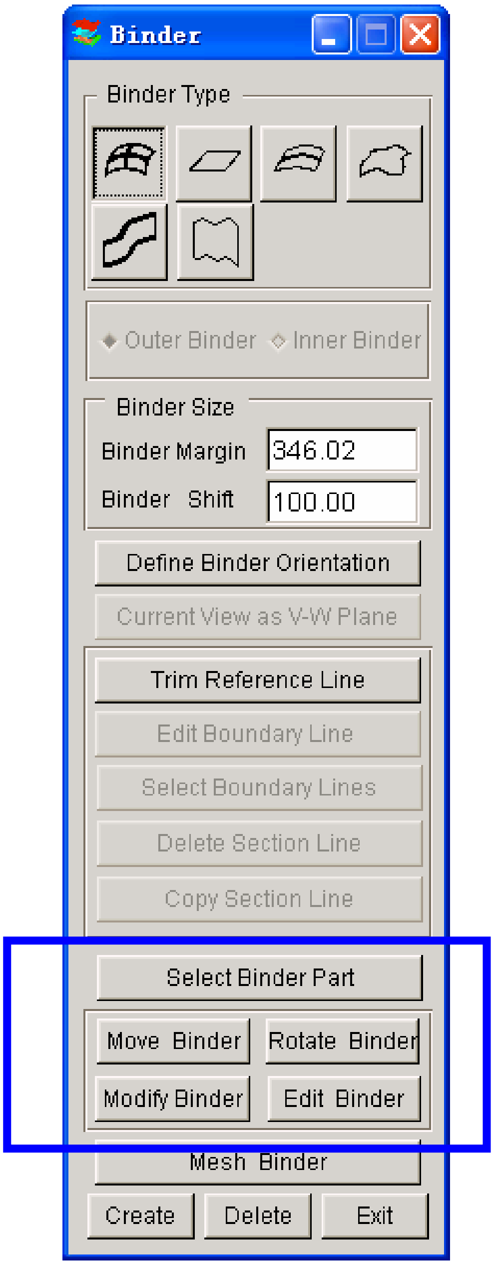


Figure 6.2.8 Binder Adjustment Options

The description for each editing function follows.

- **MOVE BINDER**

  This function allows the user to adjust the relative distance between the binder and the part. The relative distance will affect the draw depth of the part. Click the **MOVE BINDER** button, and the program will display a dialog window as shown in Figure 6.2.9.

  There are two methods to move the binder.

  1. Select **MOVE DIRECTION** U,V or W and enter the move vector directly in the data window. The value in the data window is the translation distance of the binder.

     Click **APPLY** once, and the binder will be moved along the selected direction with the defined distance.

Toggle **ON** the **REVERSE OPERATION** option, and the binder will be moved along the opposite of the direction selected when the user clicks APPLY.

2. Use the slider to move the binder. Move the slider to the right, and the binder will be moved to the positive direction along the selected axis. Move to the left, and the binder will be moved in the opposite direction. Click **APPLY** to accept the new binder position.

   The total binder movement will be displayed in the data window below the slider. The program will display the binder in the new position as the user moves the slider.

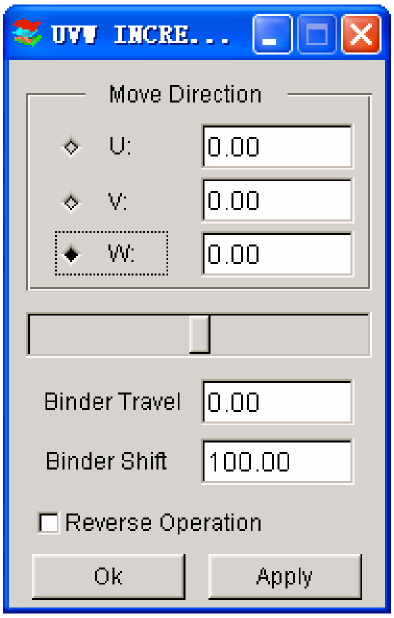


Figure 6.2.9 Move Binder

- **ROTATE BINDER**

This function allows the user to rotate the binder to a desirable orientation relative to the part. Click the **ROTATE BINDER** button, and the program will display a dialog window as in Figure 6.2.10. This function enables the user to create or select a Local Coordinate System (LCS) for rotating the binder. The binder is rotated about the W axis of the LCS.

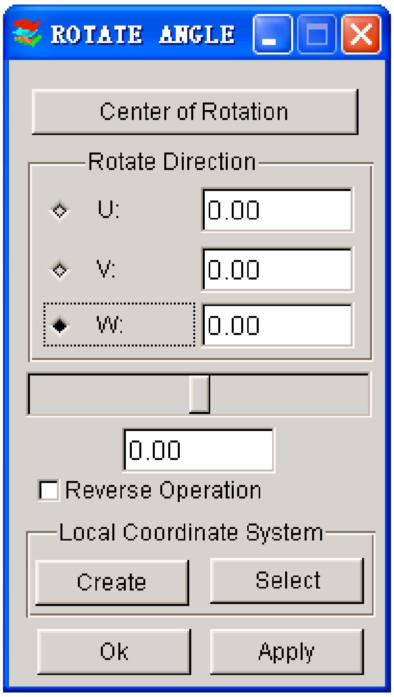


Figure 6.2.10 Rotate Binder

After defining the origin, the LCS is created with U, V and W parallel to the Global Coordinates. The program will display a dialog window as in Figure 6.2.11 to define the LCS. Select a point as the origin and click **OK** to create the LCS.

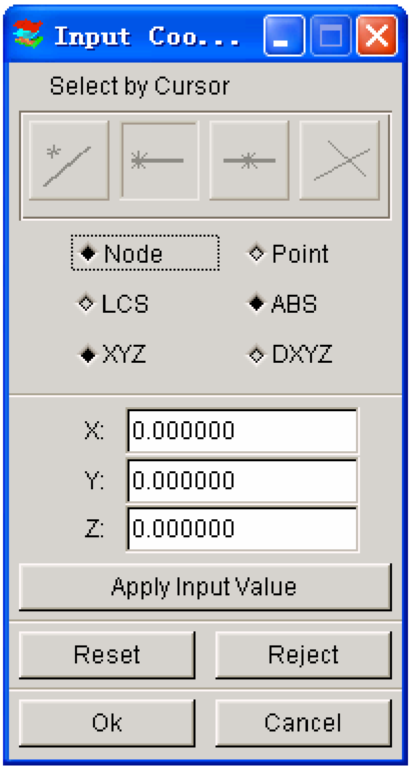


Figure 6.2.11 Select Origin of the LCS

Click the **SELECT** button in Figure 6.2.10, and the program will display the SELECT LCS menu to prompt the user to select an existing LCS.

Once the LSC is defined, the user has a choice of either rotating the binder by the defined rotational angle or using the slider to rotate the binder in dynamic mode.

To define the rotation, the user may select the U, V or W axis and enter the rotational angle in the data window next to the axis. Click one of the following buttons.

Click **APPLY** once, and the binder will rotate counter clockwise about the selected axis of the LCS with the defined angle.

Click **OK** to exit the function.

Toggle **ON** the **REVERSE OPERATION** option, and the binder will rotate about the defined axis of the LCS in the opposite of the selected direction when the APPLY button is clicked.

To use the slider, the user selects the U, V or W axis and moves the slider to rotate the binder. Move the slider to the right to rotate the binder in a counter clockwise direction about the selected axis. Move the slider to the left to rotate in the opposite direction. The program shows the new binder position as the user moves the slider. Click **APPLY** to accept the new binder position.

The program displays the total rotational angle about the selected axis in the data window below the slider.

- **MODIFY BINDER**

This function provides the user with four options to edit the binder by modifying one of two cross section lines on the binder.

Click the **MODIFY BINDER** button, and the program displays the SELECT LINE window and prompts a message, SELECT ONE LINE TO EDIT. Two cross section lines are displayed as shown in Figure 6.2.112.

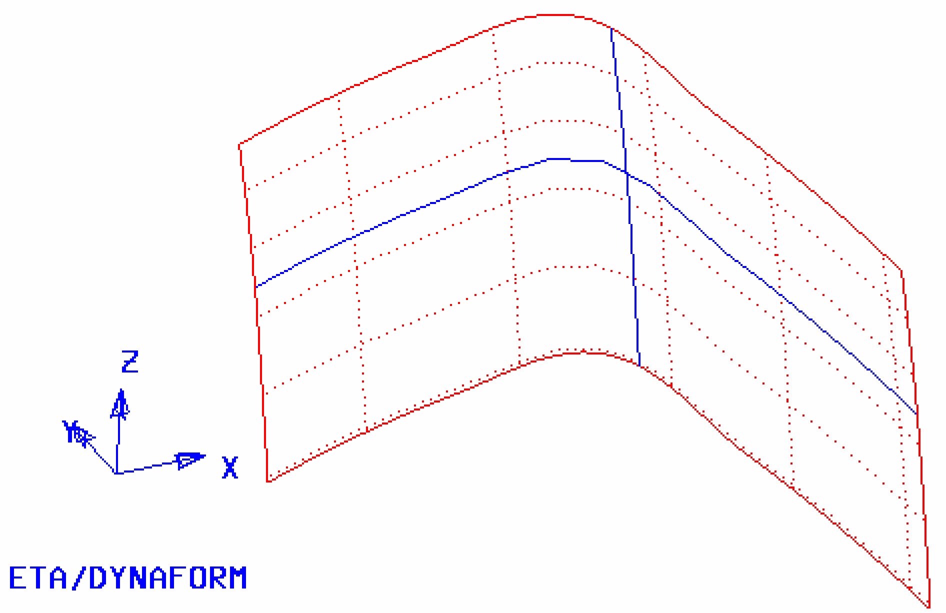


Figure 6.2.12 Modify Binder

After selecting a section line, the program displays a CURVE EDIT window. There are four option tabs available to the user.

1. **SIMPLIFY**

This function simplifies the selected curve by reducing the number of points of the line. Refer to Figure 6.2.13 for the dialog window.

TOL

These are the criteria used to simplify the curve. If the distance between two adjacent points is less than the defined criteria, one of the points will be removed from the curve

PREVIEW

This function shows the new section line after some points are removed

RESELECT CURVE

This function enables the user to select the other section line to be simplified.

APPLY accepts the simplified curve without exiting the function.

OK accepts the simplified curve and exits the function.

CANCEL discards the simplified result and exits the function.

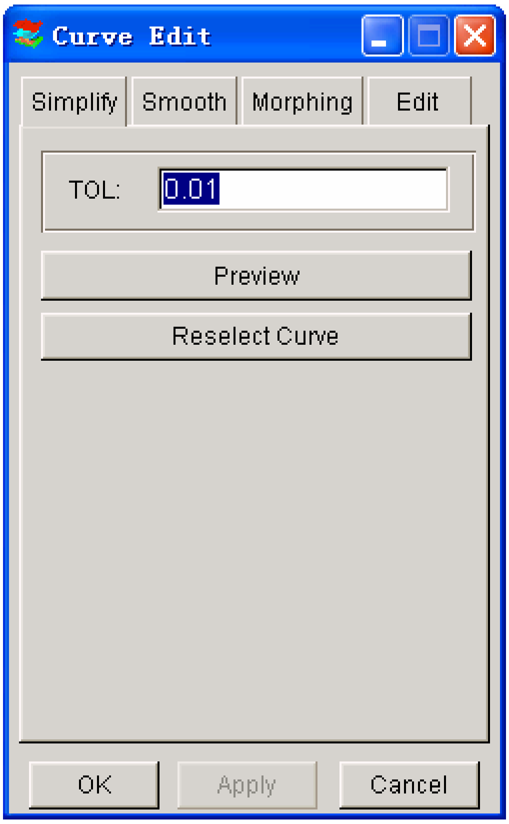


Figure 6.2.13 Simplify Curve

2. **SMOOTH**
   This function enables the user to smooth the selected section line. There are three methods to smooth the section line. Refer to Figure 6.2.14 for the available methods.

   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 data field next to the slider as the smooth coefficient and click the **APPLY** button to smooth the curve. The user can also drag the slider to smooth the curve in dynamic mode. The program will display the smoothed curve as the slider is moved. The data field next to the slider will be updated simultaneously to show the user the smooth coefficient. Click **SELECT POINT** to select constraint point(s) on the curve. Only the points other than the selected points will be smoothed on the curve.

   THROUGH FIXED POINTS

   When this function is toggled **ON**, the **SELECT POINT** button will be activated. Click the **SELECT POINT** button, and the program will highlight all the points on the selected curve. A new spline curve will be created passing through the selected point(s).

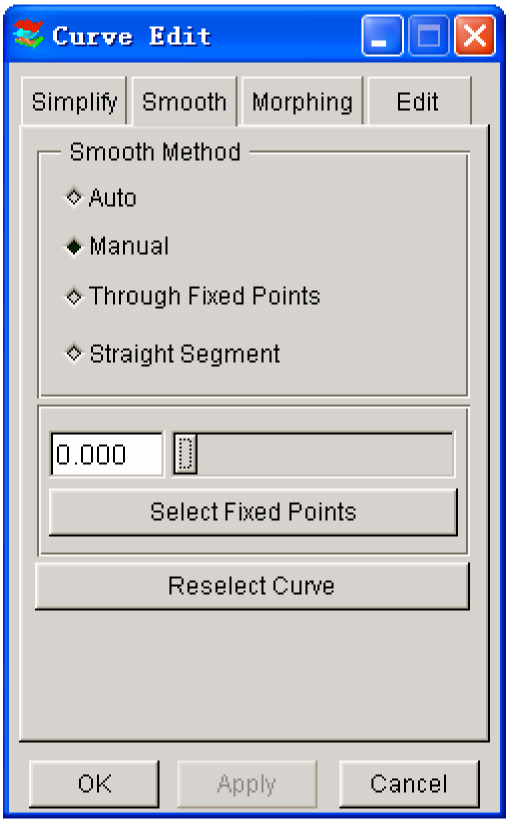


Figure 6.2.14 Smooth Curve

3. MORPHING

This function allows the user to modify the section line by morphing. There are four methods available in the MORPHING function. Figure 6.2.15 illustrates the characteristics and results of each morphing method.

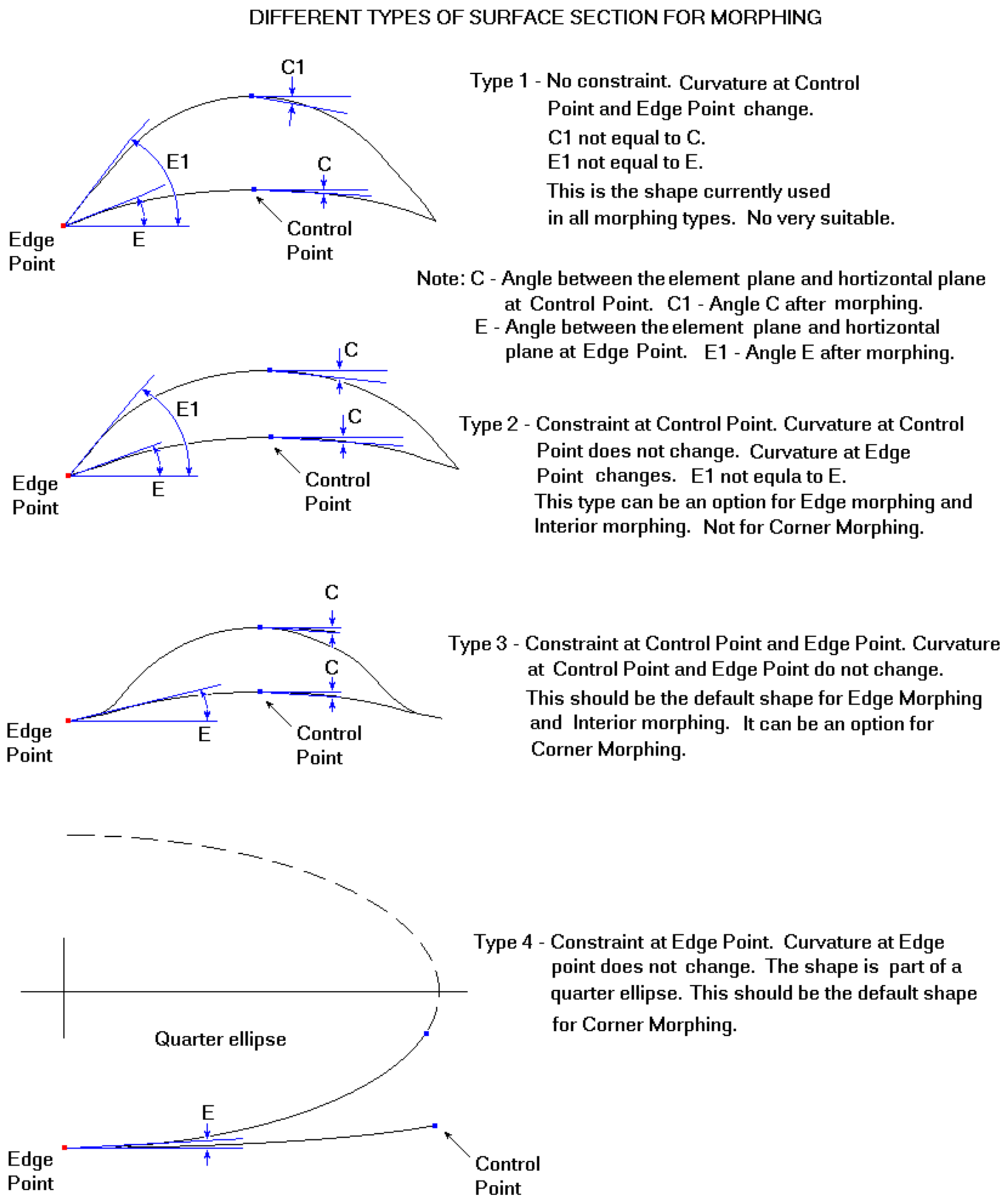


Figure 6.2.15 Morphing Methods

NO CONSTRAINT – There is no constraint imposed on the section line when it is morphed. The morphed shape will assume a natural spline curve. Refer to Type 1 in Figure 6.2.15 for the section line change after morphing.

KEEP C ANGLE – The tangent at the control point on the section line is constrained. The morphed shape will maintain the original tangent direction at the control point of the section line. Refer to type 2 in Figure 6.2.15 for the section line change after morphing.

KEEP E ANGLE – The tangent at both end points on the section line is constrained. The morphed shape will maintain the original tangent direction at the end points of the section line. Refer to type 3 in Figure 6.2.15 for the section line change after morphing.

KEEP CONVEX – Maintains the convex sharp by changing the curvature of the section line.

Figure 6.2.16 shows the dialog window for the MORPHING function. Click **SELECT MORPHING POINT** to select the control point. The program will highlight the points on the section line and prompt the user to select one as the control point. The program draws a vector at the morphing point to indicate the morphing direction. The user may move the cursor to see the new shape of the section line in real time. Click the **RIGHT MOUSE BUTTON** to accept the morphed shape. Clicks **APPLY** to update the binder shape according to the morphed section line. eta/DYNAFORM calculates a default morphing direction that is normal to the section line and the binder surface. The user may click on the **USER DEFINE** option to enter the desirable vector for the morphing direction.

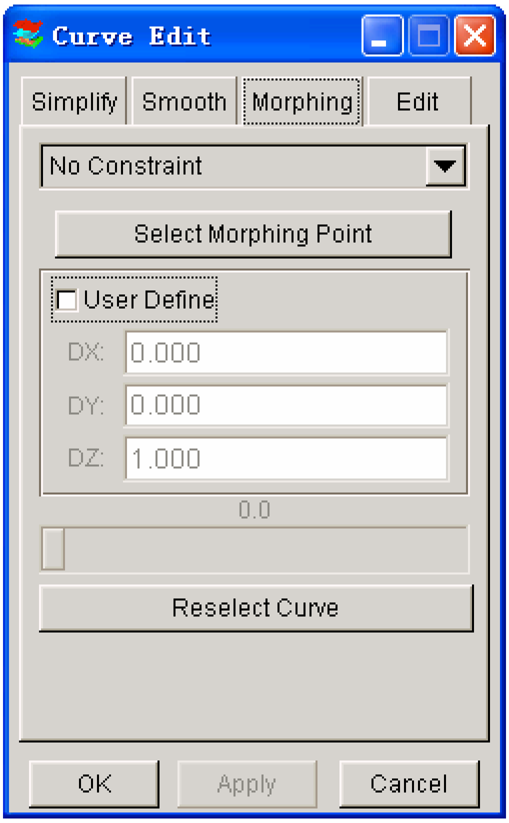


Figure 6.2.16 Morphing Curve

4. EDIT

This function provides a method to edit the section line by editing the points on the select line. Figure 6.2.17 shows the available options for editing the section line.

ADD POINT

When this option is toggled **ON**, the **BY CURSOR LOCATION** and **BETWEEN TWO POINTS** buttons are activated.

The user may toggle **ON** the **BY CURSOR LOCATION** button to pick any location on the section line to add a point.
Toggle **ON** the **BETWEEN TWO POINTS** option**,** and click the **SELECT POINT** button. The program will display points on the section line and prompt the user to select two adjacent points. After two adjacent points are selected, the program will activate the **POINT NUMBER** field for the user to enter the number of points to be added between two selected points. If the second selected point is not adjacent to the first one, the program will issue an error message, **INVALID POINT SELECT, REJECTED.**

MOVE CURVE

This function enables the user to move a selected section line. The binder surface will be moved with the section line. Toggle the option **ON**, and the program will activate three data windows next to DX, XY and DZ. The user may enter any three displacement vectors with respect to the global X, Y and Z direction. Click **APPLY** to move the section line and the binder.

MOVE POINT

This function allows the user to move a point on the section line in order to move the binder accordingly.
Toggle the option **ON**, and the **SELECT POINT** button and three data windows next to DX, DY and DZ will be activated. The user may click **SELECT POINT** to select a point on the section line, enter the displacement value with respect to the global X, Y and Z direction, and click **APPLY** to move the section point. The binder shape will be modified according to the new shape of the section line.

REMOVE POINT

This function enables the user to remove the control points from the selected section line.
Toggle the option **ON**, and the **SELECT POINT** button is activated for the user to select the point(s) on the section line. Click the **MIDDLE MOUSE BUTTON** to complete the point selection. Refer to Figure 6.2.18 for an illustration of the control points.

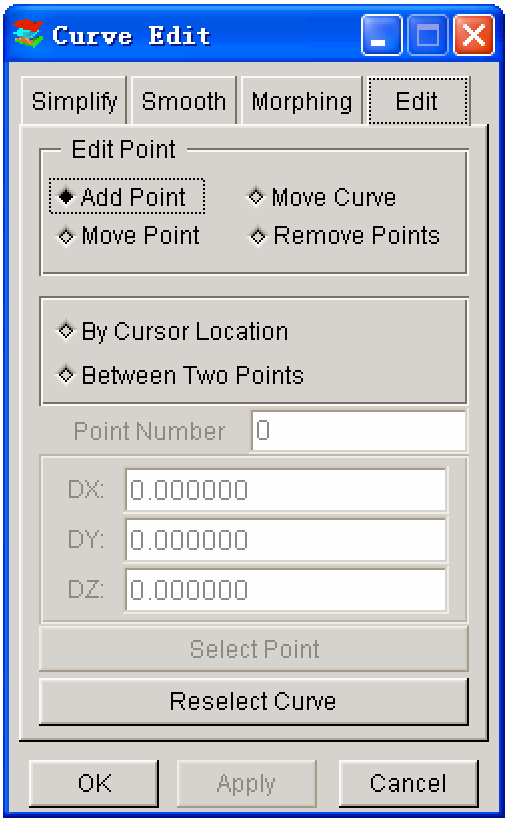


Figure 6.2.17 Edit Curve

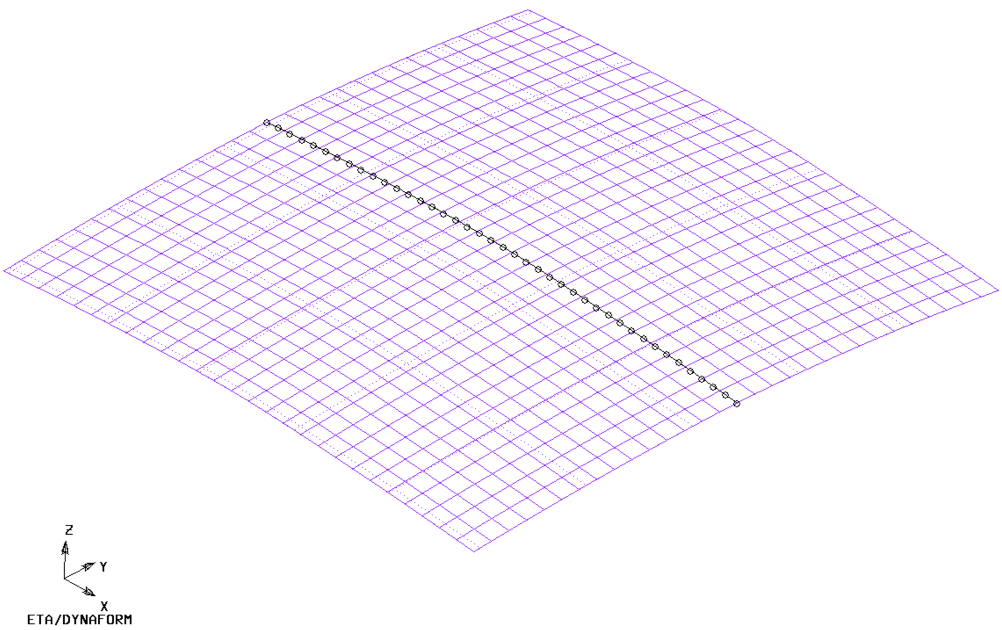


Figure 6.2.18 Control points on section curve

- **EDIT BINDER**

  This function enables the user to edit the binder through the selected U-V line of the binder surface. The user can define the number of the U-V line thought Section on Line dialog box show as in Figure 6.2.19.

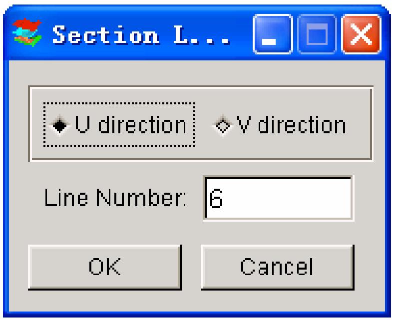


Figure 6.2.19 Define the number of U-V line

Click OK to accept the number of U or V line and then select one of the section line. The edit function is similar to the Modify Binder before.

- **MESH BINDER**

  This function is used to generate the binder mesh from the binder surface. DYNAFORM displays a dialog window as shown in Figure 6.2.20.

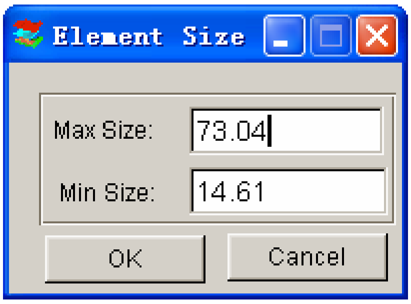


Figure 6.2.20 Mesh Parameters

The program 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 data windows.

MAX SIZE – Controls the maximum element size of the mesh.

MIN SIZE – Controls the minimum element size of the mesh.

- **CREATE**

  This function creates a binder surface and adds it to the part called C_BINDER.

Note: If there is a part called C_BINDER in the current database, eta/DYNAFORM will create a part called BINDER01. Subsequently, if there is BINDERn, BINDERn+1 will be created. eta/DYNAFORM will recognize the binder created from the Binder menu.

- **DELETE**

  Deletes surface and mesh in the binder from the database.

- **EXIT**

  Closes the BINDER dialog window.

6.2.2 FLAT BINDER

This function creates a flat binder. There are two methods to define the orientation of a flat binder as shown in Figure 6.2.20.

DEFINE BINDER ORIENTATION

Refer to Section 6.2.1 for a detailed description defining binder orientation.

SELECT 3 NODE/POINT

This function enables the user to select three nodes to define binder orientation. The first two nodes define the U direction of the binder surface, and three nodes determine the flat binder surface.

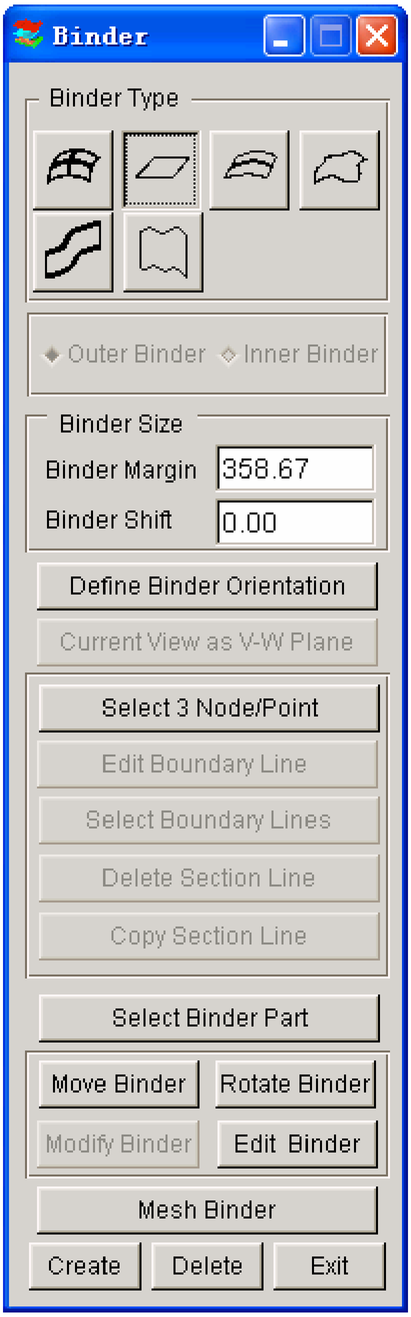


Figure 6.2.21 Flat Binder

The direction of the UV line of the binder surface can also be set by the **DEFINE BINDER ORIENTATION** button.

All other functions are the same as in **TWO-LINE BINDER** as described in Section 6.2.1.

6.2.3 CONICAL BINDER

This function is used to create a conical binder. The program uses the radius defined at each end of the binder to control the conical shape. Once the function is selected, **RADIUS 1** and **RADIUS 2** data windows are activated as shown in Figure 6.2.22.

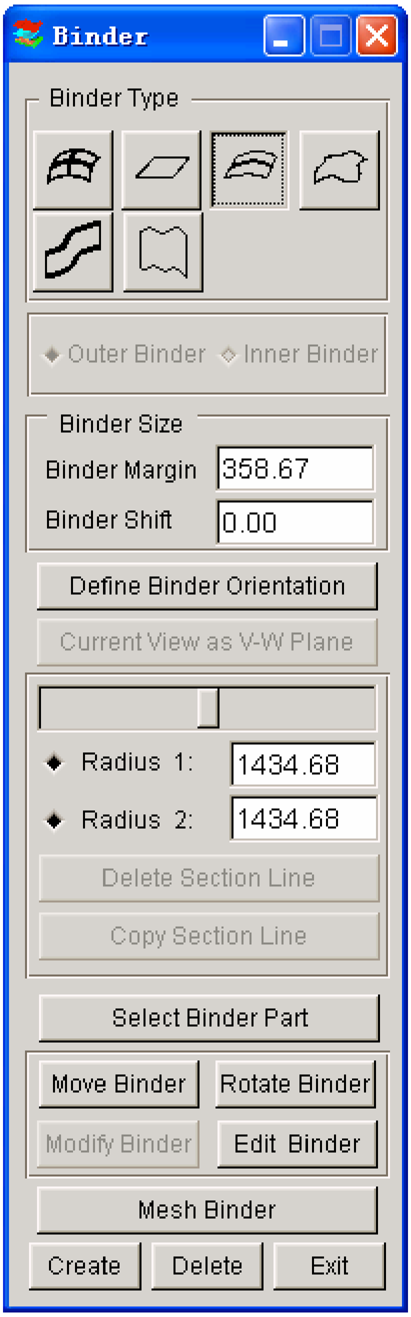


Figure 6.2.22 Conical Binder

Figure 6.2.23 shows a typical conical binder.

The slider in the dialog window allows the user to change the radius by moving the slider. Radius 1 and Radius 2 may be changed simultaneously or separately. Click on the radio button 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. The program will display the binder in real time as the user moves the slider to change the radius.

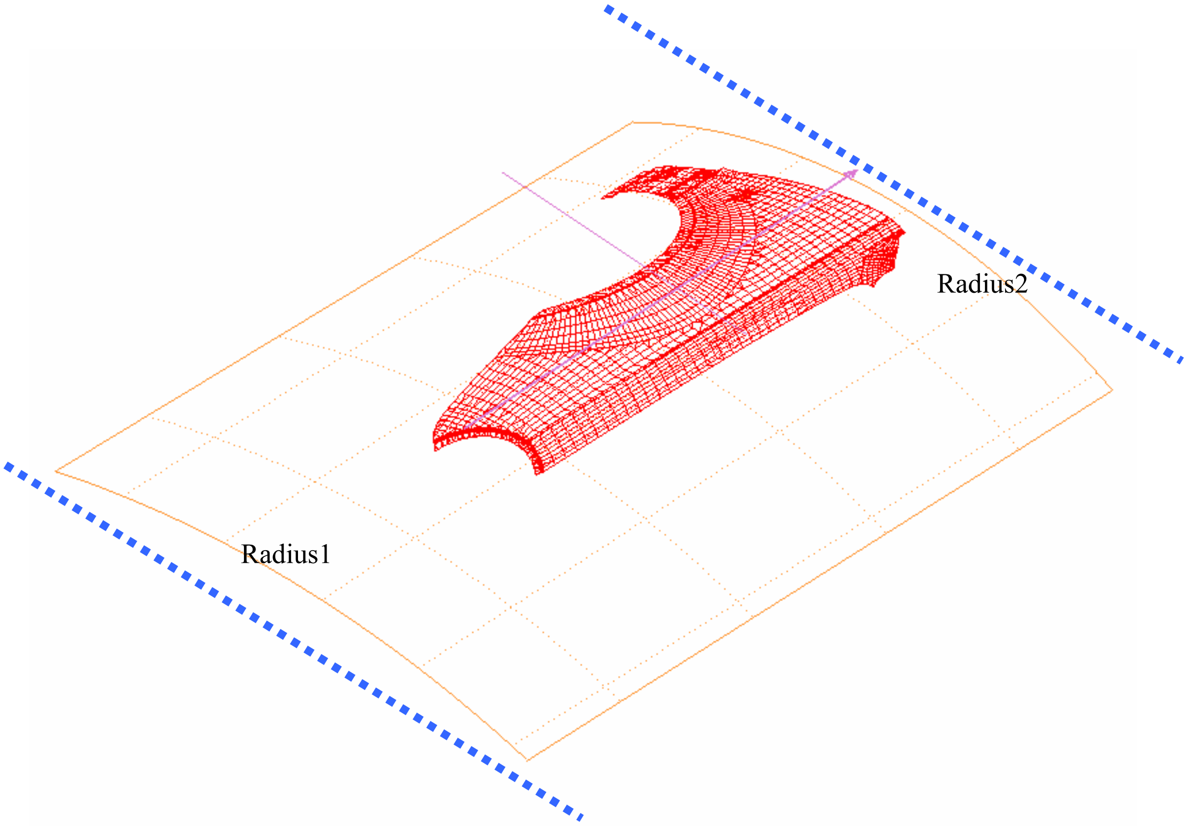


Figure 6.2.23 Conical Binder

Note: DYNAFORM calculates the default conical Radius 1 and conical Radius 2 based on the size and shape of the part.

6.2.4 BOUNDARY LINE BINDER

This function generates a binder surface from the boundary line of a part. Once the icon is clicked, the program displays a dialog window as shown in Figure 6.2.24.

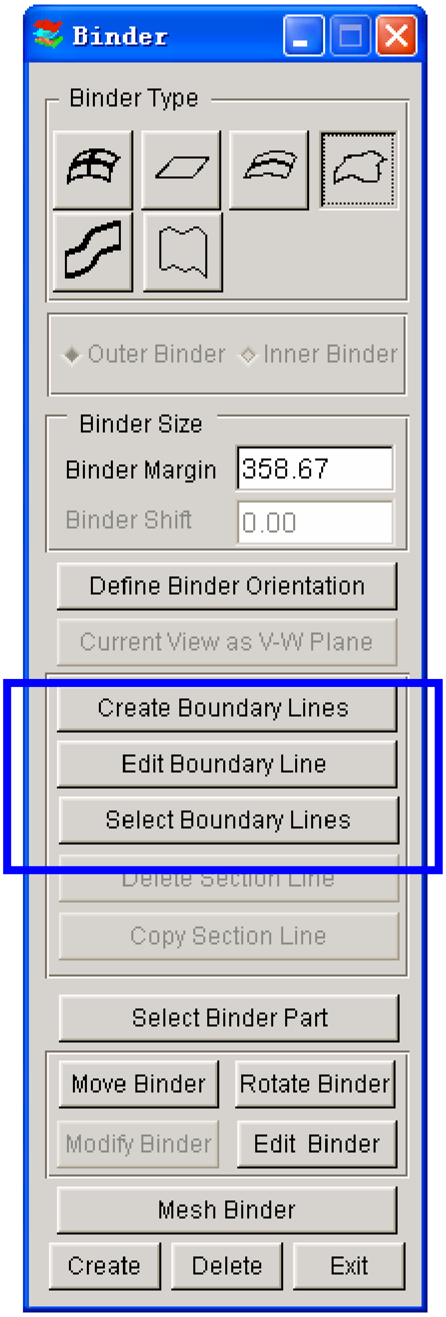


Figure 6.2.24 Boundary Line Binder

The following functions are used to generate the Boundary Line binder.

DEFINE BINDER ORIENTATION

This function defines the UV-direction of the binder surface. The cross line with the arrow controls the surface shape and must be parallel to the direction of the part boundary. Figure 6.2.25 shows a typical orientation of the U-V axes relative to the part.

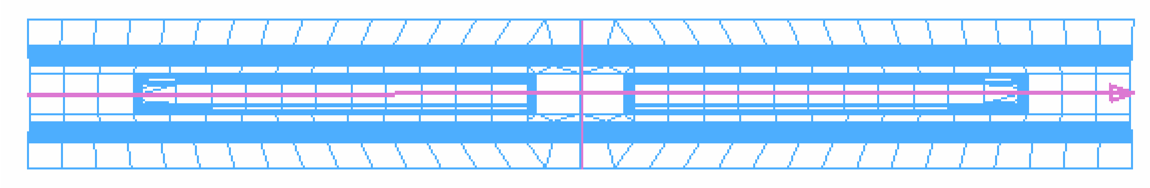


Figure 6.2.25 Define Binder Orientation

CREATE BOUNDARY LINES

This function creates the boundary lines of the part. The program will create the boundary line of the part mesh and put in the current part. Figure 6.2.27 shows typical boundary lines of a part.

EDIT BOUNDARY LINE

The functions in this menu are used to edit the boundary lines generated from the **CREATE BOUNDARY LINE** function. Figure 6.2.26 shows the options available to edit the boundary line. The description of edit line functions is given in Section 5.1.

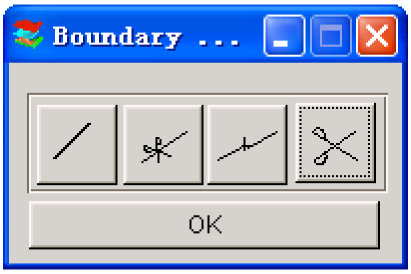


Figure 6.2.26 Edit Boundary Line

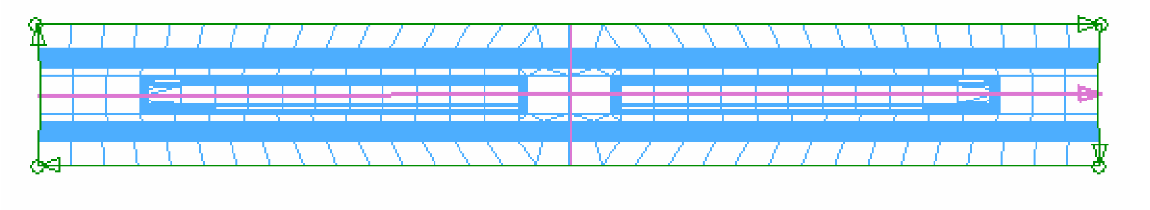


Figure 6.2.27 Created Boundary Line

SELECT BOUNDARY LINE

This function is used to select the desired boundary lines to generate the binder surface. The selected line will be highlighted as shown in Figure 6.2.28. Click the **CREATE** button to generate the binder surface. The generated binder is shown in Figure 6.2.29.

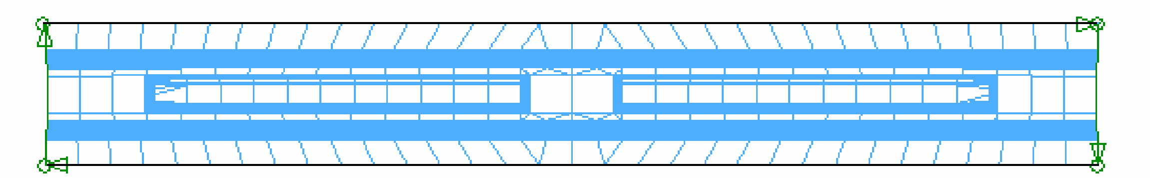


Figure 6.2.28 Selected Boundary Line

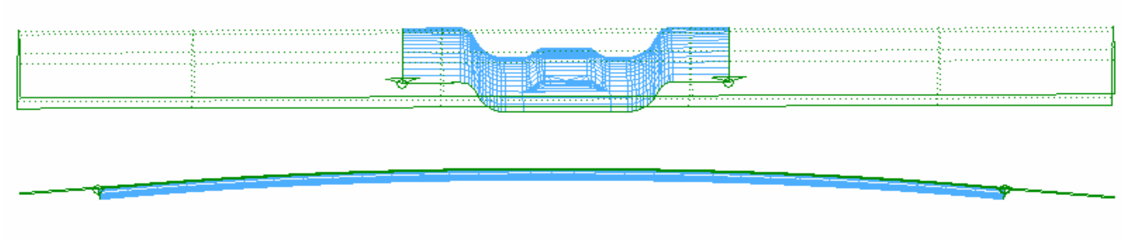


Figure 6.2.29 Created Binder

If the created binder is not acceptable, the user may edit the boundary line and create it again.

For a detailed description of other editing functions, refer to Section 6.2.1, TWO-LINE BINDER.

6.2.5 FLANGE BINDER

This function generates a binder surface from the character lines of flanges of a part. Figure 6.2.30 shows the available functions in the **FLANGE BINDER** dialog window.

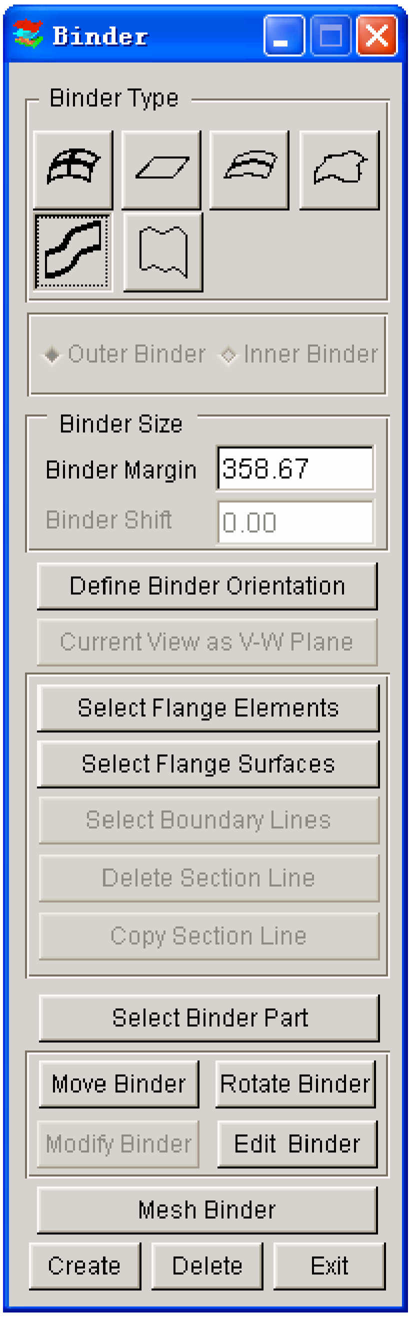


Figure 6.2.30 Flange Binder

The operation of **FLANGE BINDER** is similar to **BOUNDARY LINE BINDER** except that the user must select either elements or surfaces to define the flanges of a part.

DEFINE BINDER ORIENTATION

This function defines the UV-direction of the binder surface. The cross line with arrow controls the surface shape and must be parallel to the direction of the part boundary. Figure 6.2.24 shows a typical orientation of the U-V axes relative to the part.

SELECT FLANGE ELEMENTS

This function is used to select elements on the flanges of a part. The program will display the

SELECT ELEMENT dialog window for the selection of elements. Refer to Section 4.3.2 for a detailed description.

SELECT FLANGE SURFACES

This function is used to select surface on the flanges of a part. The program will display the **SELECT SURFACE** dialog window in order to select surfaces. Refer to Section 4.3.3 for a detailed description.

EDIT BOUNDARY LINES

This function allows the user to edit the boundary lines of the flange surface where it is not suitable to generate the binder surface. See a detailed description on the EDIT LINE functions in Section 6.2.1.

After the elements or surfaces are selected for the flanges, click the **CREATE** button to create the binder surface. Figure 6.2.31 shows typical flange elements of a part.

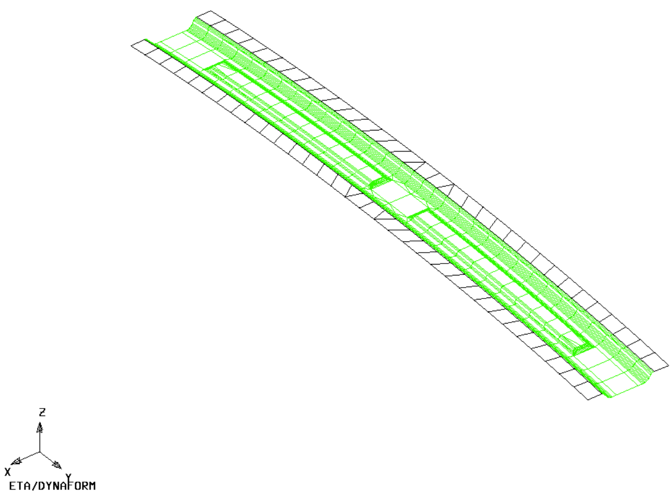


Figure 6.2.31 Selected elements on flanges

Figure 6.2.32 shows a typical binder surface generated from the flange elements.

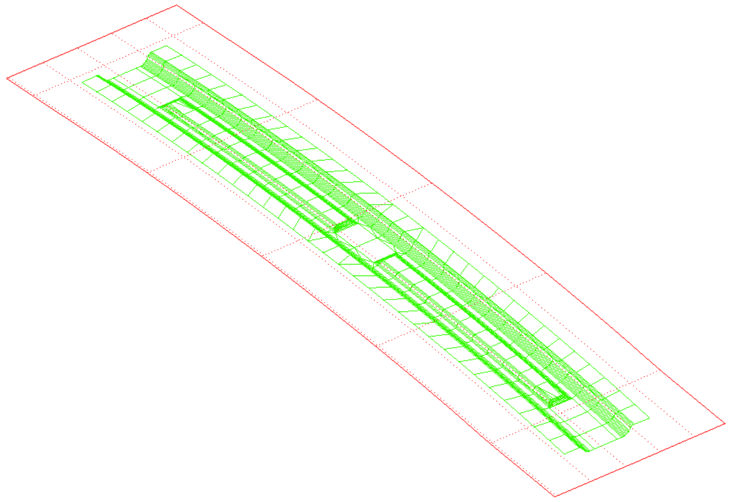


Figure 6.2.32 Typical Flange binder

For a detailed description of other binder editing functions, refer to Section 6.2.1, TWO-LINE BINDER.

6.2.6 FREE FORM BINDER

This function generates a binder surface from the section lines of binder defined on the screen. Figure 6.2.33 shows the available functions in the **FREE FORM BINDER** dialog window.

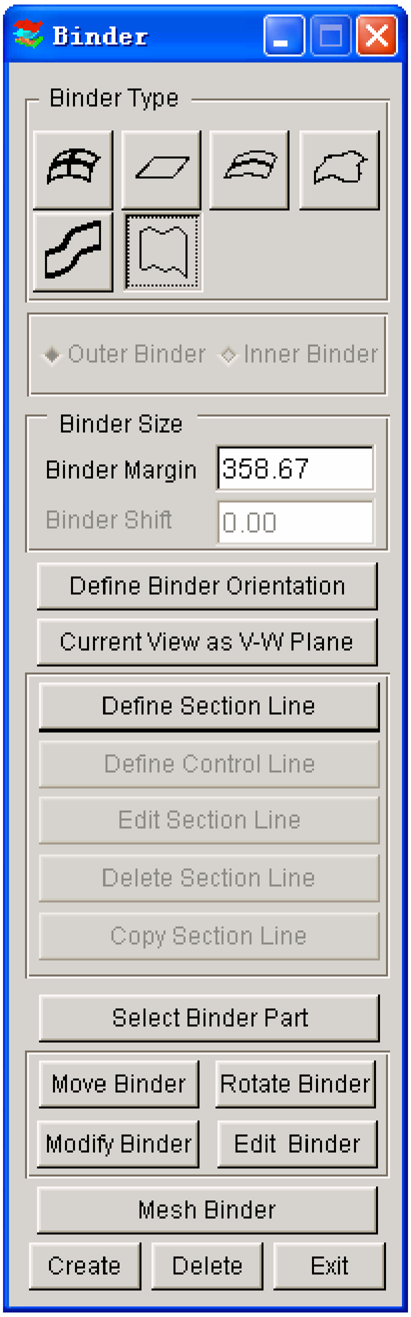


Figure 6.2.33 Free Form Binder

DEFINE BINDER ORIENTATION

This function defines the UV-direction of the binder surface. The cross line with arrow controls the surface shape and must be parallel to the direction of the part boundary. Figure 6.2.2 shows a typical orientation of the U-V axes relative to the part.

CURRENT VIEW AS V-W PLANE

This function enables the user to use the current view to define the points of the binder section line. For certain parts, it is not obvious to define the binder U-V axes from the top view using DEFINE BINDER ORIENTATION function. The user rotates the part to a view that is suitable to define the binder section. Clicks the CURRENT VIEW AS V-W PLANE button to define the binder U-V plane according to the current view. The program will define the binder V axis parallel to the screen X direction (from left to right). The binder W axis will be defined parallel to the screen Y direction (from bottom to top). The binder U axis will be 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 select locations on the screen to define the section line at both end of the binder along the binder U axis. Click Define Section Line to display the Select Point dialog, see the Figure 6.2.34 The program will display the part model in binder V-W plane automatically and prompt the user the following message.

CLICK LOCATIONS ON THE SCREEN TO DEFINE SECTION LINE

CLICK LEFT BUTTON TO SELECT, RIGHT TO REJECT, MIDDLE TO EXIT

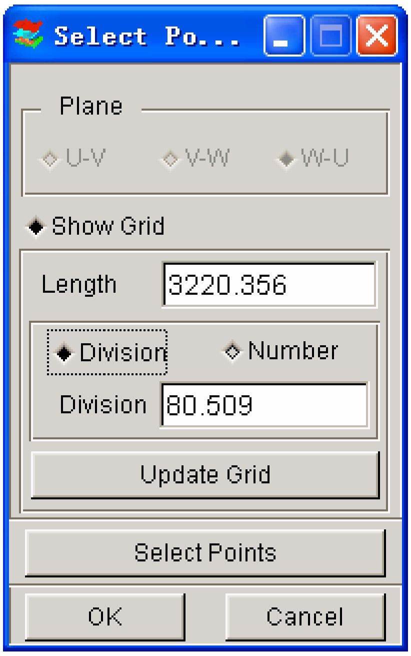


Figure 6.2.34 Selected points of section line

Show Grid: toggle on this option, the Grid will display on the V-W plane, and all the points will be selected locate on the grid. This will help user to define the section line.

Length: Define the edge length of the Grid plane.

Division: Input value to define the unit length of the grid plane.

Number: Input value to define the number of grid..

Update: Update the Grid Plane.

The user may define as many locations on the screen as necessary to define the section line. The program will extend the section line according to the Binder Margin defined in the dialog window. Figure 6.2.35 shows a typical section line defined on the screen.

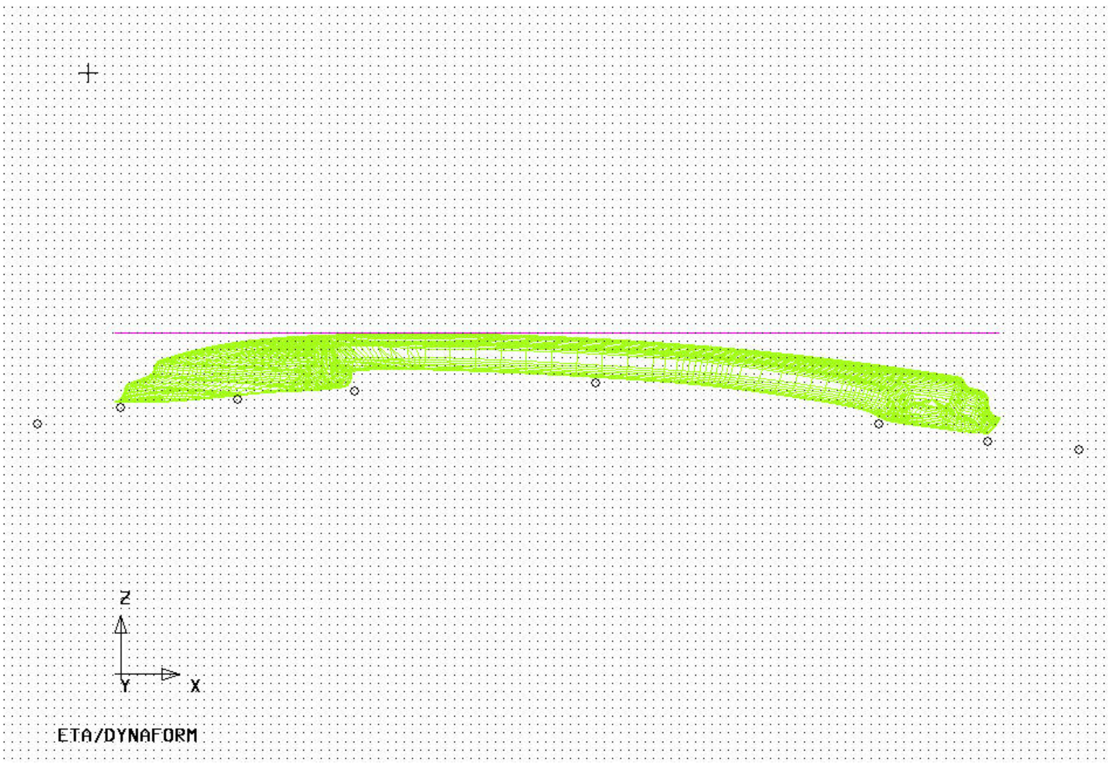


Figure 6.2.35 Typical section line defined by locations on the screen

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, the program will use a straight line between the section lines at both end of the binder. Once this function is activated, the program will display the part model in binder U-W plane to prompt the user to select location on the screen to define the control line.

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. The program will map the control line to joint the section lines. Figure 6.2.36 shows a typical control line joining the section lines.

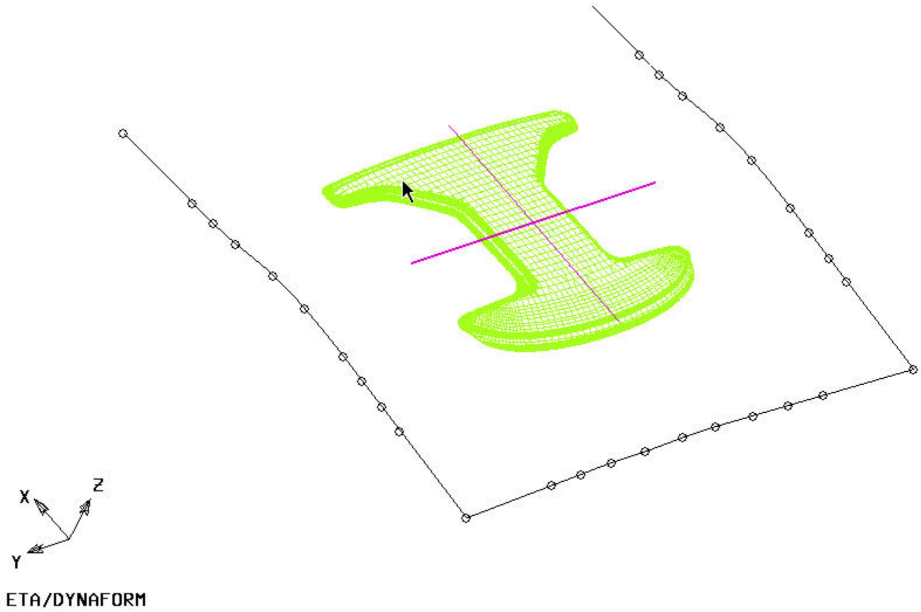


Figure 6.2.36 Typical control line joining section lines

EDIT SECTION LINE

This function allows the user to modify the shape of the section line or control line. See a detailed description on the EDIT LINE function in Section 6.2.1.

DELETE SECTION LINE

This function allows the user to delete the section line or control line. If a section line is deleted, the program will assign the existing section line to the opposite end of the binder. If the control line is deleted, the program will return to the default setting. That is to use a straight line between the section lines at both end of the binder.

COPY SECTION LINE

This function allows the user to copy the section line to the opposite end of the binder. It is usually done after a section line is modified and the user wishes to have the identical section line on the opposite end. The user would delete the other section line before executing this function.

Figure 6.2.37 shows a typical free from binder with a curved control line.

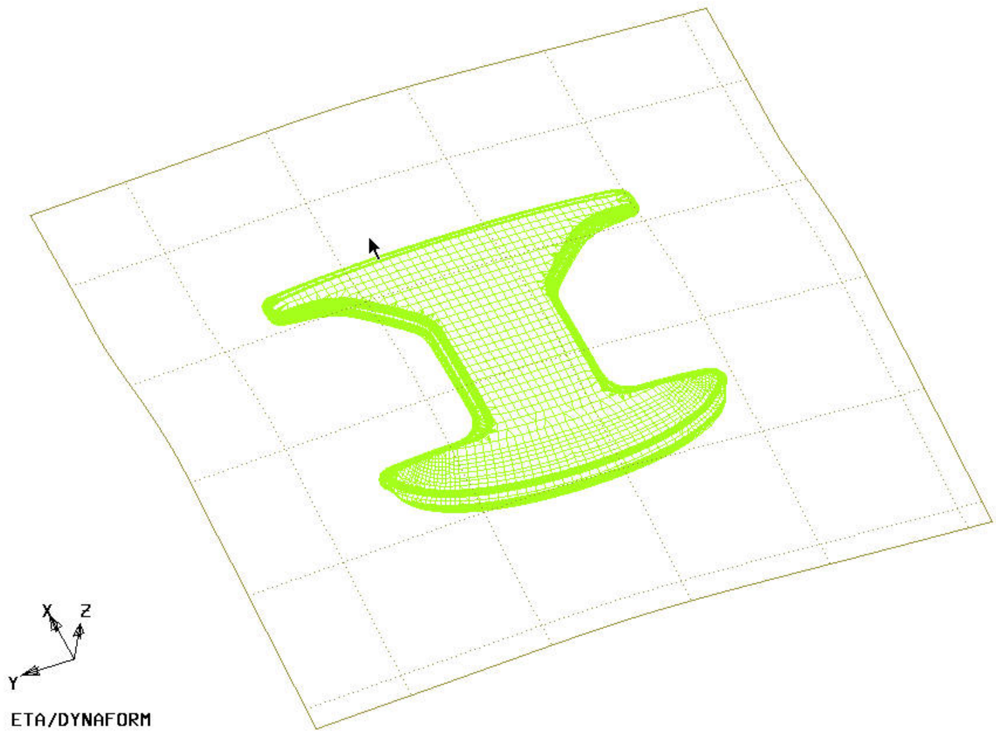


Figure 6.2.37 Typical free form binder with control line

For a detailed description of other binder editing functions, refer to Section 6.2.1, TWO-LINE BINDER.

6.3 ADDENDUM DESIGN

The functions in this menu provide the tool to create addendum surfaces and mesh on a part. The ADDENDUM dialog window contains all the functions for the generation and modification of outer or inner addenda. The functions in this menu are organized into three sections: Master Profiles, Addendum and Profile. Figure 6.3 shows the ADDENDUM dialog window.

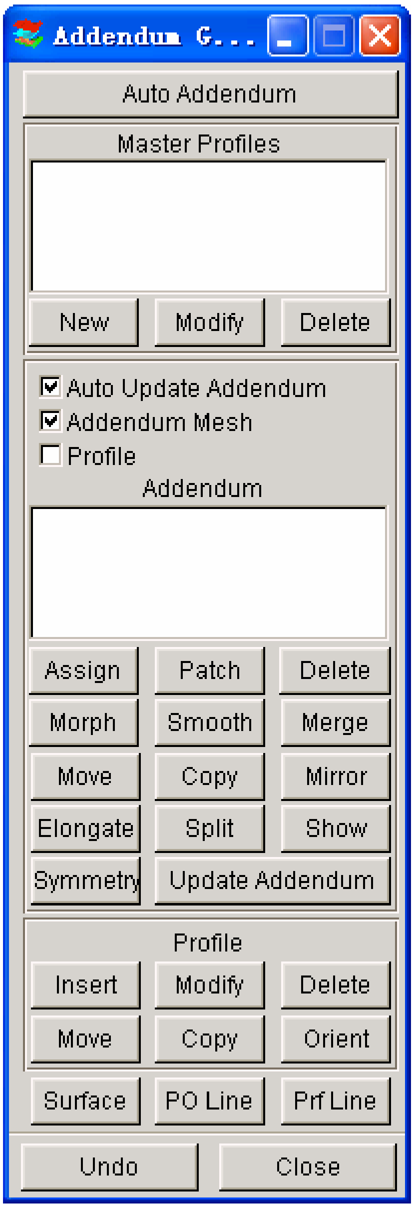


Figure 6.3 Addendum Design

Note:

1. The ADDENDUM menu requires that the Binder be defined in the database.

2. When the ADDENDUM menu starts for the first time, DYNAFORM will create three parts in the database: POP_LIN, PROFILE and ADDENDUM.

6.3.1 AUTO ADDENDUM

eta/DYNAFORM provides an option to automatically create the addendum. The created addendum is compatible with the manual profile and may be edited by the addendum editing functions. DYNAFORM displays a dialog window as shown in Figure 6.3.1.

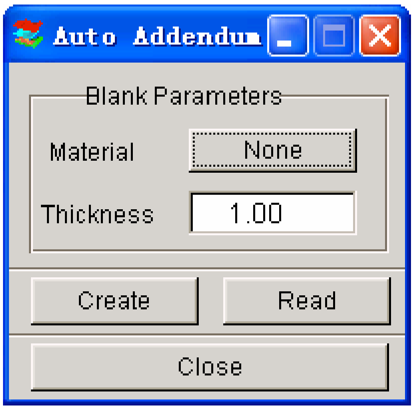


Figure 6.3.1 Auto Addendum

Because this function is based on a one step code, the user is required to supply the material property. Click **CREATE**, and the program will start the **AUTO ADDENDUM** solver to create the addendum.

READ enables the user to read in a set of profiles saved in a *.txt file.

6.3.2 MASTER PROFILE

A profile is a cross section line of the addendum that is usually cut along the outer boundary of a part with the direction normal to the boundary. 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 all 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 a schematic drawing with respect to the settings for Width, Height, Radii and Angles.

Click the **NEW** button from the **ADDENDUM** menu, and DYNAFORM will open the **PROFILE** display window. This window shows the master profile with the end section of the part and binder geometry. Figure 6.3.2 shows a Type 1 master profile with two radii.

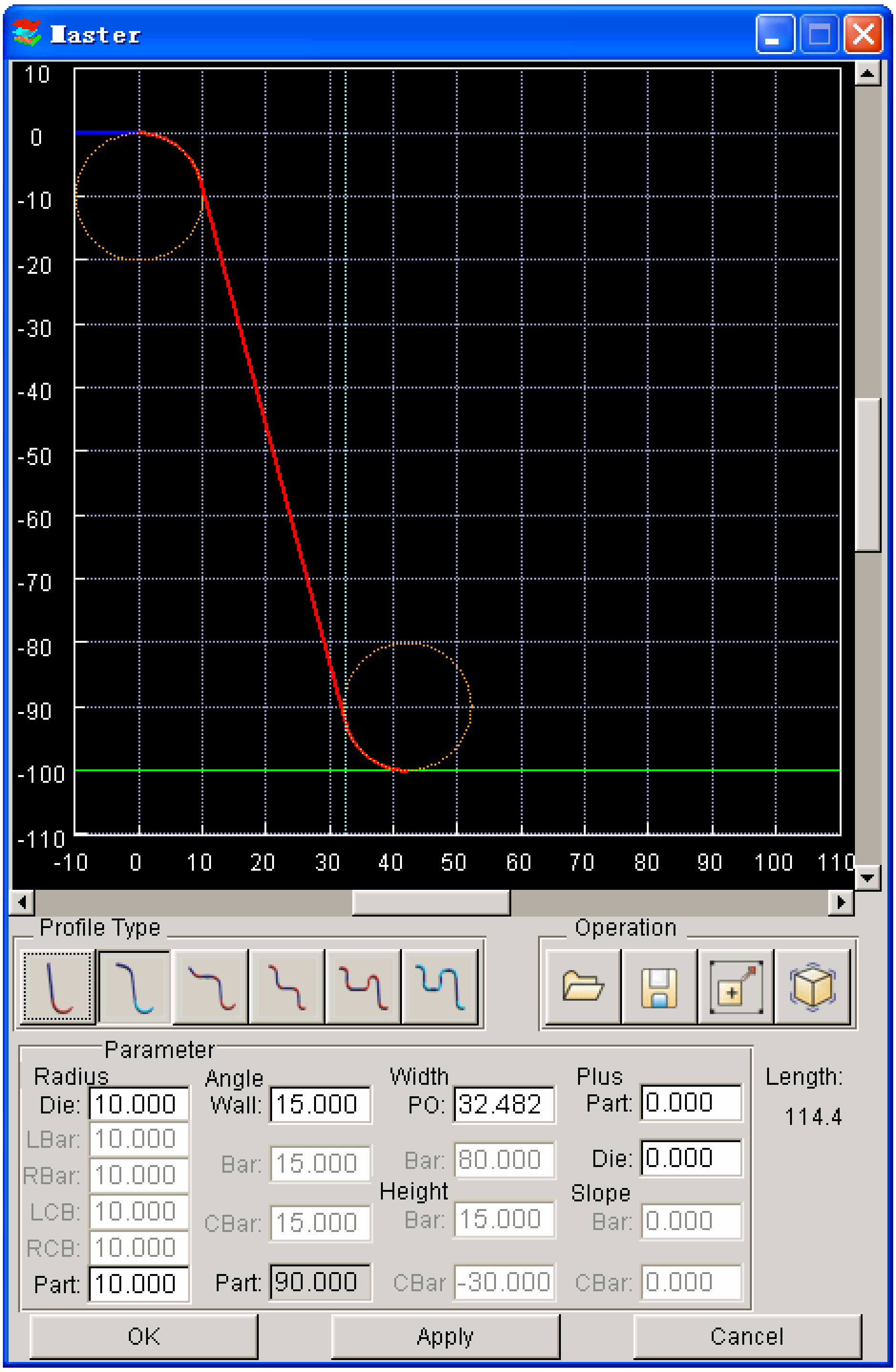


Figure 6.3.2 Master Profile

The dotted circles represent various radii parameters that can be interactively changed. Using the **LEFT** mouse button, click and drag a circle perimeter. The program shows the radius of the circle in the corresponding data window under the Radius column.

There are four different dashed lines that can be used to adjust the position of the circles. Using the **LEFT** mouse button, click and drag the line to move the circle. The program will display the corresponding parameters in the data windows.
The total length-of-line of the profile is displayed at the lower right corner of the **PROFILE GRAPHIC** window.

These lines or circles may also be modified using the corresponding data boxes. Figure 6.3.3 shows all the available parameters.

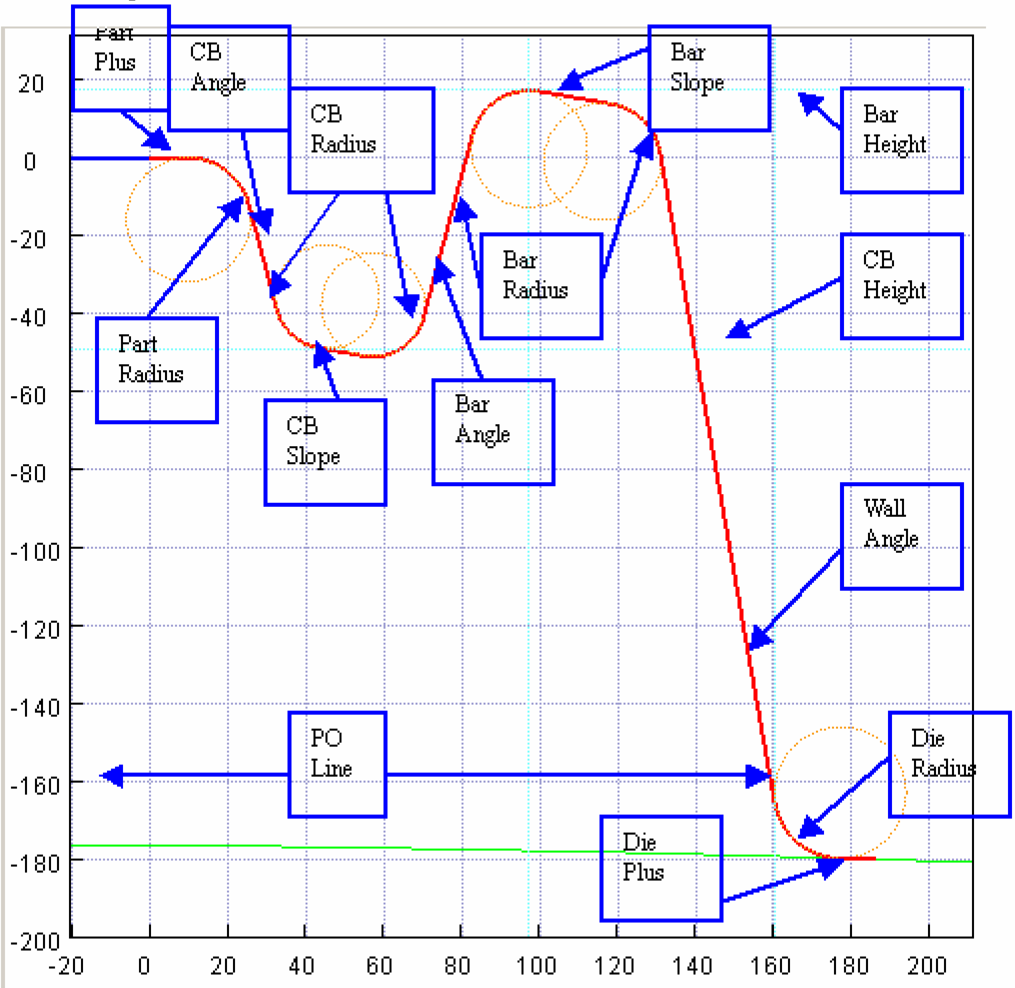


Figure 6.3.3 Various Addendum Parameters

Note: The user should create different profiles using these lines to develop a feeling for 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 see the effects on the profile.

WIDTHS:

Bar: Determines the width of the drawbar.
PO Line: Determines the position of the punch opening line (PO Line).
HEIGHTS:
Bar: Determines the height of the drawbar.
CBar: Determines the position of the CB height line (Counter bar) with respect to part boundary.

There are 6 types of master profiles available in DFE.

TYPE 1

Type 1 profile is used primarily for most parts' vertical edge. This type of profile will produce a tangential transition from the part boundary to the binder surface. **Bar width**, **Bar height** and **CB height** are deactivated as shown in Figure 6.3.4. This type of profile is always extended from the part boundary.

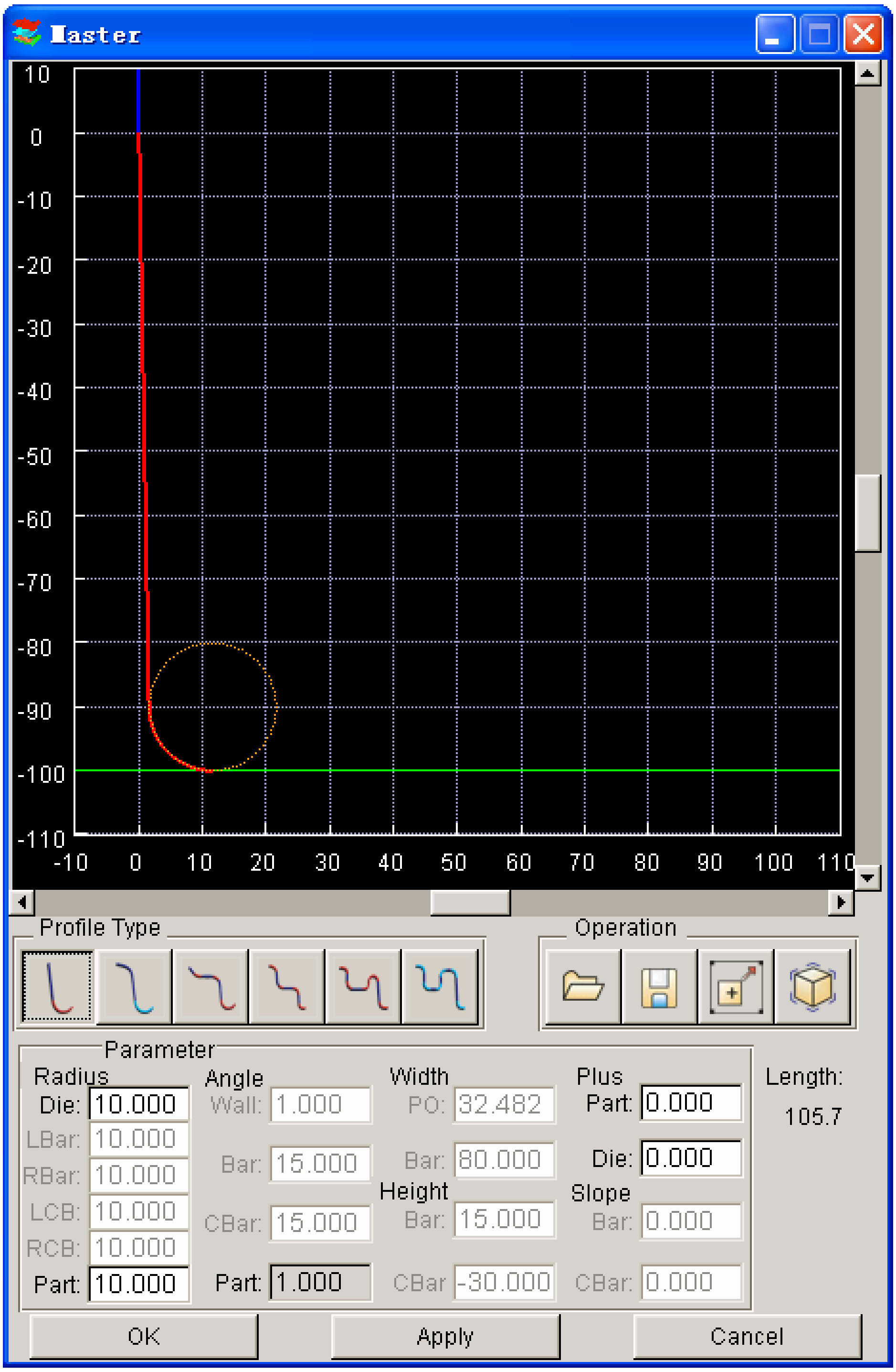


Figure 6.3.4 Profile Type1

TYPE 2

Type 2 profile produces the shortest distance between the part boundary and the punch opening line. **Bar width**, **Bar height** and **CB height** are deactivated as shown in Figure 6.3.5.

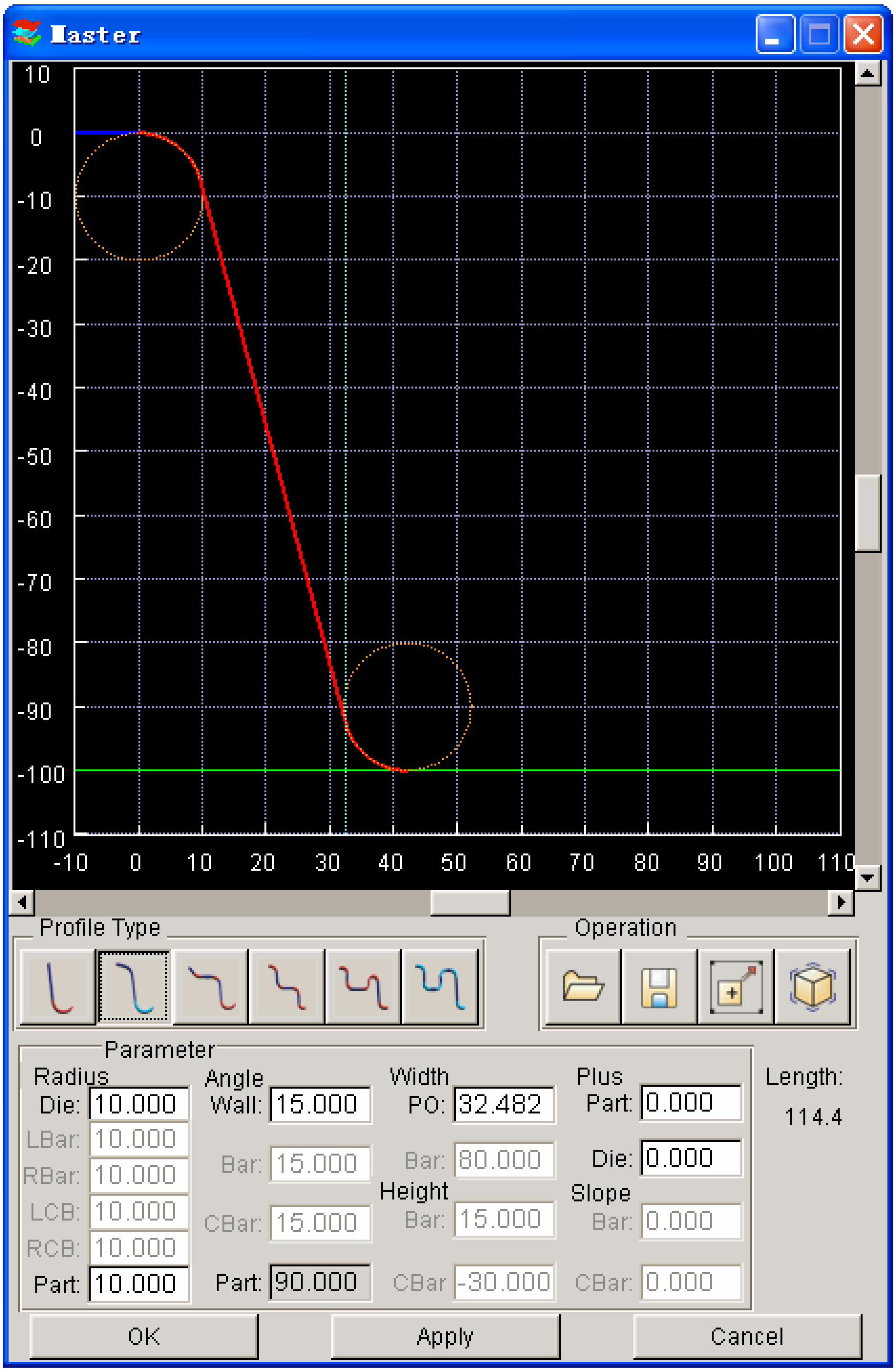


Figure 6.3.5 Profile Type 2

The position of PO depends on the Punch Radius, Die Radius and Part Plus width (if not zero).

TYPE 3

Type 3 profile is similar to Type 1 with an added step in the middle of the profile. The height of **Bar** is measured from the tangential extension of the part. **Bar width** is set to minimum and is not activated. **CB height** is also switched off. Figure 6.3.6 shows the **TYPE 3 PROFILE** dialog window.

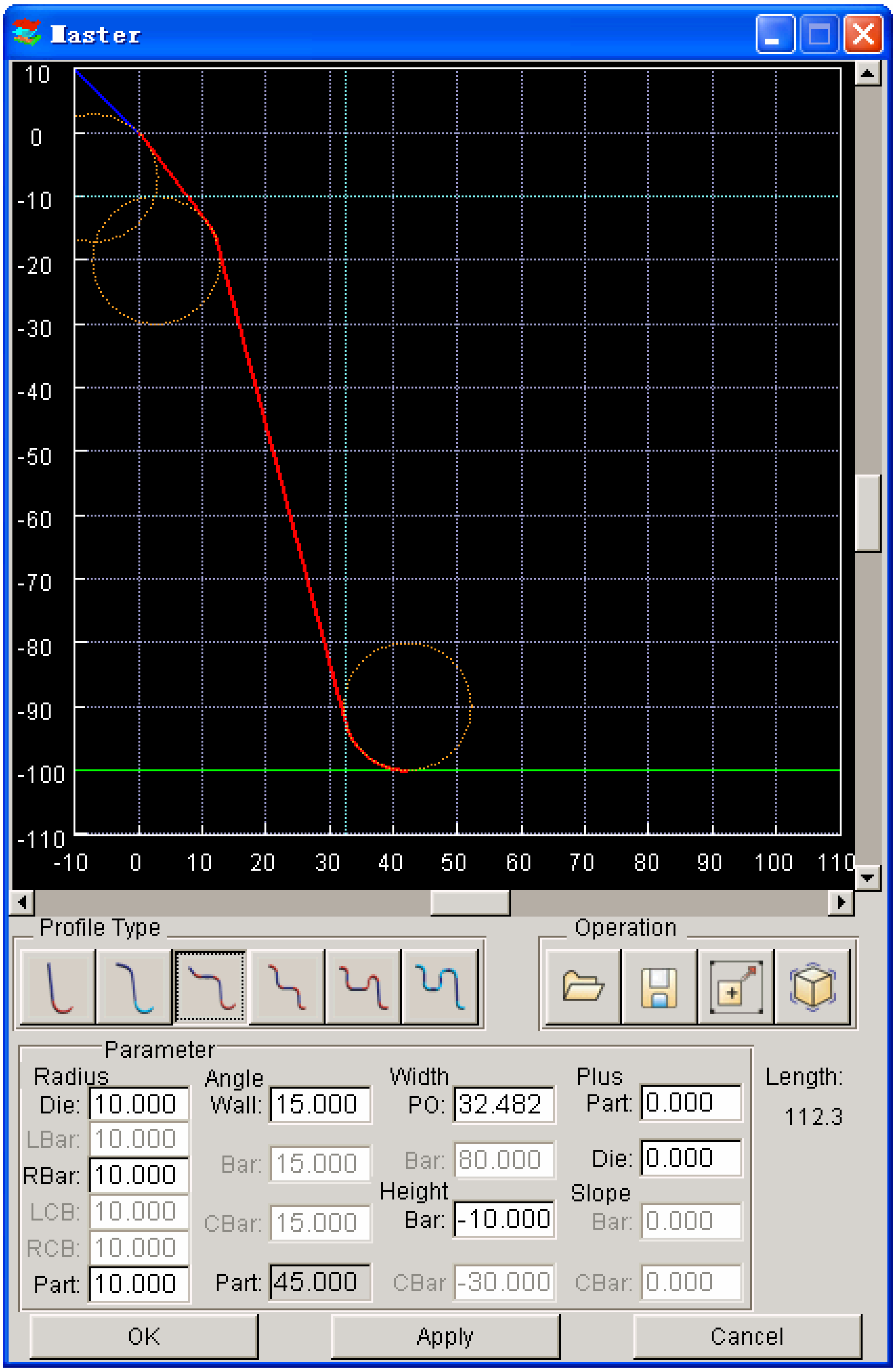


Figure 6.3.6 Profile Type 3

TYPE 4

Type 4 profile is similar to Type 2 with an additional step in the middle of the profile. **CB height** is measured from part boundary. **Bar width** and **Bar height** are deactivated. Figure 6.3.7 shows the **TYPE 4 PROFILE** dialog window.

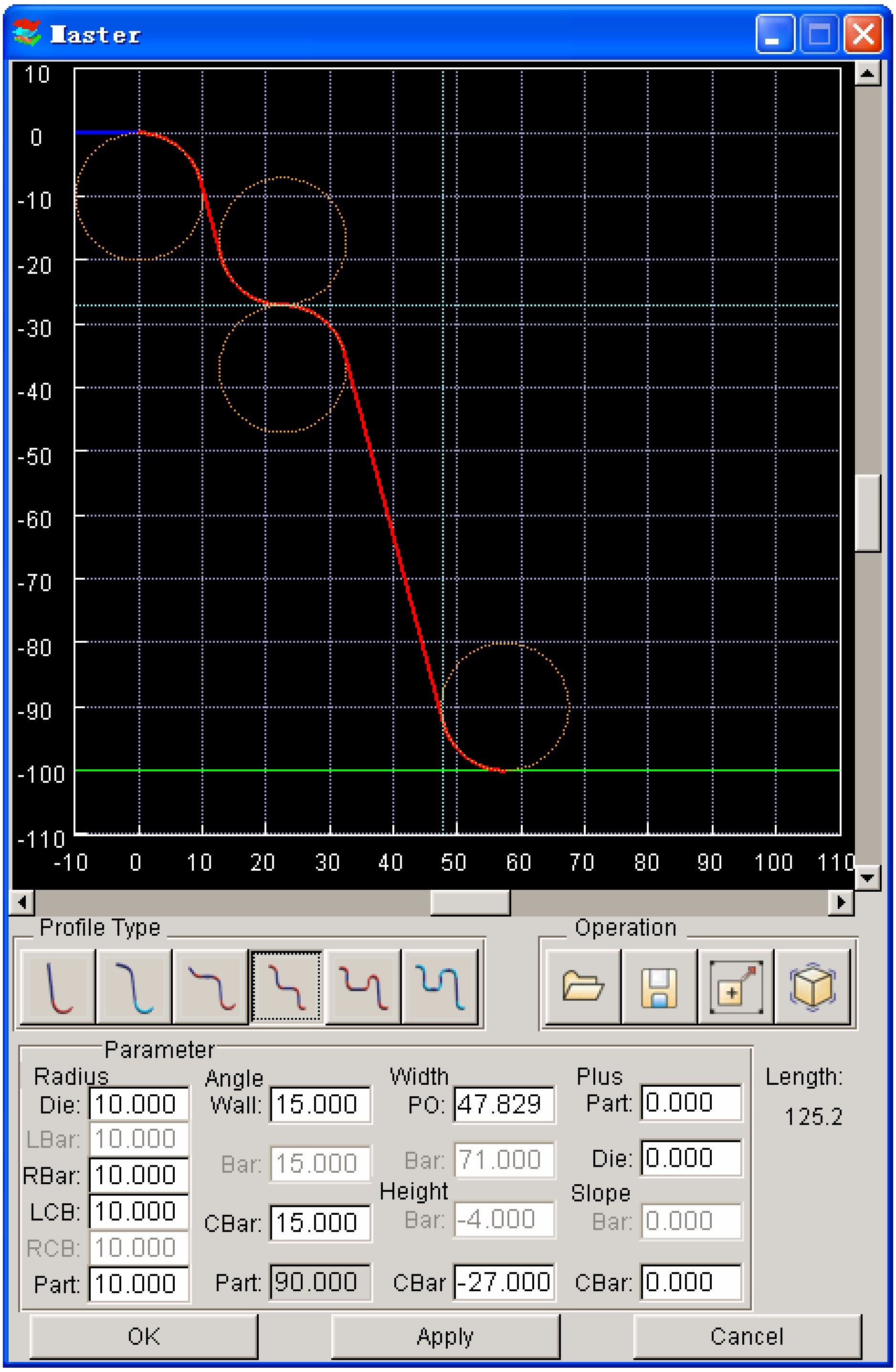


Figure 6.3.7 Profile Type 4

TYPE 5

Type 5 profile is similar to Type 4 with the addition of a simple drawbar. **Bar width** and **CB height** are deactivated as shown in Figure 6.3.8.

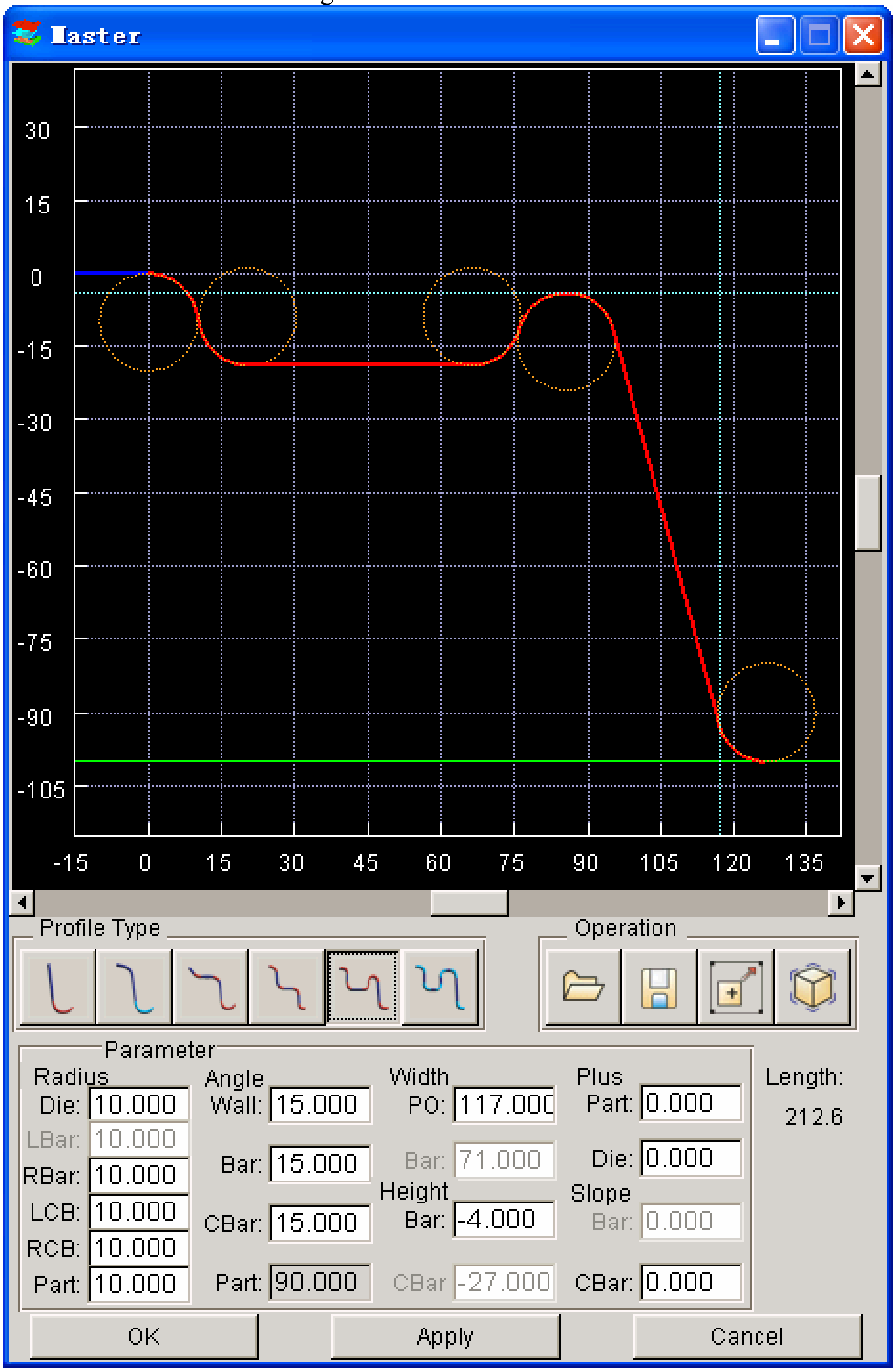


Figure 6.3.8 Profile Type 5

TYPE 6

Type 6 profile is similar to Type 5 with the addition of a full-featured drawbar. All parameters are activated as shown in Figure 6.3.9.

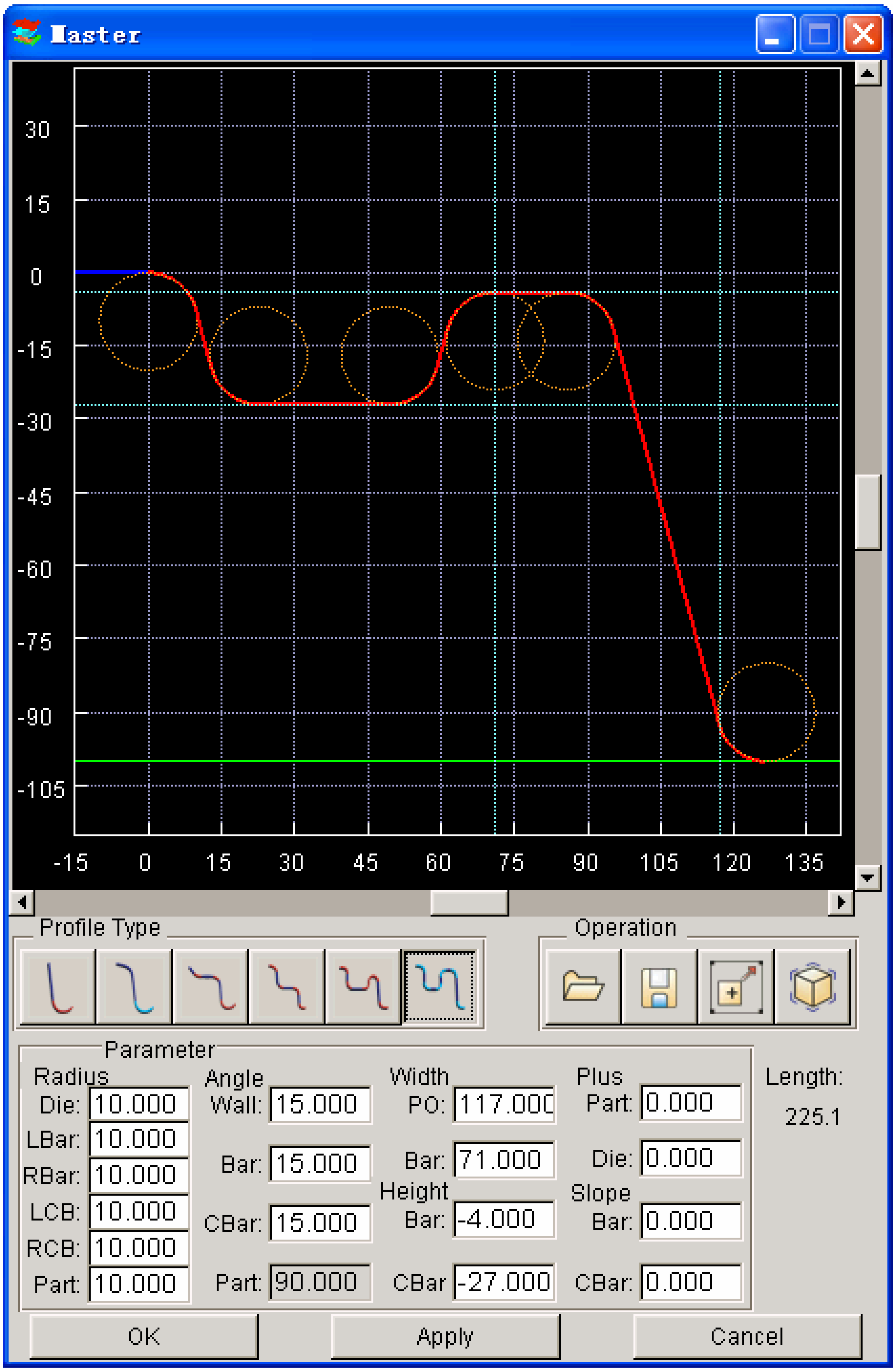
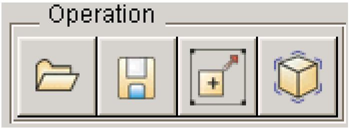


Figure 6.3.9 Profile Type 6

OPERATION 

The functions under the OPERATION group 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.

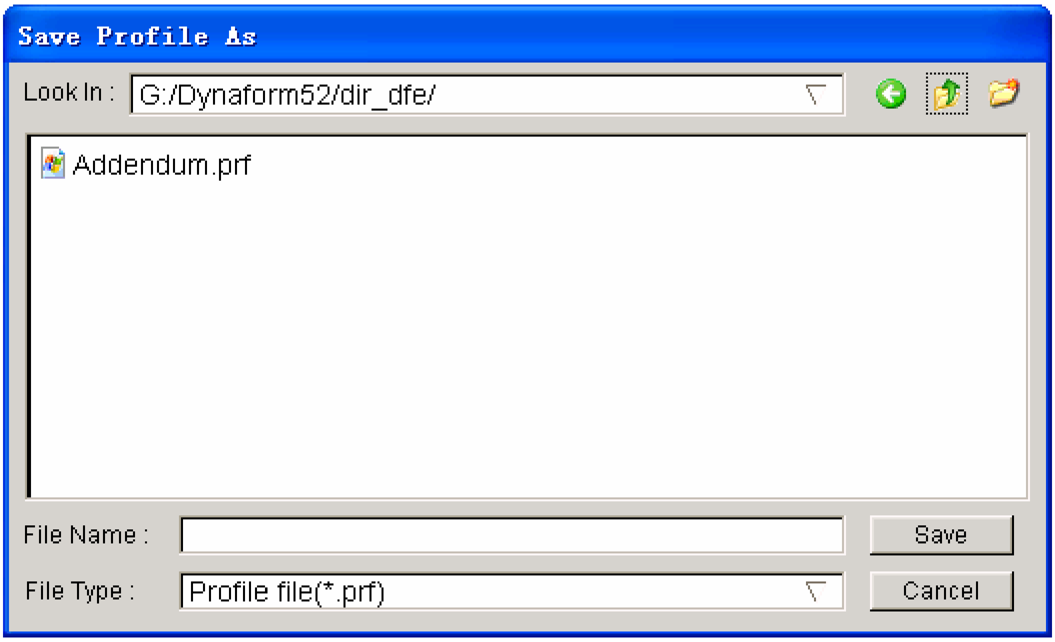
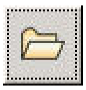
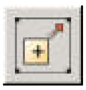


Figure 6.3.10 Save Profile Parameters in the File

 **OPEN**

This function enables the user to open a profile file (*.prf) to import the parameters into the current master profile.

 **WINDOW ZOOM**

Enables the user to define the corners of the zoom window by positioning the cursor on the profile display window to zoom in a part of the profile.

 **FILL**

Automatically rescales the profile to fill the profile display window.

After selecting **NEW** to add a Master Profile, the name of the profile will be added in the MASTER PROFILE LIST window as shown in Figure 6.3.11.

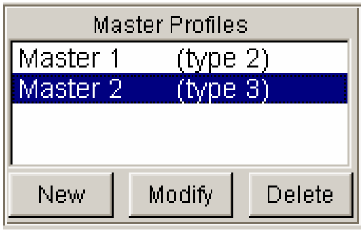


Figure 6.3.11 Master Profile Operations

Select a Master Profile from the list, and click **MODIFY**. The program will open a PROFILE window as in Figure 6.3.2. The user may modify the selected Master Profile in this window.

Select a Master Profile from the list, and click **DELETE** to delete the selected Master Profile.

Note: If there is an addendum based on the modified master profile, DYNAFORM will automatically update the addendum after the master profile is modified. If a segment of the addendum is based on the master profile, this master profile can not be deleted until the segment be deleted first..

6.3.3 ADDENDUM

This menu provides functions to create addendum from the master profile, edit and modify the addendum, etc. The available functions in the ADDENDUM menu are shown in Figure 6.3.12.

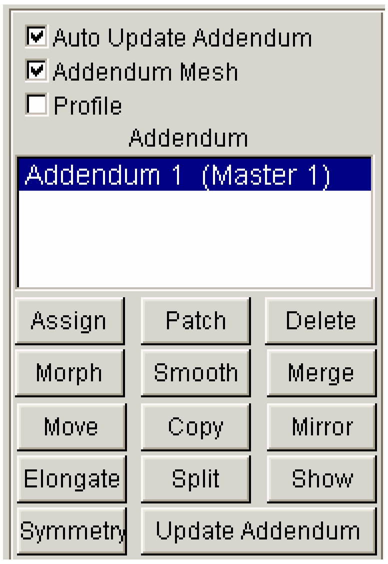


Figure 6.3.12 Create and Edit Addendum

The following three options allow the user to control the update and display of the addendum and profile. Each of the options can be selected individually any time during the addendum generation.

AUTO UPDATE ADDENDUM (toggle)

If this option is toggled **ON,** the addendum will be automatically updated when the profile is modified.

ADDENDUM MESH (toggle)

If this option is toggled **ON**, the addendum mesh will be displayed.

PROFILE (toggle)

If this option is toggled **ON**, the profiles will be displayed.

6.3.3.1 ASSIGN

eta/DYNAFORM provides two methods to create the addendum. If the **BY SEGMENTS** option is toggled **ON**, the user is can create the addendum by multiple segments with different master profiles. If the **BY SEGMENTS** option is toggled **OFF**, the entire addendum will be created based on a master profile.

After the user selects a Master profile and clicks the **ASSIGN** button as in Figure 6.3.12, DYNAFORM opens a dialog window (Figure 6.3.13).

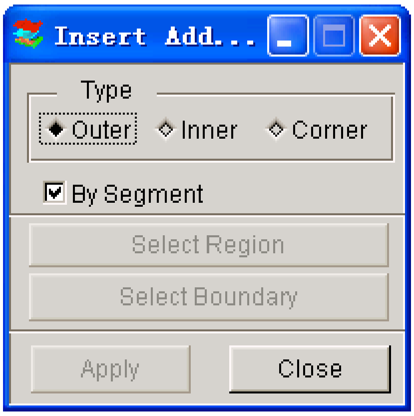


Figure 6.3.13 Insert Addendum

Select one of the options, Outer, Inner or Corner addendum, and click the **APPLY** button.

OUTER ADDENDUM (toggle on Outer)

If **BY SEGMENTS** is toggled **ON**, the **SELECT REGION** button is activated. Click **SELECT REGION,** and the program highlights the outer boundary of the part and prompts with the following message.

SELECT A NODE ON PART BOUNDARY AS STARTING POINT

After selecting a node on the part boundary, the program will prompt the user to select the end point of the region.

SELECT A NODE ON PART BOUNDARY AS END POINT

After two nodes are selected on the outer boundary, click **APPLY** to create a partial addendum in the defined region. Figure 6.3.14 illustrates a typical partial addendum.

If **BY SEGMENT** is toggled off, clicking the **APPLY** button will create the addendum around the entire outer boundary. Figure 6.3.15 illustrates a typical complete addendum of a part.

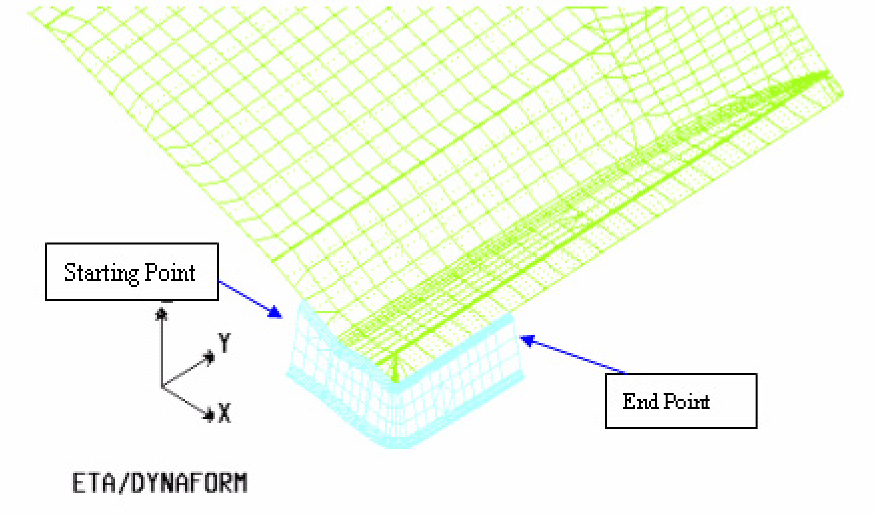


Figure 6.3.14 Create Addendum by Segment

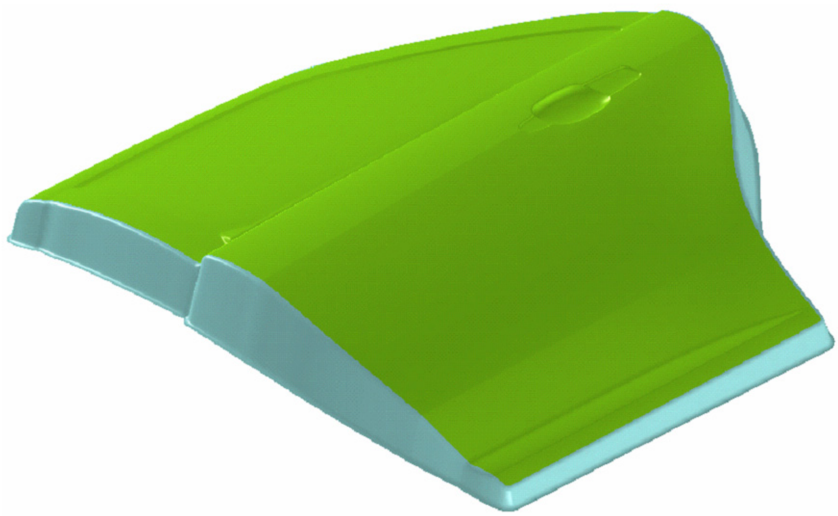


Figure 6.3.15 Create the entire Addendum by a master profile

INNER ADDENDUM (toggle on Inner)

The **SELECT REGION** button will be activated when the **INNER** option is toggled **ON**. If **BY SEGMENTS** is toggled **ON**, click the **SELECT REGION** button to select two nodes on the same inner boundary. Click **APPLY** to generate the partial addendum in the selected region. If **BY SEGMENTS** is toggled **OFF**, click the **SELECT REGION** button to select a node on an inner boundary.

CORNER ADDENDUM (toggle on Corner)

This function is used to connect a part with flange and the addendum. In this case, the flange may be part of the binder. Refer to Figure 6.3.17 for a typical part with flange on the binder. Follow these steps to create a corner addendum.

1. When the **CORNER** option is selected, the dialog window in Figure 6.3.13 will change to Figure 6.3.16.

2. Click the **SELECT PROFILE** button to select a profile on the addendum boundary. The selected profile will be highlighted as shown in Figure 6.3.18.

3. Select two nodes on the part boundary as shown in Figure 6.3.19. The program will prompt the user to conform if the defined corner is correct. Click **YES** to accept the defined corner region.

4. The **APPLY** button in Figure 6.3.15 is activated. Click **APPLY** to generate the corner addendum as shown in Figure 6.3.20.

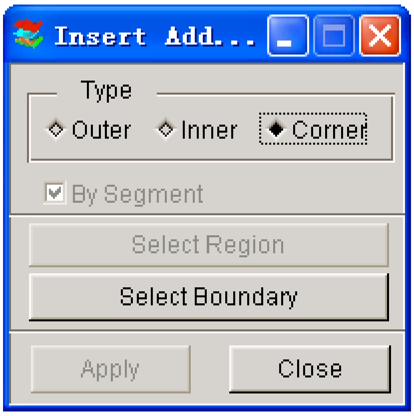


Figure 6.3.16 Corner Addendum

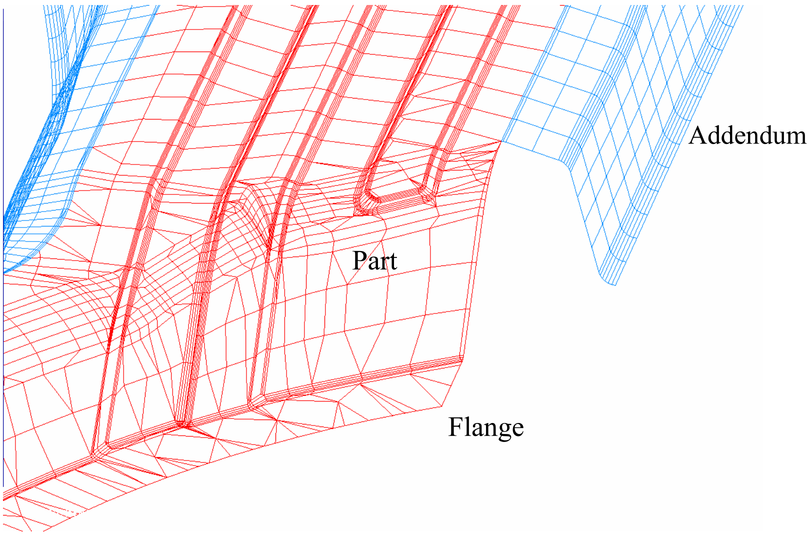


Figure 6.3.17 Flange on binder surface

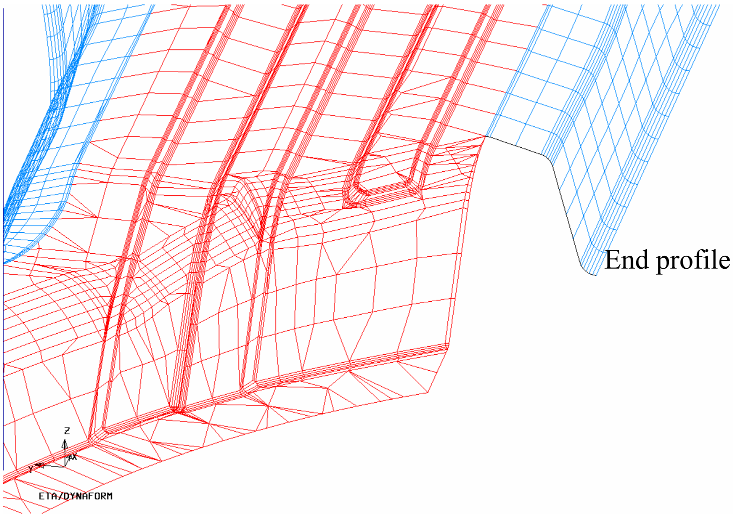


Figure 6.3.18 Select end profile on addendum

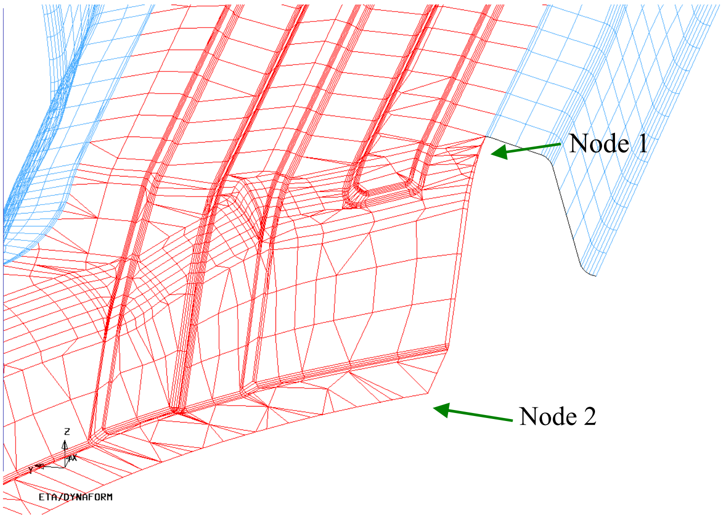


Figure 6.3.19 Select two nodes on part boundary

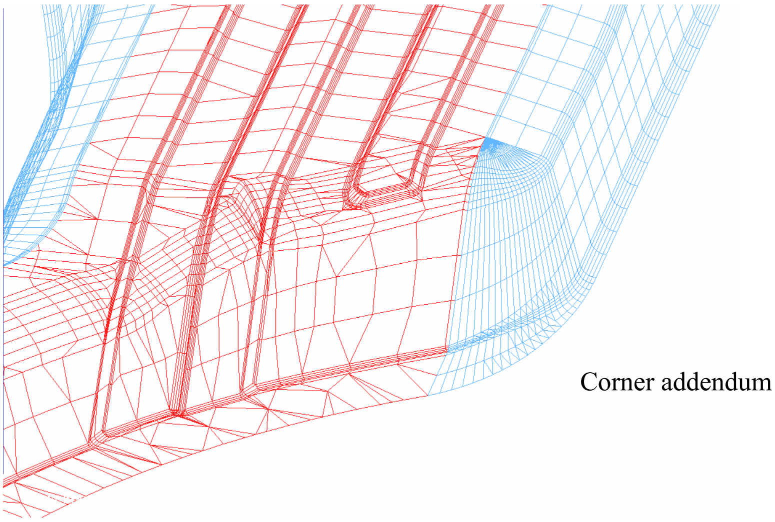


Figure 6.3.20 Generated corner addendum

6.3.3.2 PATCH

This function is used to fill in a section of the addendum between two adjacent regions. The filled part of the addendum is typically a transition area between two adjacent regions generated with different profile types. Figure 6.3.21 shows typical addendum regions of different profile types.

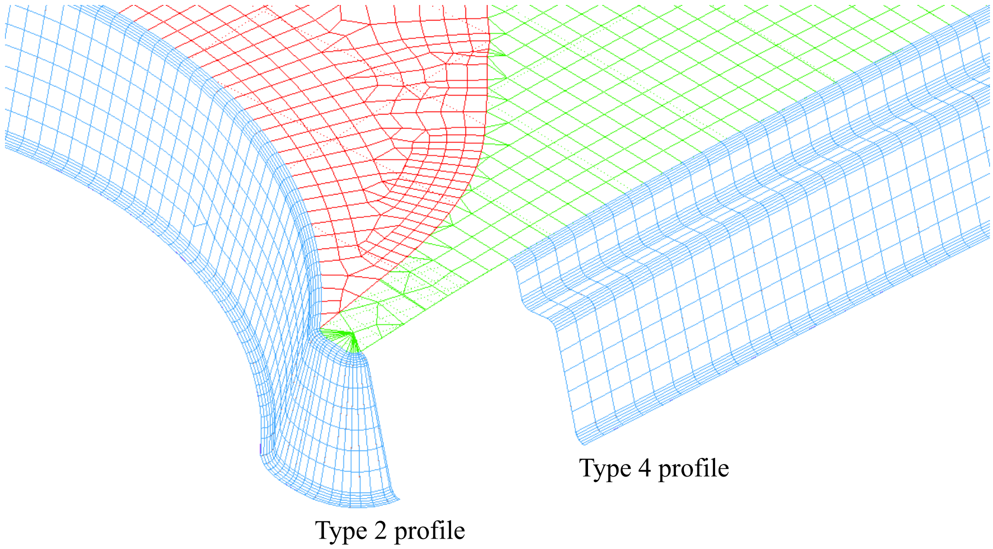


Figure 6.3.21 Transition area between two adjacent regions

Using this function, the program prompts the user to select two end profiles of adjacent regions. After the profiles are selected, DYNAFORM will fill in the portion of the addendum between the selected profiles. Figure 6.3.22 shows a typical transition area generated by the patch function.

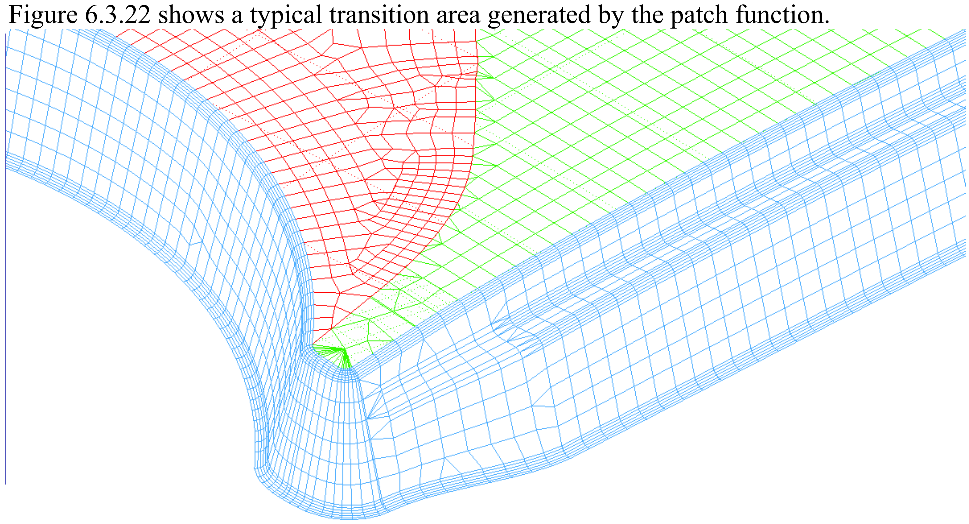


Figure 6.3.22 Typical result of Patch function

6.3.3.3 DELETE

This function enables the user to delete the selected addendum. The user selects an addendum from the ADDENDUM LIST window and clicks the **DELETE** button in as shown in Figure 6.3.12. The program will highlight the selected addendum and display a dialog window as shown in Figure 6.3.23 to prompt the user to confirm the deletion of the addendum.

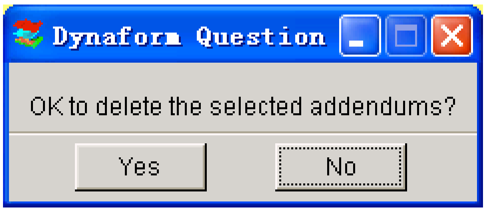


Figure 3.3.23 Confirmation to delete addendum

Click **YES** to delete the selected addendum or **NO** to reject the selection.

6.3.3.4 MORPHING

This function enables the user to modify a segment of addendum by morphing the POP Line, Bar Height, Bar Width, CBar Height and Wall Angle. Click one of the options and then the **APPLY** button, DYNAFORM will display a dialog window as in Figure 6.3.24. The program will display the profiles in red and the POP Line in blue as shown in Figure 6.3.25. The Button Update Addendum used to update the addendum after the operation of the Morphing.

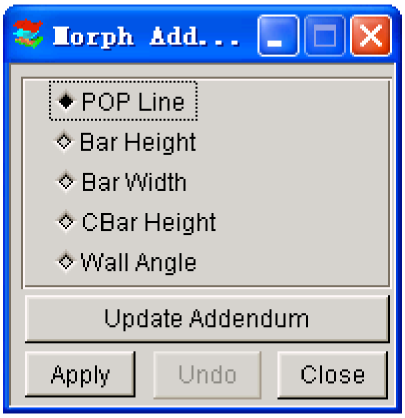


Figure 6.3.24 Morphing Addendum Options

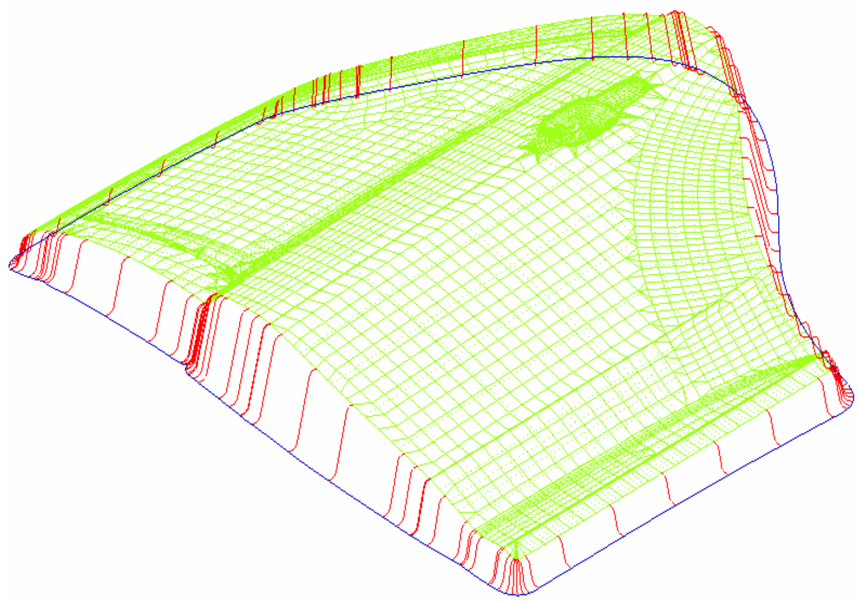


Figure 6.3.25 Profiles and POP Line

POP Line

This function allows the user to morph part of the POP Line. The program will prompt the user to select the starting profile and end profile to define the region of the POP Line to be morphed. If there are more than one profile region defined in the addendum, the program will highlight the starting and end profile in each region with a circle at the profile. After selecting two profiles, DYNAFORM will prompt the user to confirm the region and will highlight the POP Line in the region as in Figure 6.3.26. The operation of morphing the POP Line is same as the LINE MORPHING function described in Section 6.4.1. Figure 6.3.27 shows the morphed POP Line and the related profiles. After the POP Line is morphed, click the **CLOSE** button to exit the dialog window. The program will update the profiles and addendum according to the morphed POP Line. Figure 6.3.28 shows the updated addendum.

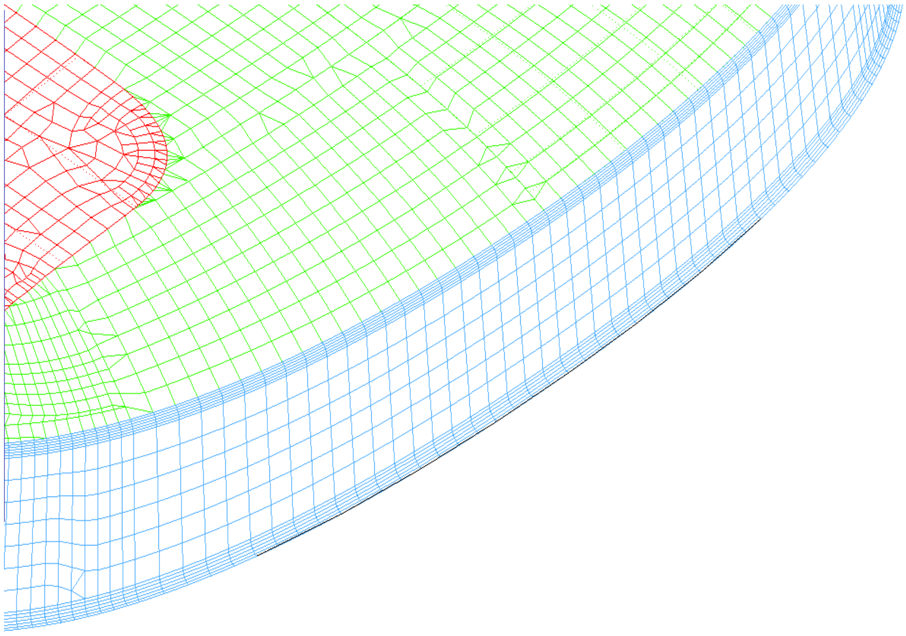


Figure 6.3.26 Selected POP Line for morphing the POP Line

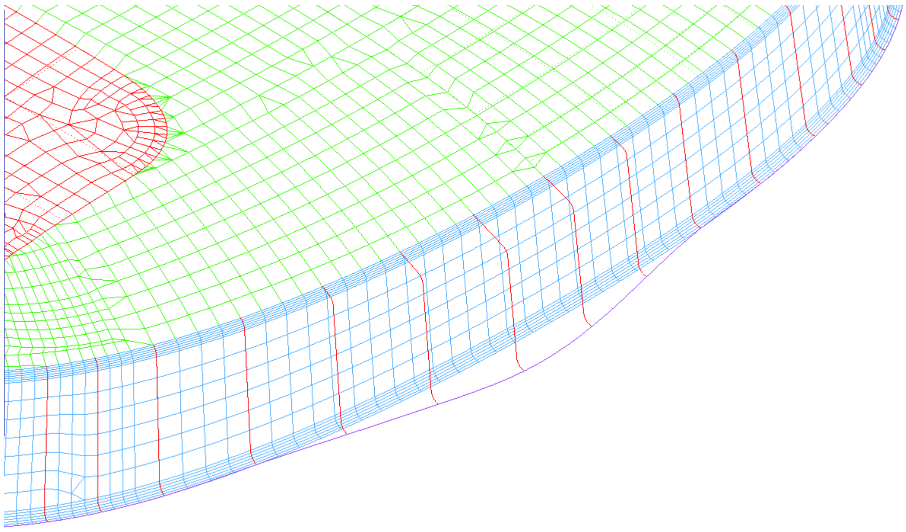


Figure 6.3.27 Morphed POP Line and profiles

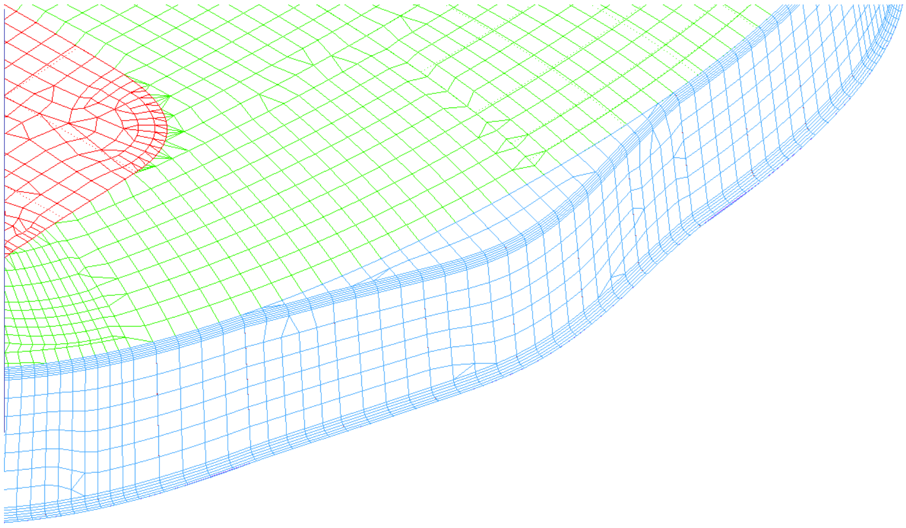


Figure 6.3.28 Updated addendum according to the morphed POP Line and profiles

BAR HEIGHT

This function allows the user to morph the drawbar height of the profiles in the user-defined region. The program will prompt the user to select the starting profile and end profile to define the region. The operation is similar to POP Line morphing except that it morphs the control line on the top of the drawbar 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 program will prompt the user to select the starting profile and the end profile to define the region. This operation is similar to BAR HEIGHT line morphing except that it morphs the control line on the top of the drawbar 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 program will prompt the user to select the starting profile and the end profile to define the region. This operation is similar to POP Line morphing except that it morphs the control line on the bottom of the counter bar 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 program will prompt the user to select the starting profile and the end profile to define the region. This operation is similar to POP Line morphing except that it morphs the POP Line to alter the wall angle of the profiles.

6.3.3.5 SMOOTH

This function enables the user to modify a segment of addendum by smoothing the POP Line, Bar Height, Bar Width, CBar Height and Wall Angle. The program will display a dialog window similar to Figure 6.3.24 and display the profiles and the POP Line in the graphic window as shown in Figure 6.3.25.
Select the **SMOOTH** option, click the **APPLY** button as in Figure 6.3.24, and the program will display a dialog window as shown in Figure 6.3.29.

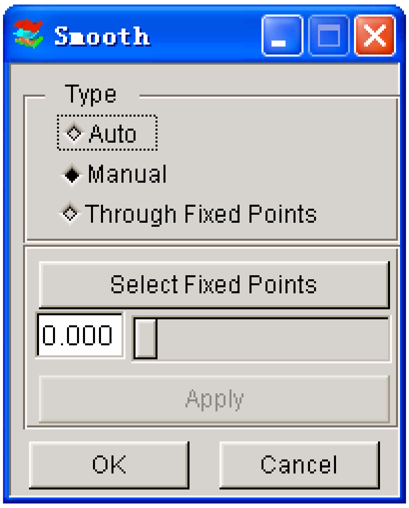


Figure 6.3.29 Smooth Options

The smoothing operation of each option is similar to the morphing function except that the control line is smoothed instead of morphed.

The following example of smoothing the POP Line demonstrates different smooth methods.
There are three methods to smooth the POP Line. Please refer to Section 6.2.1, EDIT BINDER, for a detailed description of smooth methods.

AUTO SMOOTH

Smooths the entire POP Line automatically.

MANUAL SMOOTH

Smooths the entire POP Line by moving the slider. The SELECT FIXED POINTS option enables the user to select more than one fixed point. These selected points will not be moved during the smooth operation. A typical result of MANUAL SMOOTH WITH FIXED POINTS is shown in Figure 6.3.30.

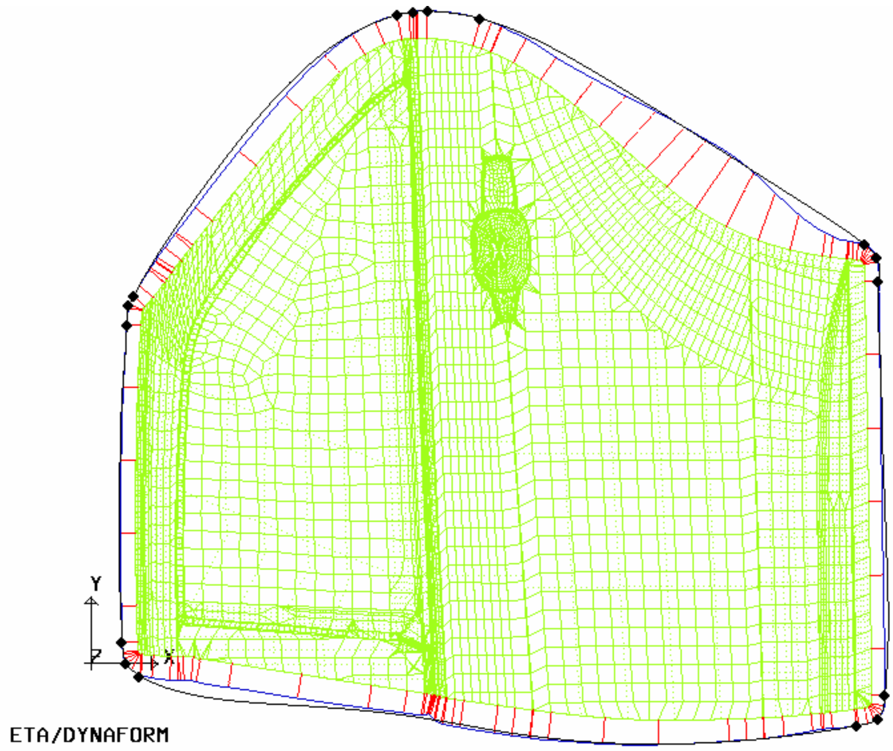


Figure 6.3.30 Manual Smooth With Fixed Point

THROUGH FIXED POINTS

This function allows the user to smooth a segment of the POP Line by selecting a number of fixed points on the POP Line. Click the **PREVIEW** button to view the smoothed POP Line. If the smoothed shape is acceptable, click **APPLY** to accept the smoothed POP Line. Click **OK** to exit the SMOOTH dialog window. Click the **CLOSE** button in the SMOOTH ADDENDUM dialog window to exit the menu. The program will update the profiles and the addendum according to the smoothed POP Line.

6.3.3.6 MERGE

This function enables the user to merge (or map) the control line of the POP Line, Bar Height, Bar Width, Wall Angle or CBar Height to a selected line. DYNAFORM will move the profiles according to the distance from the control line to the selected line.

POP LINE

Modifies profiles according to the merged POP Line.

BAR HEIGHT

Modifies profiles according to the merged control line of Bar Height.

BAR WIDTH

Modifies profiles according to the merged control line of Bar Width.

CBAR HEIGHT

Modifies profiles according to the merged control line of Cbar Height.

WALL ANGLE

Modifies profiles according to the merged control line of Wall Angle.

Figure 6.3.31a illustrates the definition of a straight line to which the POP line can merge. Figure 6.3.31b shows the result of the profiles after the POP Line is merged to the selected line. The program will update the addendum mesh after the MERGE ADDENDUM dialog window is closed.

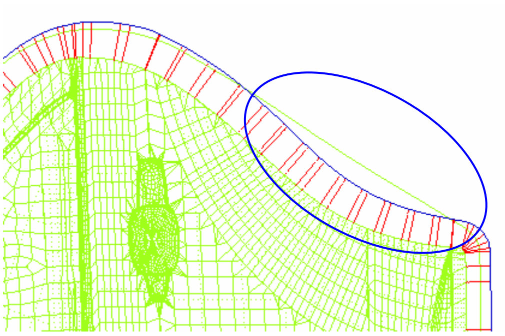


Figure 6.3.31a　Merge POP Line

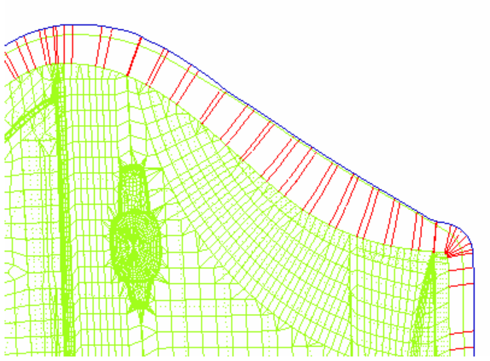


Figure 6.3.31b Merge POP Line

6.3.3.7　MOVE

This function enables the user to move a region of addendum to another location. If there are more than one addendum regions in the database, DYNAFORM will prompt the user to select a profile group.

PICK A PROFILE GROUP

The program will highlight the profiles and addendum in the selected group as shown in Figure 6.3.32a. The program will then prompt the user to select a node on the part boundary as the starting point for moving the addendum.

PICK NODE ON DIE BOUNDARY AS DESTINATION POSITION

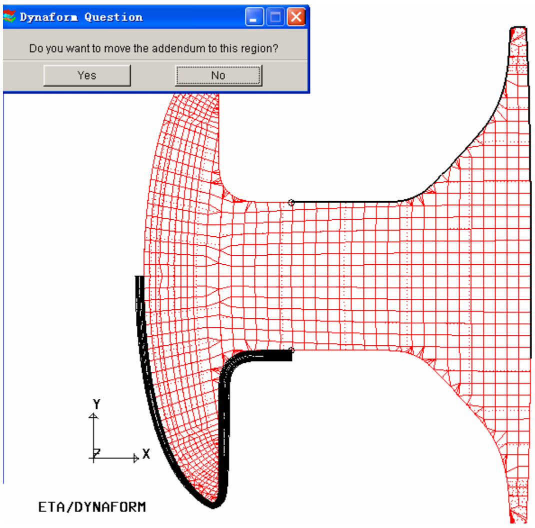


Figure 6.3.32a Move Addendum

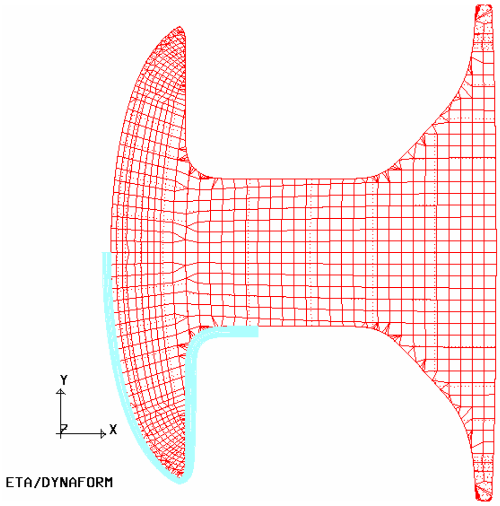


Figure 6.3.32b Move Addendum

DYNAFORM will open a DYNAFORM QUESTION window to confirm the moved location and will highlight the new boundary on the part, Figure 6.3.32b. If the location of the new boundary is not correct, select **NO** to put the new boundary on the opposite side of the reference point as shown in Figure 6.3.32c. Click **YES** to accept the boundary location. The selected addendum will be moved as shown in Figure 6.3.32d.

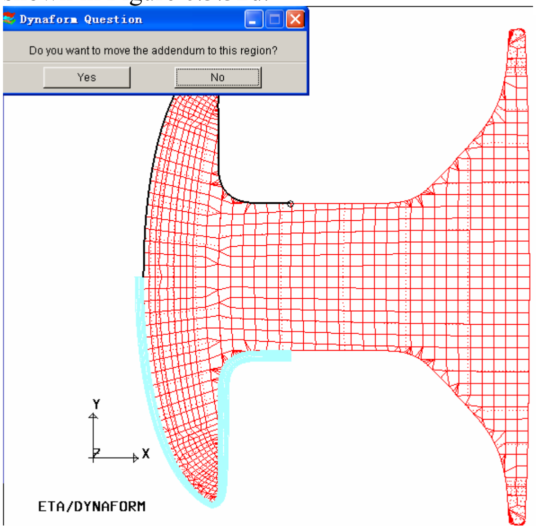


Figure 6.3.32c Move Addendum

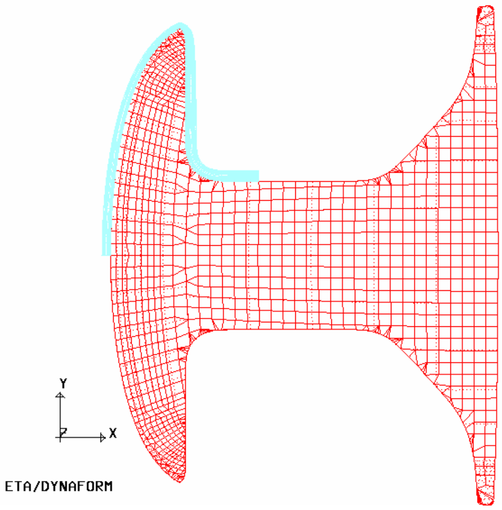


Figure 6.3.32d Move Addendum

6.3.3.8 COPY

This function copies the selected profiles and addendum to another location on the part boundary. The operation procedure is same as the MOVE function except that the original addendum is kept in the database.

6.3.3.9 MIRROR

This function generates the entire addendum defined for half symmetry by mirroring it about the symmetry plane. The user must define the symmetry axis for the part in the DFE/PREPARATION menu in order to use this function. Refer to Section 6.1.2, GEOMETRY TYPE, for the symmetry definition.

The mirror function of the program displays the symmetry axis. Select a region of addendum, and the program will display the mirrored boundary and prompt the user for acceptance as shown in Figure 6.3.33a. Click **YES** to accept. The mirrored result is shown in Figure 6.3.33b.

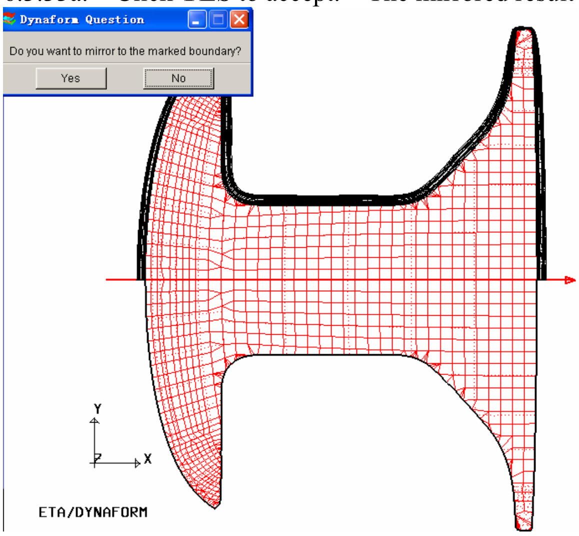


Figure 6.3.33a Move Addendum

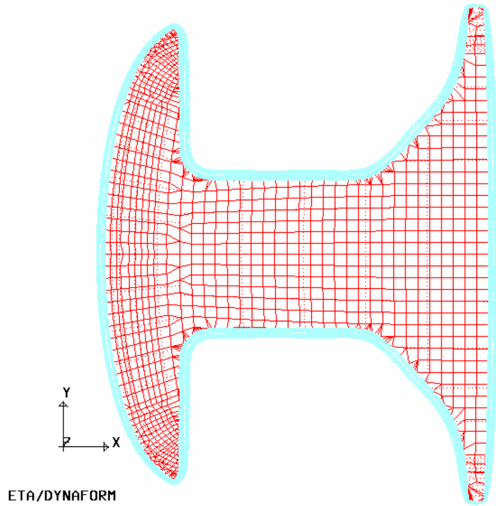


Figure 6.3.33b Move Addendum

Note: If the generated addendum by the MOVE, COPY or MIRROR functions interferes with existing addendum, the operation will be aborted.

6.3.3.10 ELONGATE

This function is used to estimate the elongation ratio of the blank along the specified profile direction.

If user selects this function, the program will prompt the user to select a profile. After selecting a profile, there are two section lines will highlight which created virtually by the plane of profile intersecting with the binder and the die. See the Figure 6.3.34.

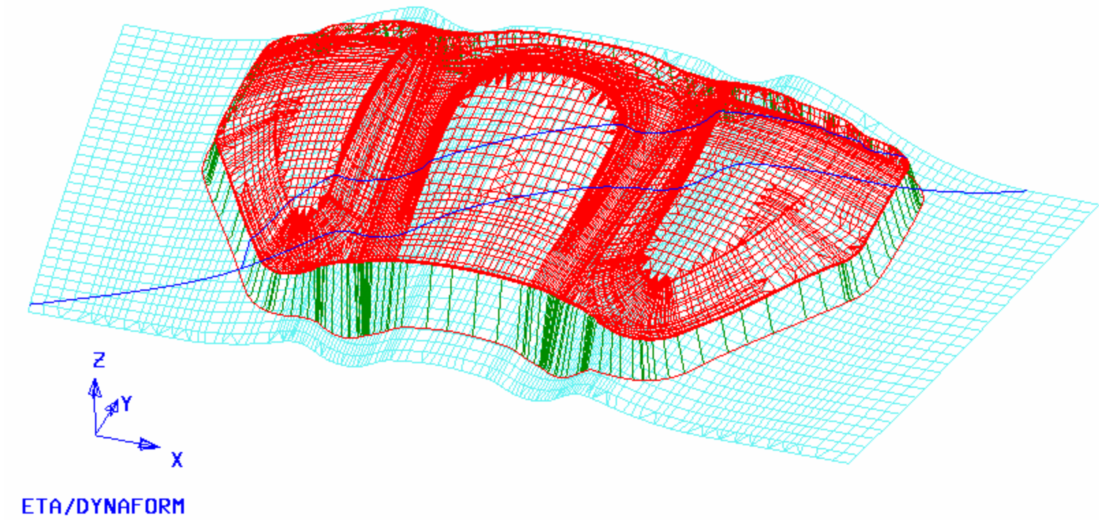


Figure 6.3.34 Select the profile

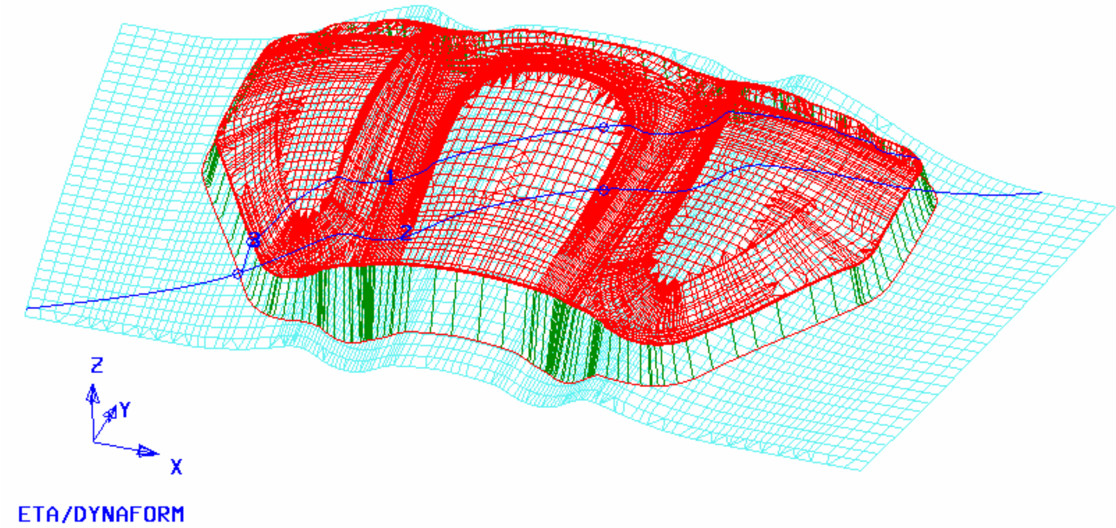


Figure 6.3.35 Click a point on the highlight line of die

Then the program will prompt the user to select a reference point on the highlighted line or click the middle button to select the default point. If selecting a point on the highlighted line of die. See the Figure 6.3.35. Those lines will be divided into three lines identifying with number 1, 2, 3 respectively and in the PROMPT MESSAGE window eta/DYNAFORM will give the result in the following format:

ELONGATION: 5.8%; LENGTH: 1: 434.1 2:521.7 3:207.7

The ELONGATION means the elongation percentage of the specify section line after deforming. It can be calculated by:

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

LENGTH 1 and LENGTH 3 mean separately the length of the punch and addendum after deforming. LENGTH 2 means the length of the line 2 before deforming.

After giving those massage like above, the program will prompt the user to select another profile to continue. The user can go on or click right button to exit this function.

This function is useful for user to help themselves to estimate in advance the level of deform for blank.

6.3.3.11 SPLIT

This function enables user to divide a region of addendum into two independent regions. This function is very useful for user who only wants to modify the addendum locally but not to change the shape of other addendums. If there is only one addendum region in the database, eta/DYNAFORM will prompt the user to select two different profiles to define the region of addendum. If there are more than one addendum region, the user need selects only one profile in a region. The program will automatically split the region into two regions with it. After split it will add a new addendum name in ADDENDUM window and add new a master profile type in Master Profiles automatically.

6.3.3.12 SHOW

This function can help the user to identify the addendum type with ID number. If the user click **Show** button, the different addendum will be identified with different ID number which will be displaying in the center of the corresponding addendum.

6.3.3.13 SYMMETRY

This function enables user to adjust the orient of profile paralleled to the symmetry plane. Before mirror an addendum for a symmetric part, it had better adjust the profiles whose boundary nodes in symmetry plane. Selecting this function, the program will prompt the user to select a profile to change orientation. If the user selected a profile with a boundary node in symmetry plane, program will change the orient of the profile to the symmetry plane automatically and prompt user continually to select other profiles. Figure 6.3.36 shows the result of the profiles before/after adjusting the boundary profile.

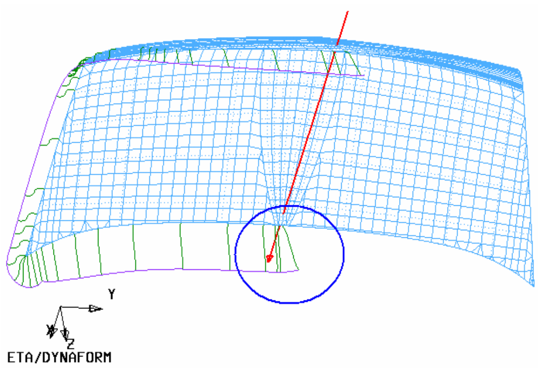


Fig.6.3.36a Before Symmetry the profile

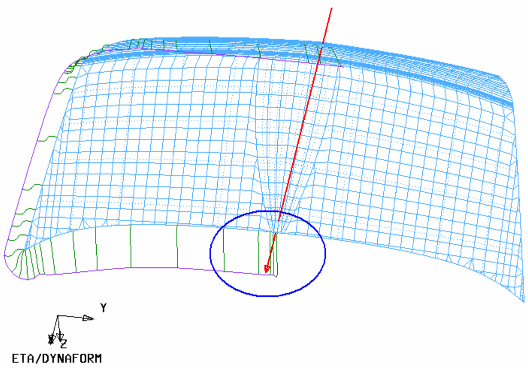


Fig.6.3.36b After Symmetry the profile

If the user selecting the profiles that have not boundary node in symmetry plane, the program will open an eta/DYNAFORM QUESTION window to confirm this operation as show in Fig 6.3.37. Click **YES** to continue this operation. The selected profile will change it orient to parallel to the symmetry plane. Click **NO** to cancel this operation and the user can reselect another profile.

Figure 6.3.37 eta/DYNAFORM QUESTION windows

6.3.3.14 UPDATE ADDENDUM

If the AUTO UPDATE ADDENDUM option is not selected, DYNAFORM will not update the addendum after the user modifies the profiles. 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 will not be necessary.

6.3.4 PROFILE

The profile functions enable the user to INSERT, DELETE, MODIFY or ORIENT one or more profiles. Figure 6.3.38 shows the functions in the PROFILE section.

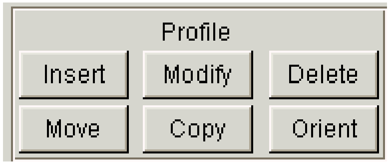


Figure 6.3.38 Profile Options

6.3.4.1 INSERT

This function enables the user to insert a profile between two adjacent profiles in the same profile group. The program displays the profiles and POP Line and prompts the user to select the first profile.

SELECT FIRST PROFILE

After the first profile is selected, the program prompts the user to select the second profile.

SELECT SECOND PROFILE

After two adjacent profiles are selected, the program prompts the user to select a point on the part boundary.

SELECT NODE ON PART AS STARTING POINT

After the user selects a node on the part boundary, the program prompts the user to select a point on the POP Line

SELECT A POINT ON POP LINE

After a point on the POP Line is selected, the program inserts a profile between the boundary node and the point on the POP Line. The inserted profile follows the same profile type of the original profile.

6.3.4.2 DELETE

This function enables the user to delete one or more profiles. The program displays a dialog window as in Figure 6.3.39.

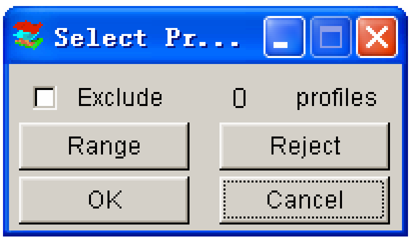


Figure 6.3.39 Select Profile

The DEFAULT option allows the user to select profiles one at a time by using the cursor on the screen.

RANGE

If RANGE is clicked, the program prompts the user to select the starting profile.

SELECT STARTING PROFILE

The program then prompts the user to select the end profile.

SELECT END PROFILE

The program highlights the profiles between the start and end. Select OK to delete 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 function without deleting any profiles.

If the **EXCLUDE** option is selected, the selected profile will be excluded from the previous selection.

6.3.4.3 MODIFY

This function enables the user to modify a profile and automatically adjust the adjacent profiles to obtain a smooth POP Line.

- Select two profiles to define the region.
- Select a profile with the cursor, or use the RANGE option to select a group of profiles in this region.
- After the profiles are selected, the program will display the PROFILE window as in Figure 6.3.2. The user can change profile types and/or change profile parameters. As profiles are modified in the profile window, DYNAFORM will update the profile in the graphic window.
  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 program will automatically update the addendum mesh according to the new profiles.

6.3.4.4 ORIENTATION

The addendum consists of a number of profiles between the part boundary and the POP Line. This function allows the user to select a profile to change the orientation relative to the POP Line
The program gives the following message in the prompt area to prompt the selection of a profile.

SELECT PROFILE TO ORIENTATION

After a profile is selected, eta/DYNAFORM prompts the user to pick a location on the POP Line. The program will move the selected profile to a plane defined by the original part boundary node and the new POP Line location.
After completing orientation of the profile, click the RIGHT mouse button to exit the function. The program will automatically update the addendum mesh if the AUTO UPDATE ADDENDUM option is selected.

6.3.4.5 COPY

This function enables user to copy an existent profile to the other different location. After have selecting this function, program will prompt user to select a profile. Then the selected profile will be highlighted and continually prompt user to select a node on the boundary of die as the starting point. And then select a point on the POP line as the second point. After have allocating two position, the specify profile will be copied to the new position

6.3.4.6 MOVE

The function of **Move** is similar to the **Copy** except that after moving the existent profile will be moved to a new position and original profile will be deleted.
Note: Two ends profiles of a region addendum could not be moved.

6.3.5 CREATE ADDENDUM SURFACE

This function automatically creates the surfaces based on the profiles and the POP Line. The created surfaces are included in the part named ADDENDUM.

6.3.6 CREATE PO LINE

This function enable user to create a PO (Punch Open) line in current part. The created lines are included in the part named ADDENDUM.

6.3.7 CREATE PRF LINE

This function enables user to create some geometric lines based on the profiles. Thus user can export those lines data to other CAD software for further edit. The created lines are included in the part named ADDENDUM.

6. 4 MODIFICATION

Functions in this menu are used to modify line, surface or element and to complete Die Face design. There are eight functions in this menu as shown in Figure 6.4.

Figure 6.4 Modification Options

6.4.1 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. The program opens the SELECT LINE window to select a line to morph.
2. After selecting a line, the program displays the Morphing Line dialog window as shown in Figure 6.4.1.

   - **MORPHING TYPES**

     There are three types of morphing: NO CONSTRAINT, KEEP C ANGLE and KEEP E ANGLE. For a detailed description of the morphing types, refer to BSE/MODIFY BINDER/MORPHING in Section 6.2.1. Click the arrow button to display the drop down selection.

   - **SELECT MORPH SEGMENT**

     This option allows the user to select a segment of the selected line to morph. The program will prompt the user to select two points on the line to define the morph segment. If no morph segment is defined, the program will morph the entire selected line.

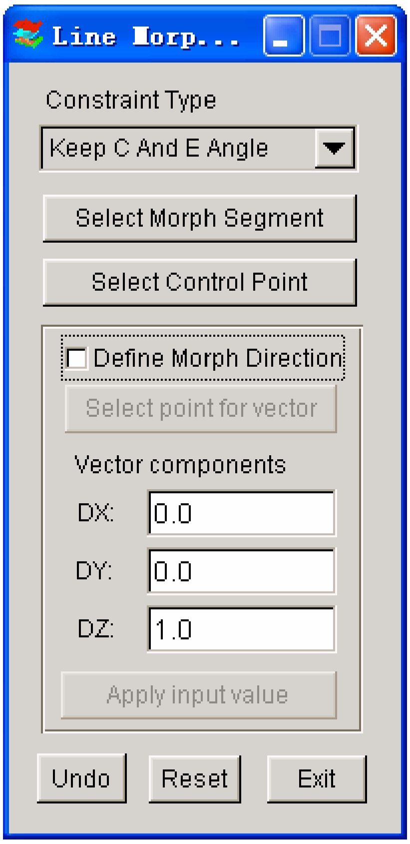


Figure 6.4.1 Line Morphing window

- **SELECT CONTROL POINT**

  The user must select a control point to morph the line. Click the left mouse button to select new control point, middle mouse button to use the previous control point, or right mouse button to return to cancel the selection. Once the control point is selected, the program displays the default morph vector at the control point. Move the mouse to morph the line. The program shows the morphed result in real time. When the desired morph result is obtained, click the left mouse button again to complete the morph operation. Click the right mouse button to cancel the morphing. If the user wants to define another morph direction, check the **DEFINE MORPH DIRECTION** box to activate the option.

- **DEFINE MORPH DIRECTION**

  This option allows user to define a new morphing direction. Once this option is selected, the program activates the **SELECT POINT FOR VECTOR** button and **INPUT COORDINATE** window to allow the user to define the morph direction. Click the SELECT POINT FOR VECTOR to select a point or node on the screen. The new morph direction is the vector from the control point to the selected point. The user may enter the vector components directly in the VECTOR COMPONENTS window and click **APPLY INPUT VALUE** to define the vector.

UNDO allows the user to reject the result from last morphing operation.

RESET resets the selected line to the original shape after a series of morphing operations.

3. Click **EXIT** to complete the morph operation. The program will display the SELECT LINE window to enable the user to select another line for morphing.

4. Click **CANCEL** in the SELECT LINE window to exit the function.

6.4.2 SURFACE MORPHING

This menu allows the user to morph the selected surface. There are five functions in this menu as shown in Figure 6.4.2.

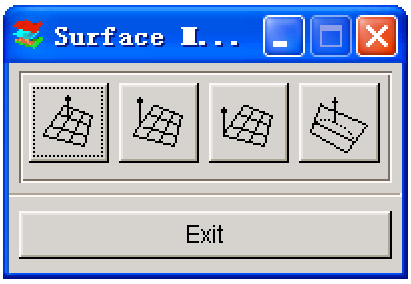


Figure 6.4.2 Surface Morphing Methods

1. INTERIOR MORPHING

This function allows the user to morph a surface by selecting a control point within the surface domain. The user may morph the entire surface or a region on the surface.

- SELECT SURFACE window will prompt the user to select a surface to morph.
- After selecting a surface, the program will display a dialog window as shown in Figure 6.4.3.

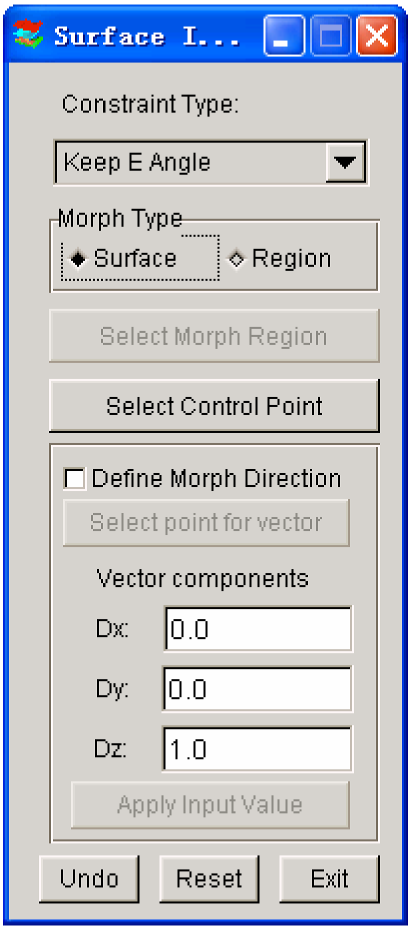


Figure 6.4.3 Surface Morphing - Interior

- The drop down button, KEEP C ANGLE, allows the user to select the type of constraint for morphing. The description of each constraint type is given in Section 6.2.1. Figure 6.2.14 illustrates the characteristics and results of each constraint type.
- If REGION is selected, the SELECT MORPH REGION button will be activated for user to select two diagonal points on the surface to define the morphing region.
- Click SELECT CONTROL POINT button to select a point on the surface (or in the region if REGION option is selected) as control point. After an interior point is selected by the cursor on the surface, the program draws the morphing vector and the U-V line at the control point. The user may click the left mouse button and move the cursor to see the morphed U-V line in real time. When the desirable shape of U-V lines is obtained, the user clicks the left mouse button to accept the morphed result. The program will update the surface according to the new U-V lines.
- The user may activate DEFINE MORPH DIRECTION option to define a new morph direction. The detailed description about this option is given in Section 6.4.1 LINE MORPHING.
- UNDO cancels the last morphing. RESET returns the surface to the original shape prior to the morphing operations.

1. EDGE MORPHING

This function uses a point on the surface boundary as the control point. The operation is the same as the INTERIOR MORPHING function.

2. CORNER MORPHING

This function uses a corner point on the surface as the control point. The operation is the same as INTERIOR MORPHING.

3. SECTION LINE MORPHING

This function enables the user to morph surface by morphing a U or V section line on the surface.

SELECT SURFACE window will prompt the user to select a surface to morph.

After selecting a surface, the program will display the U-V lines on the surface and a dialog window as shown in Figure 6.4.4.

Select the Constraint Type as in Interior Morphing function.

ADD SECTION LINE 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 by the mouse pick.

SELECT SECTION LINE allows the user to select a U or V section line as the control line.

SELECT MORPHING POINT allows the user to select a control point to morphing.

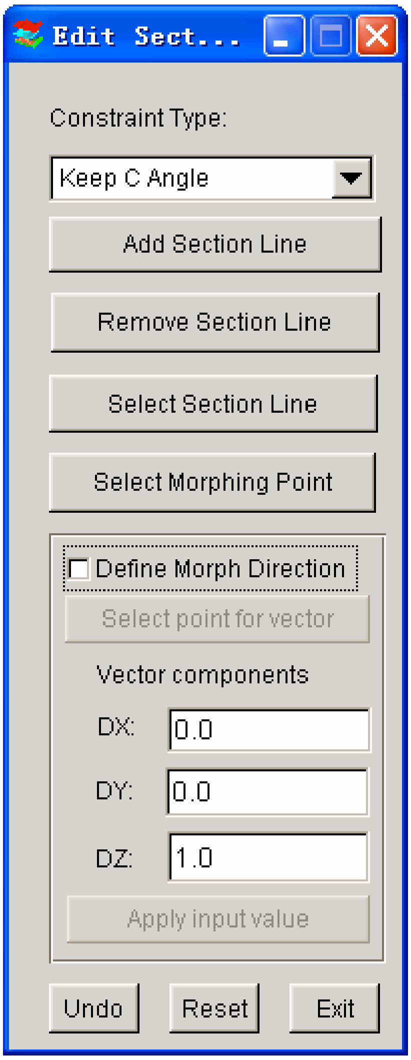


Figure 6.4.4 Section Line Morphing

- Click **SELECT MORPHING POINT** to select a control point on the selected section line. After an interior point is selected by the cursor on the control line, the program draws the morphing at the control point. The user may move the cursor to see the morphed U-V line in real time. When the desirable shape of U-V lines is obtained, the user clicks the left mouse button to accept the morphed result. The program will update the surface according to the new control line.

UNDO cancels the last morphing. **Reset** returns the surface to the original shape prior to the morphing operations.

6.4.3 ELEMENT MORPHING

This menu allows the user to morph a region of selected elements in a mesh. There are four types of morphing as shown in Figure 6.4.6. All elements within the region must be selected without excluding any elements from the region. The operation of these functions is similar to the SURFACE MORPHING functions.

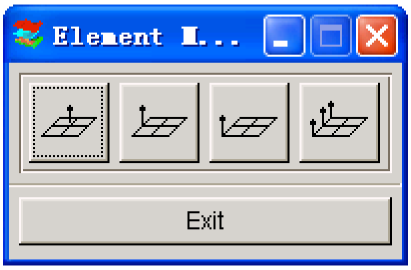


Figure 6.4.5 Element Morphing Methods

1. INTERIOR MORPHING

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

- Select a group of elements for morphing.
- The program prompts with the following message:

SELECT CONTROL NODE FOR MORPHING

Select any interior node from the selected element group.

- After the proper node is selected, the program displays a dialog window as shown in Figure 6.4.6.

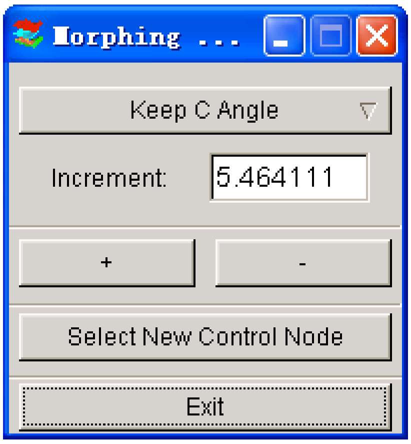


Figure 6.4.6 Morphing Dialog Window

The drop down button, KEEP C ANGLE, allows the user to select the type of constraint for morphing. The description of each constraint type is given in Section 6.2.1. Figure 6.2.14 illustrates the characteristics and results of each constraint type.

Select a constraint type and enter the morph increment in the dialog window. Click 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 6.4.7 shows a typical result (red) of element interior morphing.

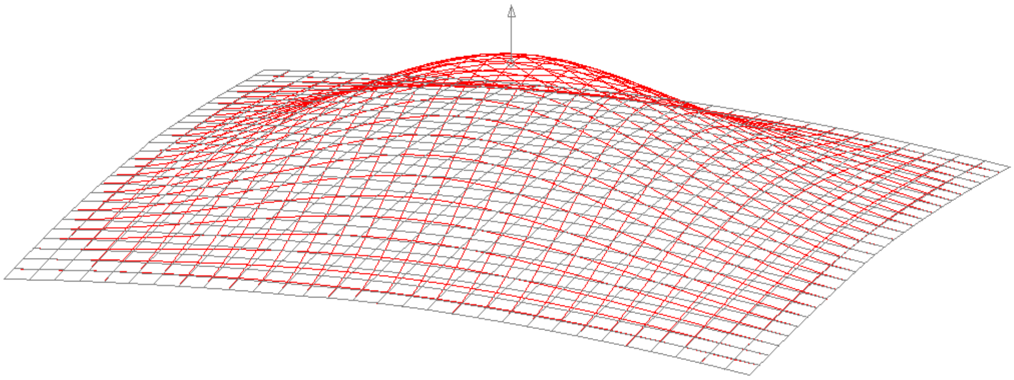


Figure 6.4.7 Element Interior Morphing

- Clicking **EXIT** updates the morphing elements and redisplays the SELECT ELEMENTS window.
- Select **ELEMENTS** to repeat the above step or **CANCEL** to exit the function.

2. EDGE MORPHING

This function uses a node on the boundary of the element region as the control node. The operation is the same as the INTERIOR MORPHING function.

3. CORNER MORPHING

This function uses a corner node of the element region as the control node. The operation is the same as INTERIOR MORPHING.

4. EDGE QUADRATIC

This function morphs 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 program uses the entire edge as the control line to morph the elements in the region.

The rest of the operations are the same as INTERIOR MORPHING.

6.4.4 DRAWBAR

The drawbar menu provides functions to generate drawbars in the Die. There are three types of drawbars available.

1. Draw bar ends with a spherical shape (tear drop head). Figure 6.4.8 shows the control parameters in the dialog window.

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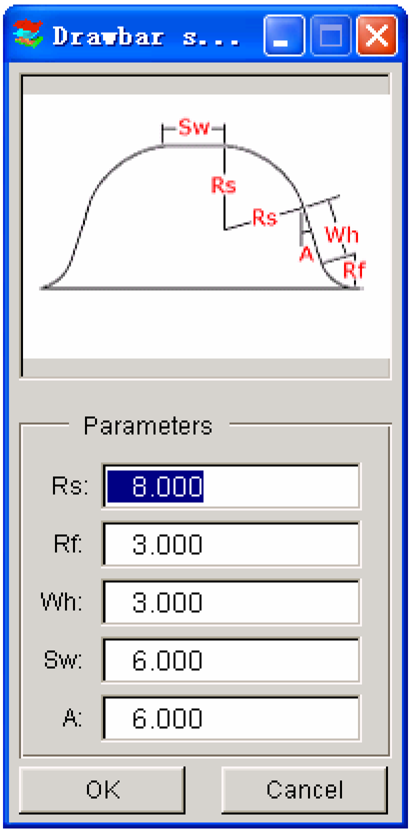


Figure 6.4.8 Parameters for Tear Drop Head

2. Draw bar ends with a conical shape (tear drop tail). Figure 6.4.9 shows the parameters in the dialog window.

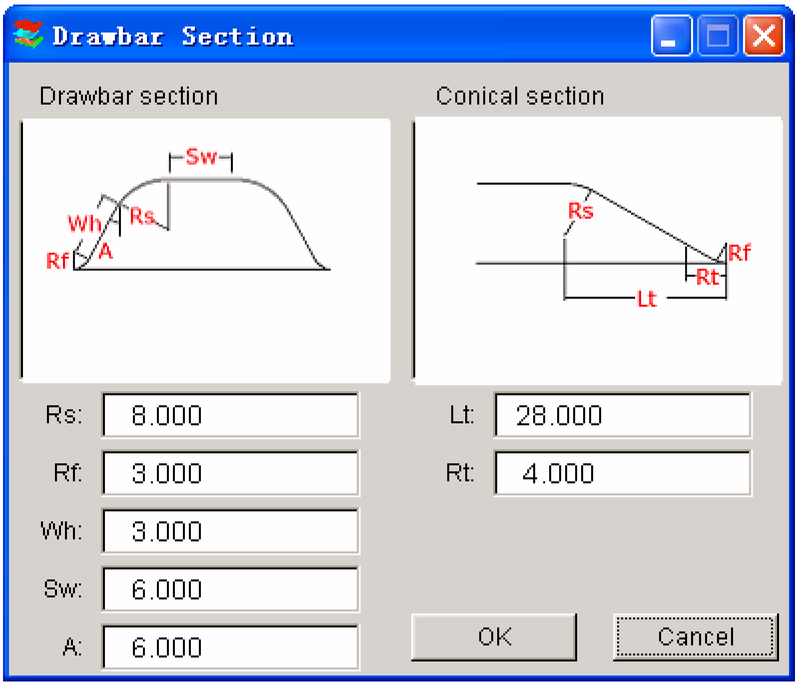


Figure 6.4.9 Parameters for Tear Drop Tail

3. Tear Drop (draw bar without main body). Figure 6.4.10 shows the parameters in the dialog window.

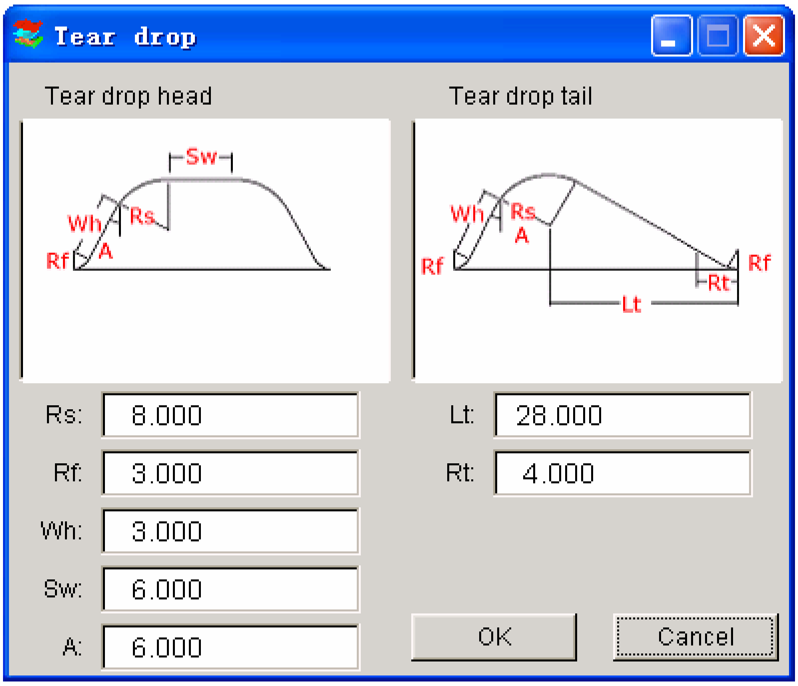


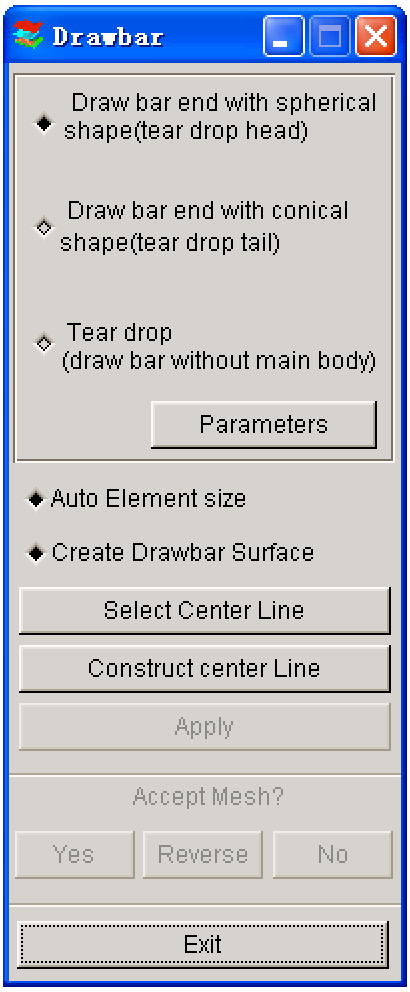
Figure 6.4.10 Parameters for Tear Drop

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

Create a drawbar using the following steps:

- Click the **DRAWBAR** function, Figure 6.4, and the program will display a dialog window as shown in Figure 6.4.11a. If the user wishes to define the element size of the drawbar, un-check the **AUTO ELEMENT SIZE** option. The dialog window will be changed to Figure 6.4.11b to allow the user to enter the desirable element size.



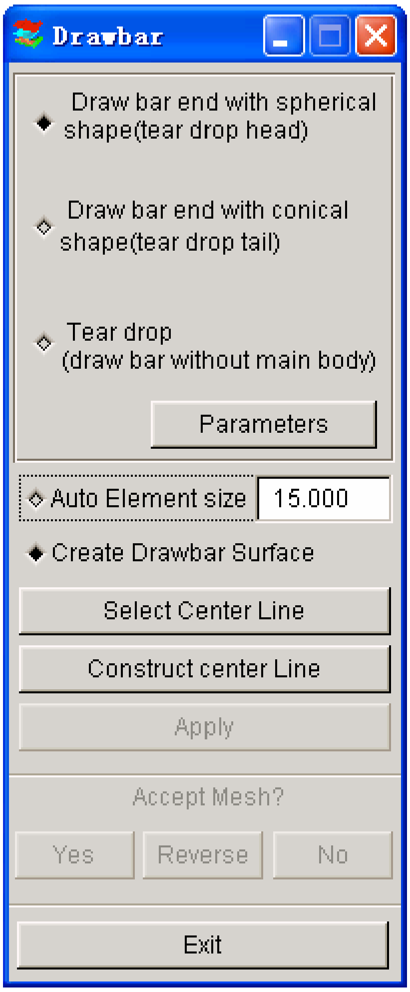


Figure 6.4.11a Drawbar with Auto Element Size Figure 6.4.11b Drawbar With User Defined Element Size

- Select one type of drawbar. Draw bar ends with spherical shape (tear drop head) is used as the default selection.

- Click the **PARAMETERS** button, and the program will display the corresponding dialog window as shown in Figure 6.4.8, Figure 6.4.9, or Figure 6.4.10.

- Select the element size for the drawbar, either by **AUTO ELEMENT SIZE** or by user-defined element size. The default option is AUTO ELEMENT SIZE where the element size of the draw bar is calculated based on the connecting mesh of the tool. The user may define any other value suitable for the specific requirements.

- **CREATE DRAWBAR SURFACE** option will generate the surfaces when the drawbar mesh is created. This option is selected as default. The user may disable the option if the drawbar surface is not required.

- Define the center line of the drawbar. The center line is the center of the drawbar's main body, including both ends.

- If there is a center line in the current database, click the **SELECT CENTER LINE** button to select the center line, or click the **CONSTRUCT CENTER LINE** button to create a line by selecting nodes on the mesh.

- After the drawbar parameters and center line are defined, click **APPLY** button to generate the drawbar. The program will activate the mesh acceptance buttons and prompt the user to accept the drawbar mesh. Click **YES** will keep the drawbar mesh or **NO** to reject the mesh. **REVERSE** buttons is used to put the drawbar on the opposite side of the tool. If the CREATE DRAWBAR SURFACE options is selected, the drawbar surface will be generated with the mesh.

  If the draw bar type is Tear Drop, the program will display a dialog window as shown in Figure 6.4.12. Click the **SELECT TWO NODES** button to select two nodes to define the direction of the center line. 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.

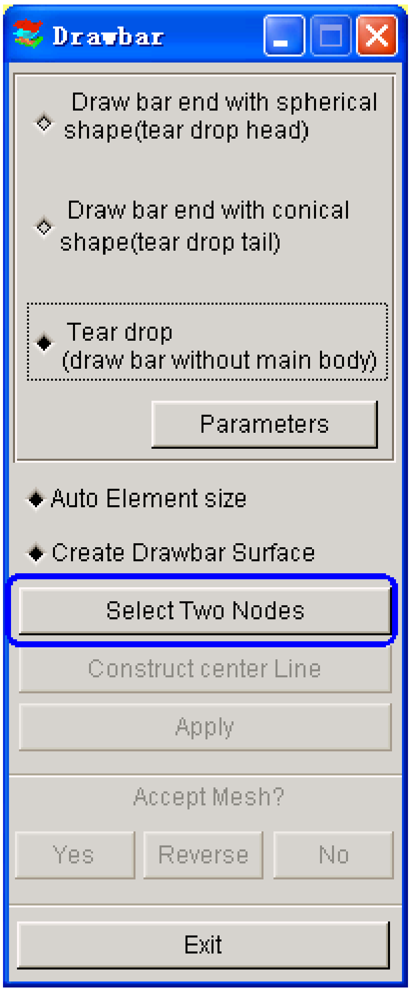


Figure 6.4.12 Tear Drop

Figure 6.4.13 shows the details of three types of drawbars.

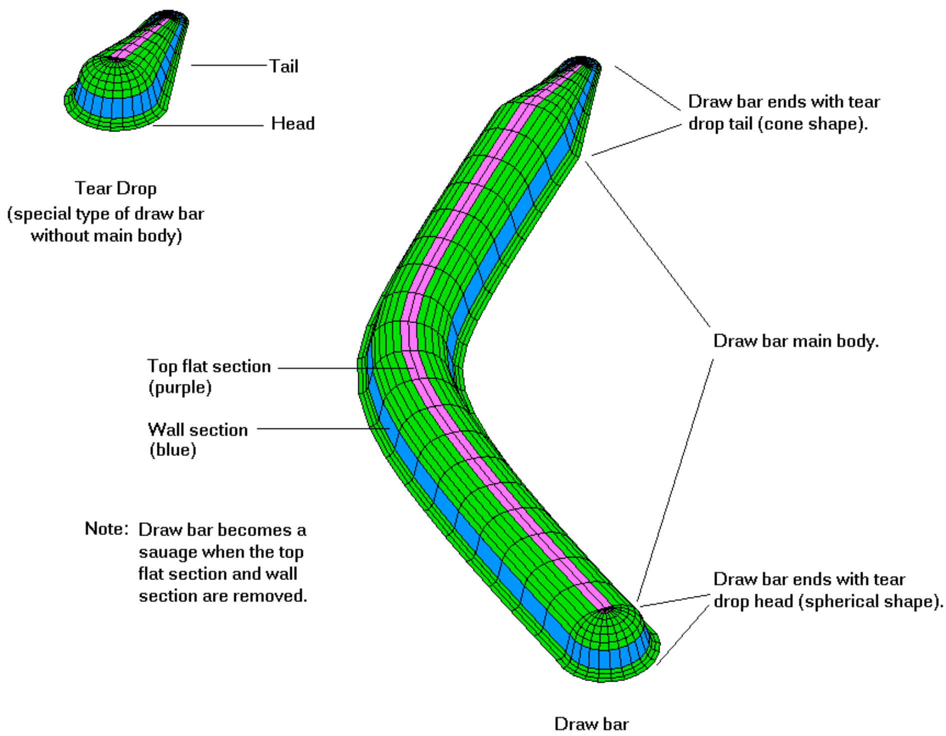


Figure 6.4.13 Three Types of Draw bars

6.4.5 DRAWBEAD TRIM

The DRAWBEAD TRIM function is used to trim the mesh along the outline of the drawbead. The operation of this function follows.

- The program displays the SELECT LINE dialog window to prompt the user to select the drawbead outline. The drawbead outline can be defined by one or more lines.
- Select a line as shown in Figure 6.4.14.
- Select **OK** to accept the selected line. The program will show the trimmed result and display a DYNAFORM QUESTION window as shown in Figure 6.4.16 to prompt the user to accept the trimmed result.
- Click **YES** to accept the mesh.

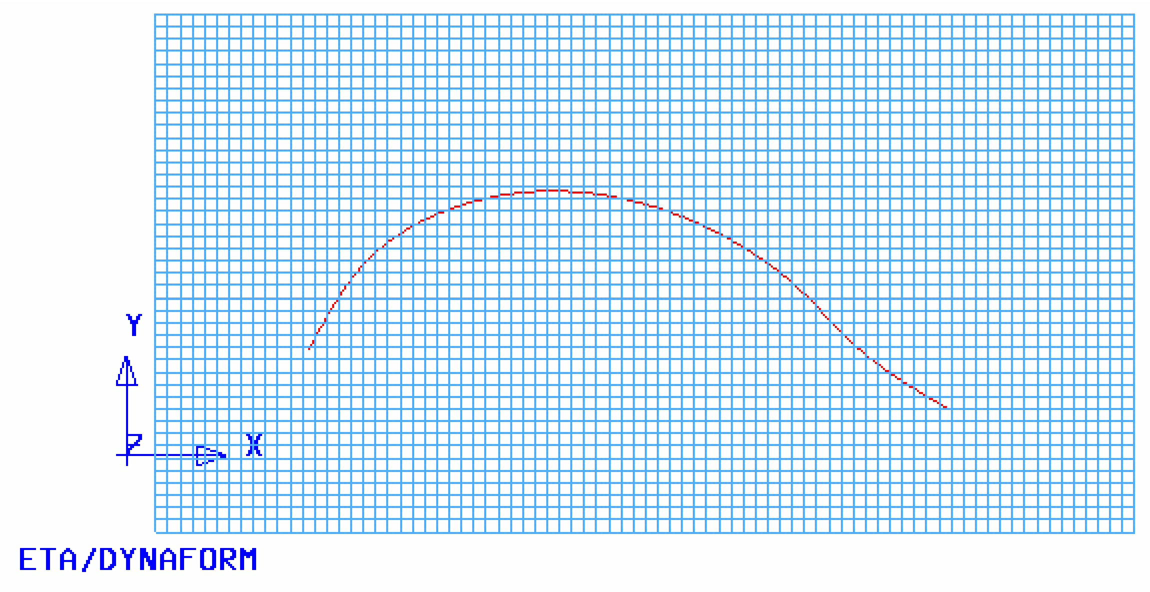


Figure 6.4.14 Select drawbead Outline

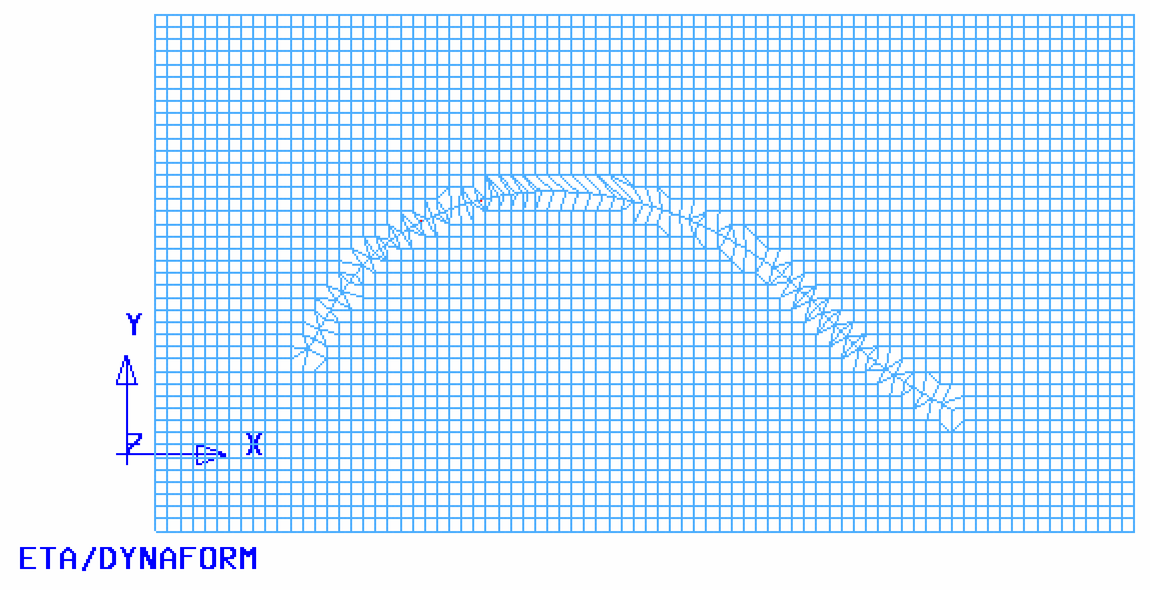


Figure 6.4.15 Trim Result

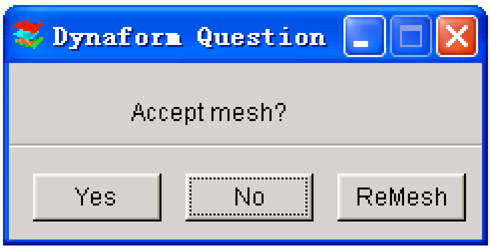


Figure 6.4.16 DYNAFORM QUESTION window

6.4.6 LASER TRIM

This function is used to trim the part from a line normal to the part. The selected line is the path of the laser beam cutting the part. The operation of this function is similar to the DRAWBEAD TRIM function except that a closed curve is required to define the trim line.

6.4.7 BINDER TRIM

After the addendum is generated for the Die, the binder must be trimmed to connect 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 there is no addendum defined, the Die Boundary line will be used to trim the binder. Figure 6.4.17 shows the functions available in the dialog window for the BINDER TRIM function.

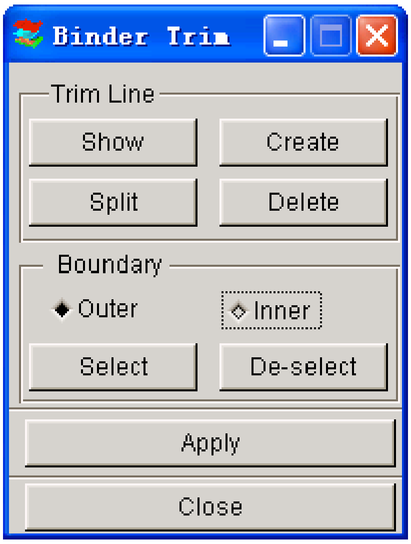


Figure 6.4.17 Binder Trim

- **EDIT LINE**

  These functions enable the user to edit the trim line. Refer to Section 6.1, LINE/POINT, for a detailed description of EDIT LINE functions.

- **BOUNDARY**

  There are two ways of trimming. The trim line can be used as an inner boundary or outer boundary. If **OUTER** is toggled **ON**, the portion outside of the trim line will be kept. If **INNER** is toggled **ON**, the portion within the trim line will be kept. 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 selecting the desired trim line, click **APPLY** to trim the binder.

6. 5 DIE DESIGN CHECK

This function allows the user to visually check the Die according to the minimum tip angle and minimum draw depth. Figure 6.5 shows the available check criteria in the DIE DESIGN CHECK function.

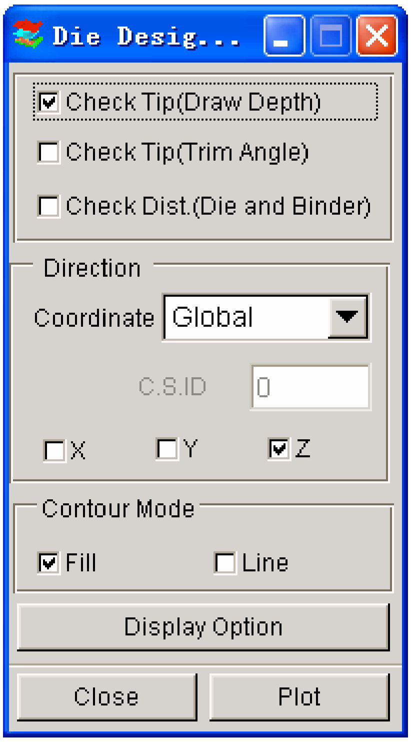


Figure 6.5 Die design check window

CHECK TIP (DRAW DEPTH)

This option is based on a flat blank. The function is the same as in Section 6.1.6, TIPPING.
Click the **PLOT** button, and eta/DYNAFORM will display the draw depth control of the Die.

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 the **PLOT** button, and eta/DYNAFORM will display the contour of the trim angle. Red color indicates the area needs cam trim and blue indicates direct trim.

CHECK DISTANCE (DIE AND BINDER)

Checks the draw depth relative to the Binder. The binder must be in the database when using this function. This check function will not work once the binder has been trimmed by the BINDER TRIM function.

Click the **PLOT** button, and eta/DYNAFORM will prompt the user to the binder. The program will display the contour of distance between the Die and Binder.
DIRECTION

This function enables the user to select a drawing direction, either the Global Coordinate System (GCS) or the Local Coordinate System (LCS).

DISPLAY OPTION

This function controls the display of the result. Refer to the eta/POST USER'S MANUAL for a detailed description of the display options.

CHAPTER 7

BLANK SIZE ENGINEERING - BSE MODULE

BLANK SIZE ENGINEERING (BSE) is an eta/DYNAFORM add-on module. The functions provided in the BSE submenu are designed to unfold a part and estimate a flat blank outline. In addition, BSE can be utilized to estimate a blank size and conduct a blank development. As shown in Figure 7.1, the BSE submenu consists of PREPARATION, MSTEP, and DEVELOPMENT.

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 window. The user must first import part geometry or part mesh using BSE/PREPARATION/IMPORT OR FILE/IMPORT. Refer to SECTION 3.6 for details on importing geometry files.

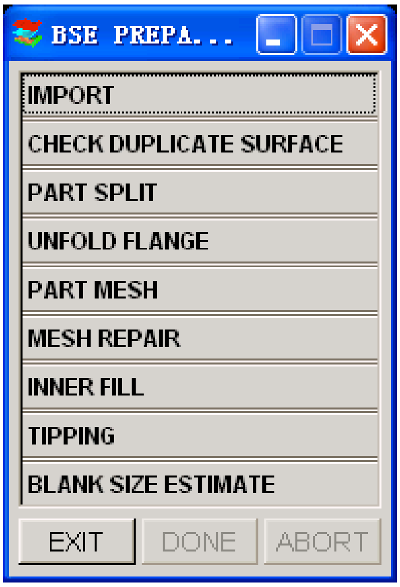


Figure 7.1.1 BSE Preparation dialog window

IMPORT—Enable the user to import the geometry model to BSE. Please refer to the Section 3.6 for more detailed information about IMPORT operation.

CHECK DUPLICATE SURFACE—Enable the user to check the duplicate surface before meshing. Please refer to the Section 5.2.22 for more detailed information about this function.

PART SPLIT—Enable the user to group surface and /or create middle surface if the user imports the three dimensional solid geometry model. The options in this function are very similar to the GROUP SURFACE AND MIDDLE SURFACE in preprocess. Please Refer to Section 5.2.23 and 5.2.24 for information about the GROUP SURFACE and MIDDLE SURFACE functions.

UNFOLD FLANGE—Enables the user to unfold the flanges on a product fully or by incremental angles on a product. Please refer to Section 6.1.1 for detailed information about unfold flange operation.

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

INNER FILL—Used to fill in any holes on the part mesh. Refer to Section 6.1.5.

Refer to Section 6.1.7 for information about TIPPING.

7.1.1 BSE (BLANK SIZE ESTIMATE)

As shown in Figure 7.1.2, the BLANK SIZE ESTIMATE dialog window enables the user to define material properties and thickness according to the prior calculation of a blank outline using BSE. The estimated blank outline will be stored in a new part named BO_LINE. An example of BLANK SIZE ESTIMATION is shown in Figure 7.1.3.

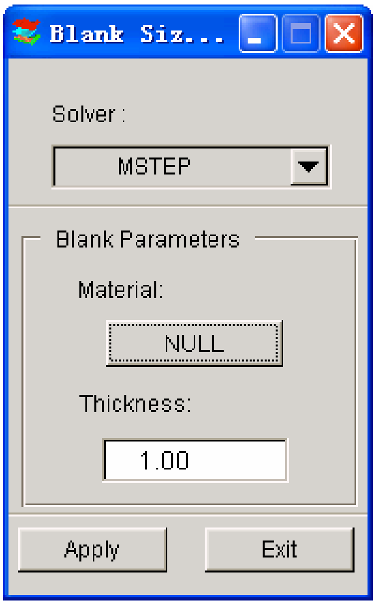


Figure 7.1.2 The Blank Size Estimate dialog window.

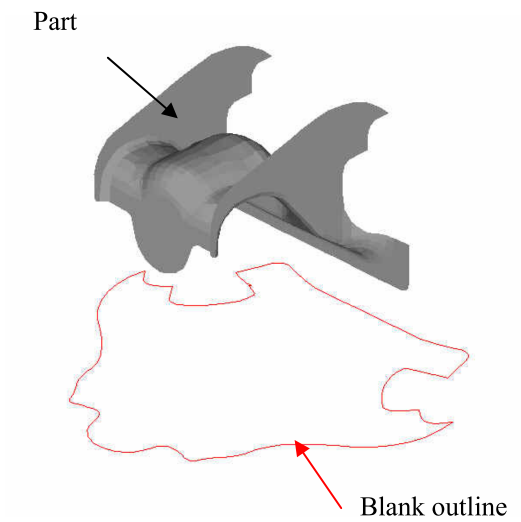


Figure 7.1.3 An example of Blank Size Estimation

SOLVER

There are two solvers for Blank Size Estimate as Figure 7.1.4 shown:

Figure 7.1.4 Solver for Blank Size Estimate

FASTBLANK is an old One-Step solver adopted by eta/DYNAFORM.

The MSTEP is a new Modified-One-Step solver developed by eta/DYNAFORM.

7.2 MSTEP

The MSTEP is a new Modified-One-Step solver developed by eta/DYNAFORM which is mainly used to quickly assess formability and estimate the blank outline in the early stage of automobile design cycle.

Figure 7.2.1 is the main interface of MSTEP. The method of defining the tools is similar to method in Quick-Setup menu to define tools. These tools marked with blue color are optional input tools. The sheet Tool is required input which is the final designed part geometry.

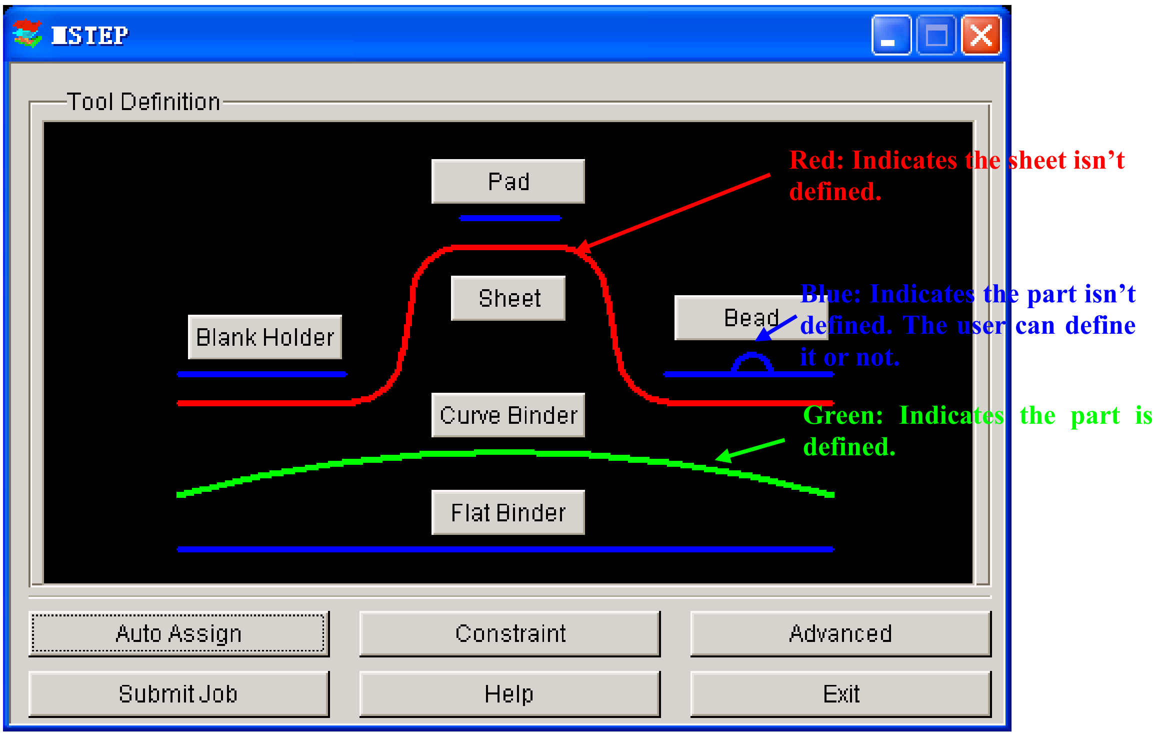


Figure 7.2.1 The Analysis dialog window for MSTEP

In the MSTEP, the program enables the user to unfold the Sheet on the Flat Binder or Curve Binder. The Flat Binder is the default position that the sheet will be unfolded on and the user need not define it. If the user doesn't assign any Binder, the program will unfold the outline and the mesh on the flat binder automatically. If defined the Curve Binder, the program will unfold the Sheet meshes to Curve Binder to the outline on the flat Binder.

AUTO ASSIGN: The program will automatically define the tools and the sheet.

CONSTRAINT: Define the constraints for the blank. Please refer to Preprocess / Boundary Condition / SPC set for **Constraint** set.

ADVANCED: Allows the user to change some default parameters that relate to MESEP settings. Figure 7.2.2 is the interface of MSTEP Setting.

ANALYSIS METHOD

Accurate -- This method enables the user to solve the forming process with Inverse Approach (One Step method).If this method toggled on, the user can define the Force and Control Parameter. The result of this method is more accurate but need more CPU time.

Fast -- This method enables the user to solve the forming process with elastic spread method. This method need less CPU time than accurate method.

FORCE

Binder hold-- Enable the user to define the binder hold force. The default value is 20000.

Pad support-- Enable the user to define the pad support force. The default value is 20000.

CONTROL PARAMETER

MAX. iteration steps-- Enable the user to define the maximal iteration steps for solution. The default value is 200.

Disp. convergence -- Enable the user to define the displacement convergence criteria. When the nodal displacement is less than this criteria, the iteration is convergent. The default value is 1.0e-003.

Friction -- Enable the user to define the fiction coefficient between the blank and the tool (binder or die). The default value is 0.125.

HELP: Provides useful tips for stamping simulation.

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

EXIT: Allows the user to dismiss the MSTEP interface..

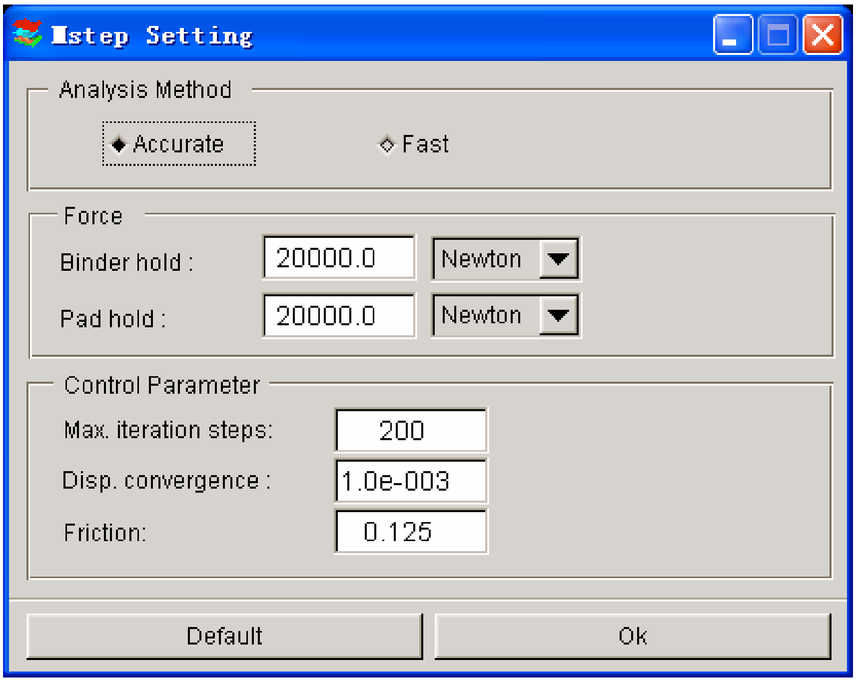


Figure 7.2.2 MSTEP Setting.

7.3 DEVELOPMENT

After obtaining the estimated blank outline, BLANK DEVELOPMENT enables the user to fine tune the blank outline for subsequent application in blank nesting, cost estimation, and forming simulation. The BLANK DEVELOPMENT dialog window provides several functions for convenient manipulation of the blank outline. The dialog window is shown in Figure 7.3.1.

Figure 7.3.1 The Blank Development dialog window.

7.3.1 BLANK GENERATION

This function enables the user to generate flat blank mesh using the blank outline. The elements generated using BLANK GENERATION will be QUAD dominant (as shown in Figure 7.3.2) which is required for an incremental stamping simulation using LS-DYNA.

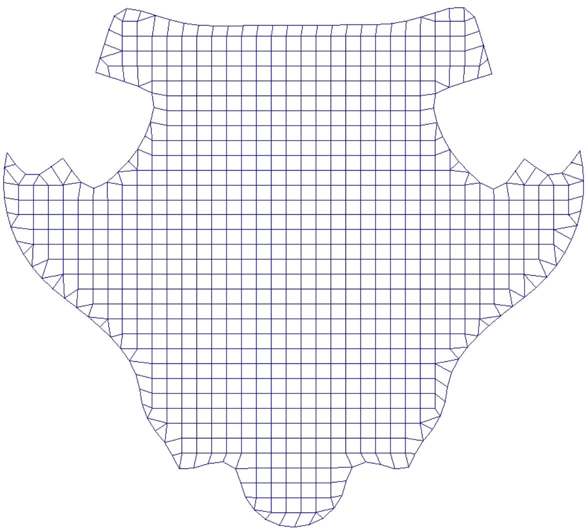


Figure 7.3.2 Blank mesh generated by Blank Generator

7.3.2 OUTER SMOOTH

The OUTER SMOOTH functions enable the user to smooth the blank boundary. Refer to Section 6.1.7 for more information about the OUTER SMOOTH functions. An example of smoothing blank boundary is shown in Figure 7.3.3.

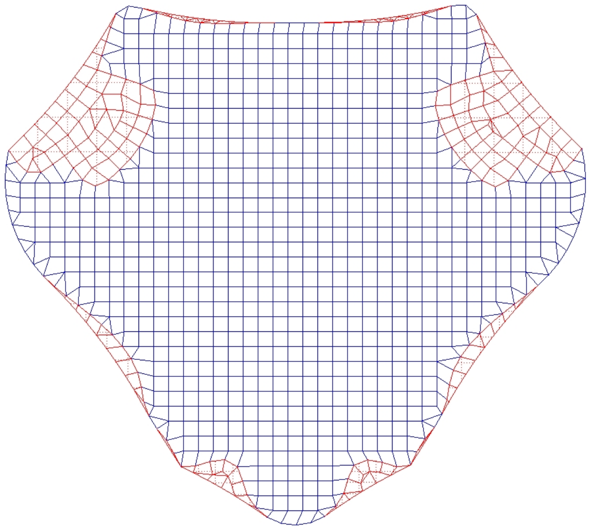


Figure 7.3.3 Blank mesh after smoothing

7.3.3 RECTANGULAR FITTING

This function enables the user to fit the estimated blank outline by using four straight lines. The dialog window is shown in Figure 7.3.4.

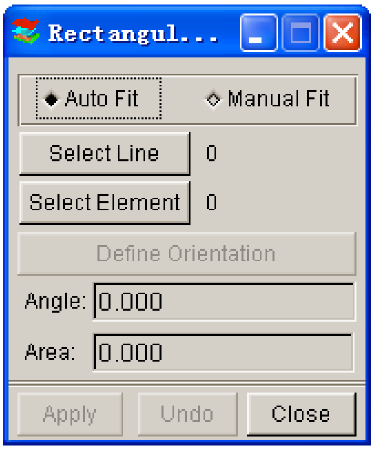
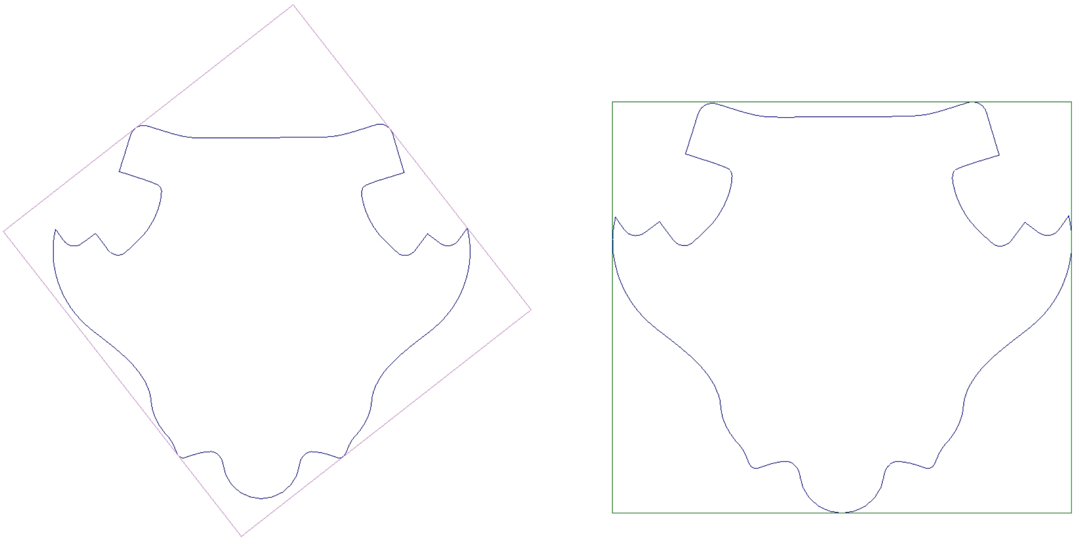


Figure 7.3.4 The Rectangular Fitting dialog window.

There are two types of RECTANGULAR FITTING: AUTO FIT and MANUAL FIT. If AUTO FIT is toggled on, the Angle (rotation) and Area (rectangular blank) will be automatically determined. An example of rectangular blank outline is shown in Figure 7.3.5.



a) Auto Fit b) Manual Fit

Figure 7.3.5 Rectangular blank outline created by Rectangular Fit

To create the rectangular blank outline:

- Select a type of RECTANGULAR FIT (AUTO or MANUAL)
- If MANUAL FIT is toggled, click on DEFINE ORIENTATION to define blank orientation.
  - Click and hold the left mouse button at the cross hair.
  - Move the mouse to rotate the rectangular outline.
  - Stop moving the mouse and release the left mouse button after obtaining the desired orientation.
- Select the estimated blank outline by clicking on the SELECT LINE button. Alternatively, the user can select the blank elements generated by BLANK GENERATION.
- Click APPLY.

7.3.4 EVALUATION REPORT

This function enables the user to output the result of BSE to a report in a HTML file.

7.3.5 EXPORT

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

7.3.6 BLANK NESTING

This function enables the user to nest the blank in the strip. optimized for material usage in a progressive strip. Visit this function by clicking the **BSE->Development->Blank Nesting.** Figure 7.3.6 shows the types of binder available in the BLANK NESTING menu.

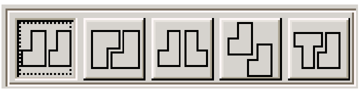


Figure 7.3.6 Blank Nesting Types

ONE-UP NESTING

This type enables user to generate the nesting result in one row.

TWO-PAIR NESTING

This type enables the user to generate the nesting result with two pair nesting blanks with their orientation opposite.

MIRROR NESTING

This type enables the user to generate the nesting result with mirror nesting blanks.

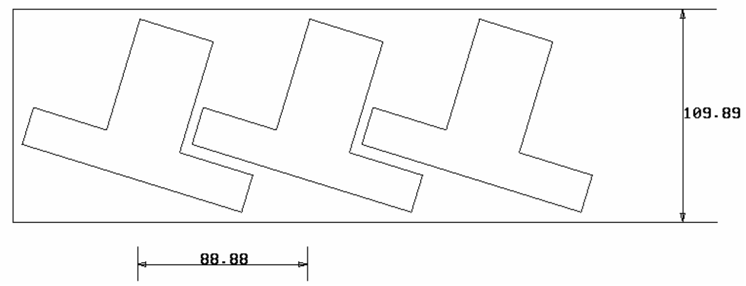
TWO-UP NESTING

This type enables the user to generate the nesting result with two up nesting blanks with their orientation is accordant.

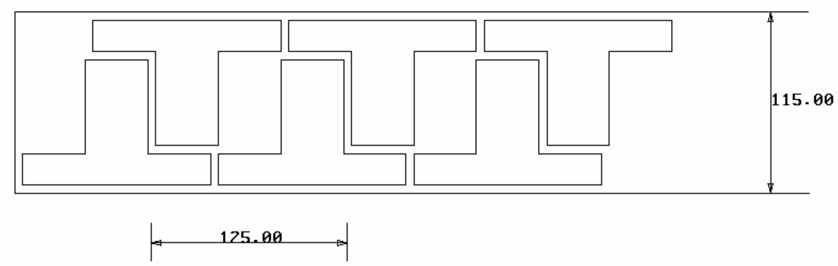
TWO DIFFERENT BLANKS NESTING

This type enables the user to generate the nesting result with two different blanks.

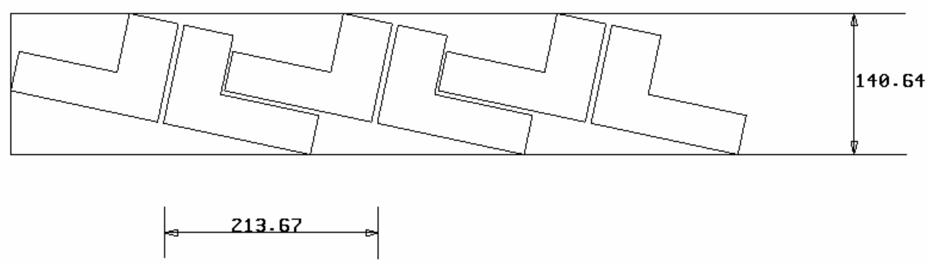
An example of blank nesting is shown in Figure 7.3.7. A detailed description of each function is given in the following section.



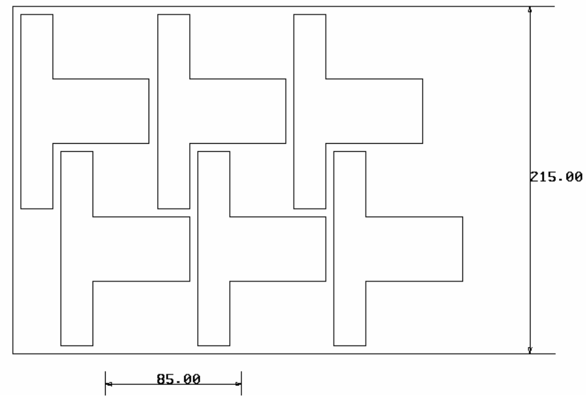
a) One-up Nesting



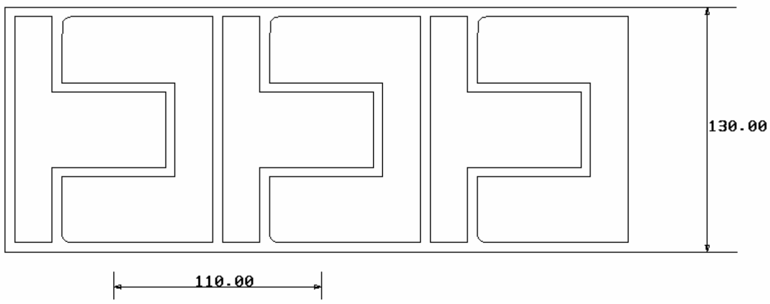
b) Two-pair Nesting



c) Mirror Nesting



d) Two-up Nesting



e) Two different blanks

Figure 7.3.7 Blank Nesting

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

7.3.6.1 ONE-UP NESTING

This function enables the user to generate the nesting result in one row. The dialog window is shown in Figure 7.3.8.

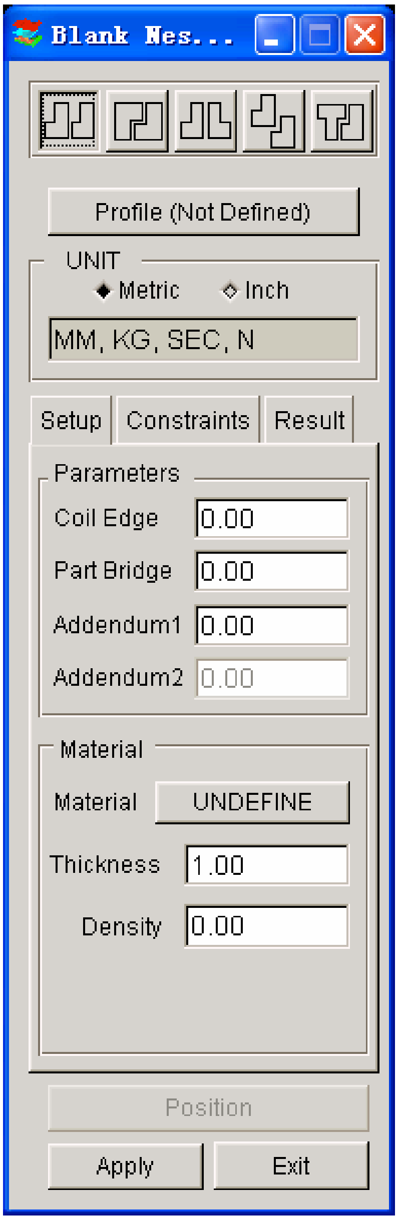


Figure 7.3.8 One-up Nesting

- **Profile(Not Defined)**

  This parameter is used to define the Blank Outline. Click the **Profile (Not Defined)** button in Figure 7.3.8, and the program will display the Select Line menu to prompt the user to select lines. If the profile is selected, the button name will be set as Profile.

UNIT(toggle on)

Metric
This option enables the user to export the Nesting result with metric unit.

Inch
This option enables the user to export the Nesting result with inch unit.

SETUP

In this Tab, parameters used to control the gaps of blank nesting such as Coil Edge, Part Bridge and Addendum1 are defined. Figure 7.3.9 describe these parameters.

- **Coil Edge**

  Coil edge allows user to define the gap between part and strip edge.

- **Part Edge**

  Part Edge allows user to define the gap between parts.

- **Addendum1**

  Addendum1 allows user to expand the part

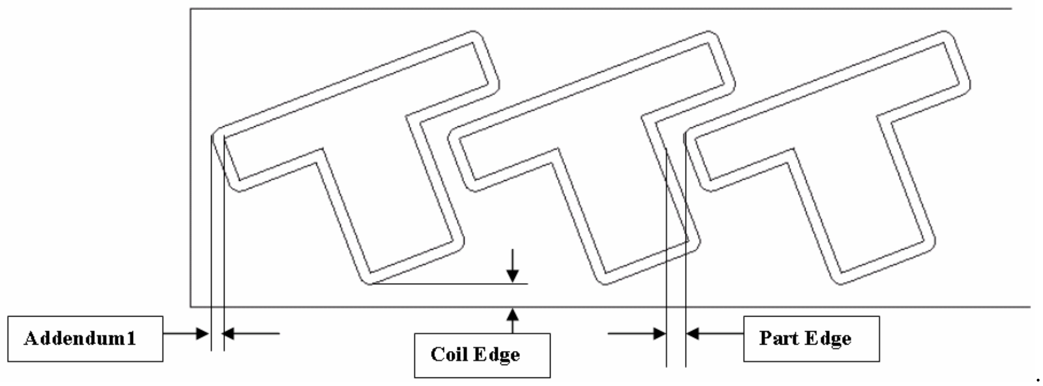


Figure 7.3.9 Coil Edge, Part Bridge and Addendum1 parameters

The Material group enables the user to input the Material parameters so that the program can give the probable cost of the strip. Those parameters include Materials, Thickness, Density and Unit Price.

CONSTRAINTS

This function enables the user to constrain the dimension of the strip and the angle of blank to the strip. See

Figure 7.3.10.

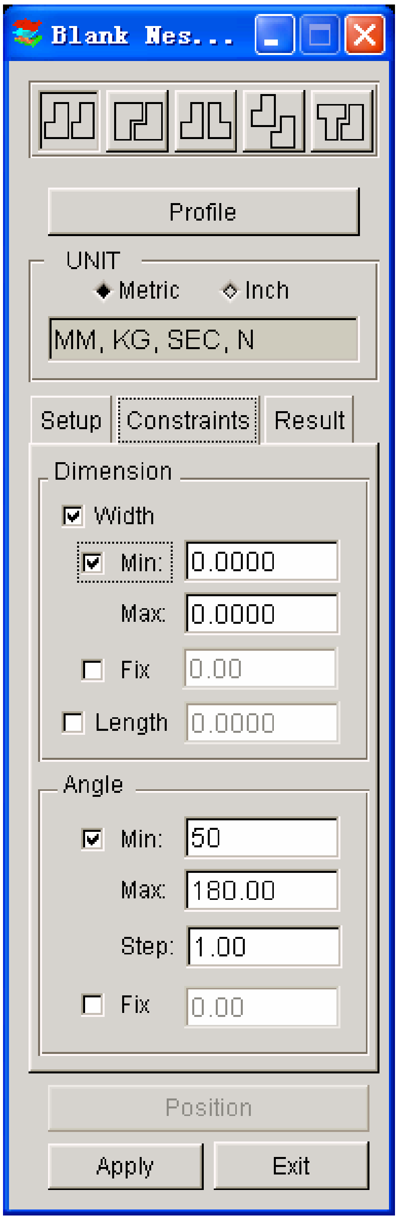


Figure 7.3.10 Constraints

DIMENSION OPTION

Width is used to constrain the width of the strip

- User inputs the range of the blank width with Min and Max width, program will calculate a series of results according to this float range of the width.
- Fix width used as the width constraint.

- **Length**

  The Fix Length can be the length constraint.

ANGLE OPTION

- User inputs the range of the blank angle with Min and Max value, program will calculate a series of results according to the float range of the angle.
- Fix angle used as the angle constraint.

APPLY

Click the **Apply** button in Figure 7.3.10, and the program will run the blank nesting and show result as shown in Figure 7.3.11.

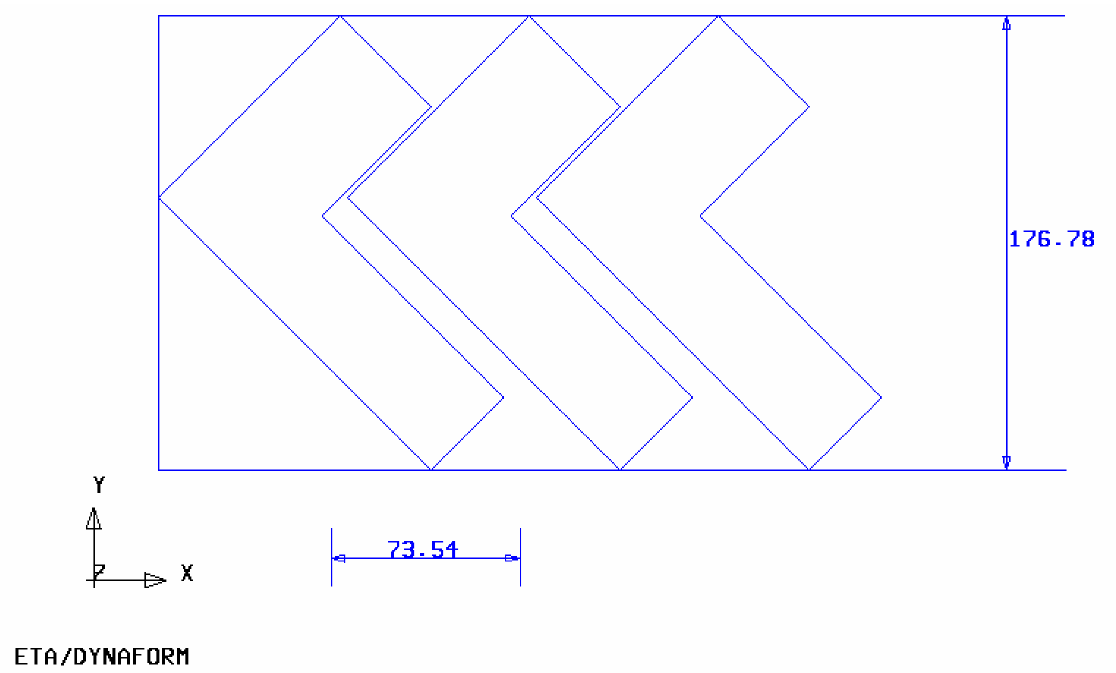


Figure 7.3.11a The result of One-up Nesting

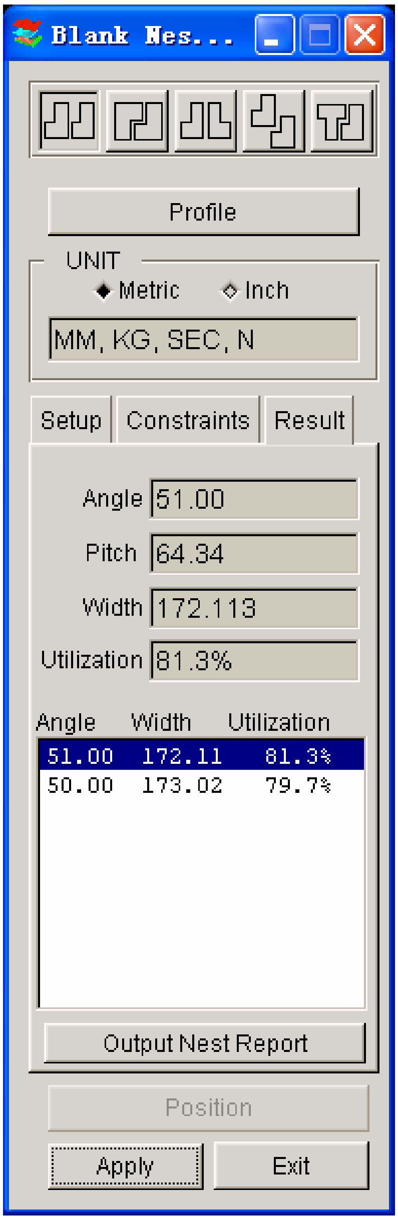


Figure 7.3.11b Result of Blank Nesting

The program will give the user a series of available blank nesting results that meet the given constrain. Users can click the mouse on the result list and the selected scheme will highlight and the corresponding constraints are displayed in the text box above the window including

Angle, Pitch, Width, Utilization. As shown in Figure 7.3.11b there are two schemes for users to select. Users can reduce the constraint range according the first solver results to get more detailed nesting scheme.

- **Output Nest Report**

  This function enable the user to output the result as a *.htm file.

7.3.6.2 TWO-PAIR NESTING

This function enables the user to generate the nesting result with two pair nesting blanks, maintaining their opposed orientation. The setting of Setup and Constraint in this function is similar with that in the One-up Nesting, about relevant please refer to section 7.3.5.1.

In this function the Position option is activated after selecting the profile. This option enables the user to adjust the position between the two blanks. Program will move and/or rotate the profile marked with 2 around with profile 1 to adjust its position. Click this button and the program will open the Auto Position window as Figure 7.3.12 shown.

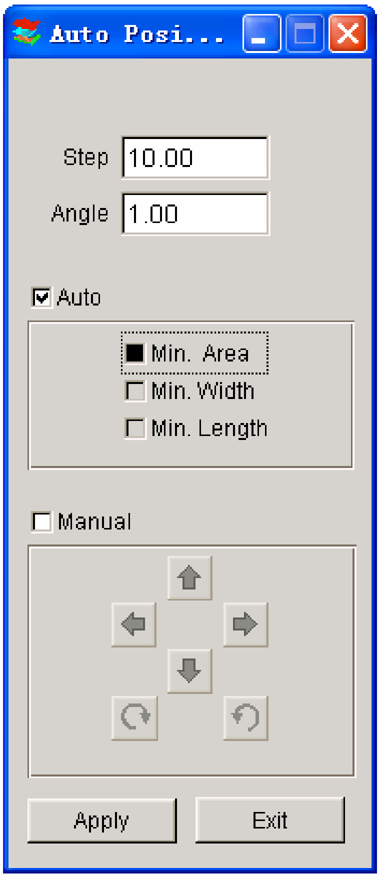


Figure 7.3.12 Auto Position

Transformation incremental step

- **Step**

  This is the increment value for transformation. If the user adjusts the position of the blank through Manual method for transformation, the blank will move an increment value for each user click of the transformation button, If through Auto method, program will calculate a series of results according the step.

- **Angle**

  This is the increment value for rotation. If the user adjust the position of the blank through Manual method for rotation, the blank will rotate an increment angle for each user click of the rotation button one time, if through Auto method, program will calculate a series of results according the increment.

Auto method

This function enables the user to position the nesting blank automatically through the selected optimized target.

- Min. Area

  The program will position the nesting blank with the minimal area of the strip.

- Min. Width

  The program will position the nesting blank with the minimal width of the strip.

- Min. Length

  The program will position the nesting blank with the minimal length of the strip.

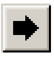
Manual method

This function enables the user to position the nesting blank manually through transforming and/or rotating the nesting blank.

- Transformation

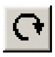
   Move upwards the second profile with an increment step (the Value in Step box).

   Move downwards the second profile with an increment step.

   Move rightwards the second profile with an increment step.

   Move leftwards the second profile with an increment step.

- Rotation

   Rotate clockwise the second profile with an increment angle (the Value in Angle box).

   Rotate anti-clockwise the second profile with an increment angle.

Click Apply button to get the position result.

Click Exit button to exit this window.

7.3.6.3 MIRROR NESTING

This function enables the user to generate the nesting result with mirror the original blank. The mirrored blank is eudipleural with the original blank. The setting of Setup and Constraint in this function is similar with that in the Two-Pair Nesting. Please refer to section 7.3.6.1. and 7.3.6.2 for all relevant parameters.

7.3.6.4 TWO-UP NESTING

This function enables the user to generate the nesting result with two pair nesting blanks with two up nesting blanks with their orientation is accordant. The setting of Setup and Constraint in this function is similar with that in the Two-Pair Nesting. Please refer to section 7.3.6.1. and 7.3.6.2 for all relevant parameters.

7.3.6.5 TWO-DIFFERENT NESTING

This function enables the user to generate the nesting result with two pair nesting blanks with two different blanks. The setting of Setup and Constraint in this function is similar with that in the Two-Pair Nesting. Please refer to section 7.3.6.1. and 7.3.6.2 for all relevant parameters.

CHAPTER 8

QUICK SETUP

The functions provided in the QuickSetup (QS) user graphic interface (GUI) help the user to quickly setup a forming simulation. The QS menu consists of several standard-forming operations as shown in Figure 8.1.

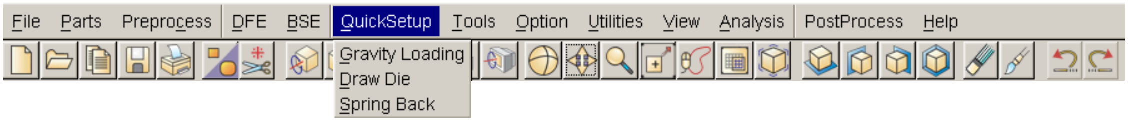


Figure 8.1 The Quick Setup menu.

The standard-forming processes supported by QS are the following:

Gravity Loading

Draw Die – Crash Form
– Inverted Draw (Single Action)
– Toggle Draw (Double Action)
– Four Piece Draw (Double Action Stretch Draw)

Springback

The QS GUI is designed and embedded in eta/DYNAFORM Version 5.1 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 geometry offset. Compared to a simulation solution using geometry offset, it provides a quicker solution for a stamping simulation without significantly compromising accuracy.

The following table outlines the major differences between the Quick Setup and the traditional DYNAFORM setup procedure.

Quick Setup	**Traditional Setup**
Automated interface limits flexibility	Manual interface can duplicate any tooling configuration: pads, multiple tools, etc.
Reduces modeling setup time	Requires more setup time
Automated travel curves	Manual definition of travel curves
Contact Offset	Geometrical Offset

Most of the QS GUI contains following options:

- **Auto assign** will assign tools that have the 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.

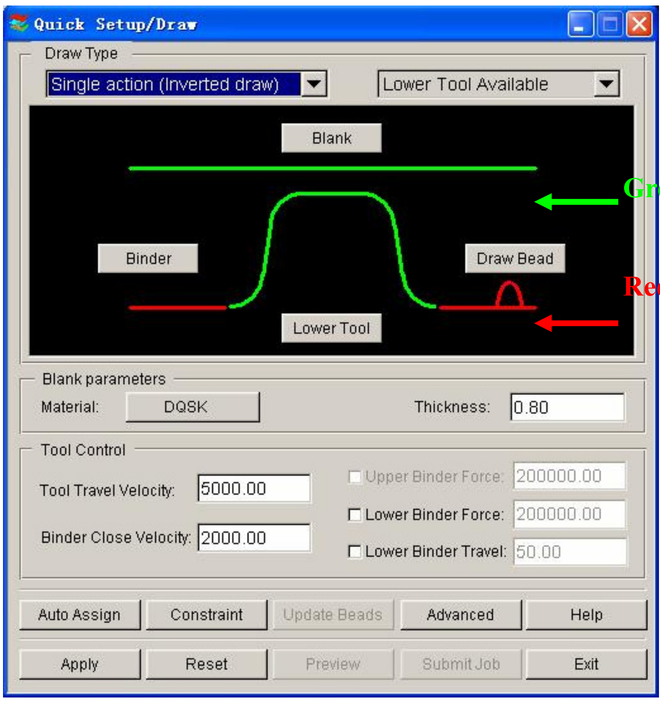
- **Reset** allows the user to recover the stage before creating the mating tool.

- **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 dismiss the QS interface.

A typical QS GUI is shown in Figure 8.2.



Green indicates that the tool is defined.

Red indicates that the tool or blank is not defined.

Figure 8.2 A typical Quick Setup user graphic interface.

8.1 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 8.1.1.

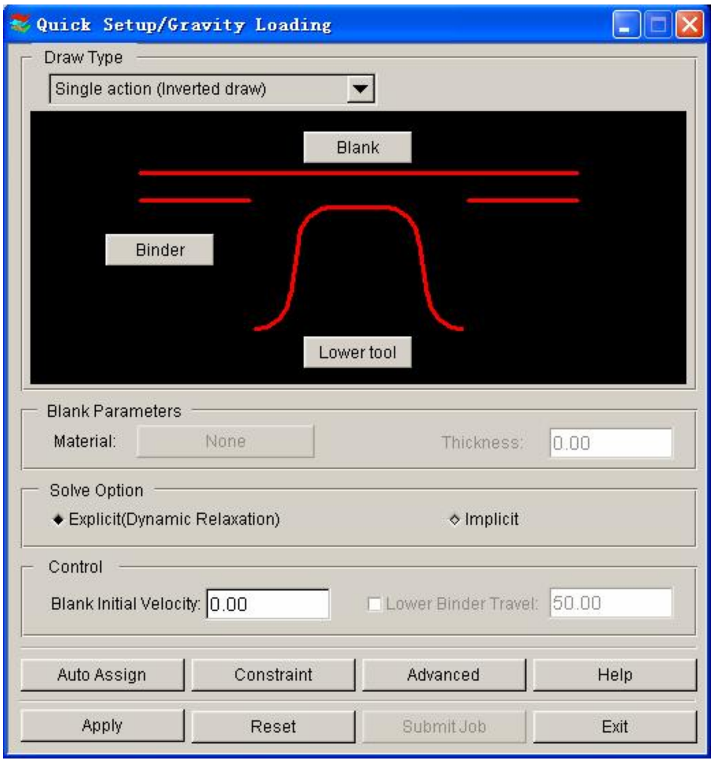


Figure 8.1.1 A typical Gravity Loading GUI for Single Action draw operation.

Draw Type

- Enables the user to select one of the four standard draw operations provided in QS GUI

Tool definition GUI

- Defines the tool and blank for a gravity loading simulation

Blank Parameters

- Defines blank material and properties

Solve Option

- The user can 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. The Explicit method is recommended for a gravity loading simulation because, although the Implicit method will yield an accurate solution, it is difficult to achieve converged non-linear equilibrium iterations when using this method.

Control

- Defines the initial velocity of blank

8.1.1 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 8.1.1), the tool definition GUI consists of BLANK, BINDER and LOWER TOOL.

By clicking on either the BLANK, BINDER or LOWER TOOL button, the dialog window as shown in Figure 8.1.2 will prompt the user to select the method of defining blank or tooling.

(a) (b)

Figure 8.1.2: The Tool Definition dialog window (a) Define Blank (b) Define Tool

If the blank mesh is available in a saved Nastran file, the user can click IMPORT MESH 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 DAT. The BLANK MESH function (as shown in Figure 8.1.2a) is then used to mesh the blank geometry. This function is similar to BLANK GENERATOR 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 8.1.2b) meshes the tool 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 8.1.2a) allows the user to select the proper parts from the SELECT PART dialog window (Figure 8.1.3) and assign them as Blank, Lower Tool and Binder in QS GUI. By clicking on SELECT PART, the DEFINE BLANK dialog window will appear, 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 window (Figure 8.1.2b) will display the DEFINE TOOL dialog window as shown in Figure 8.1.4.

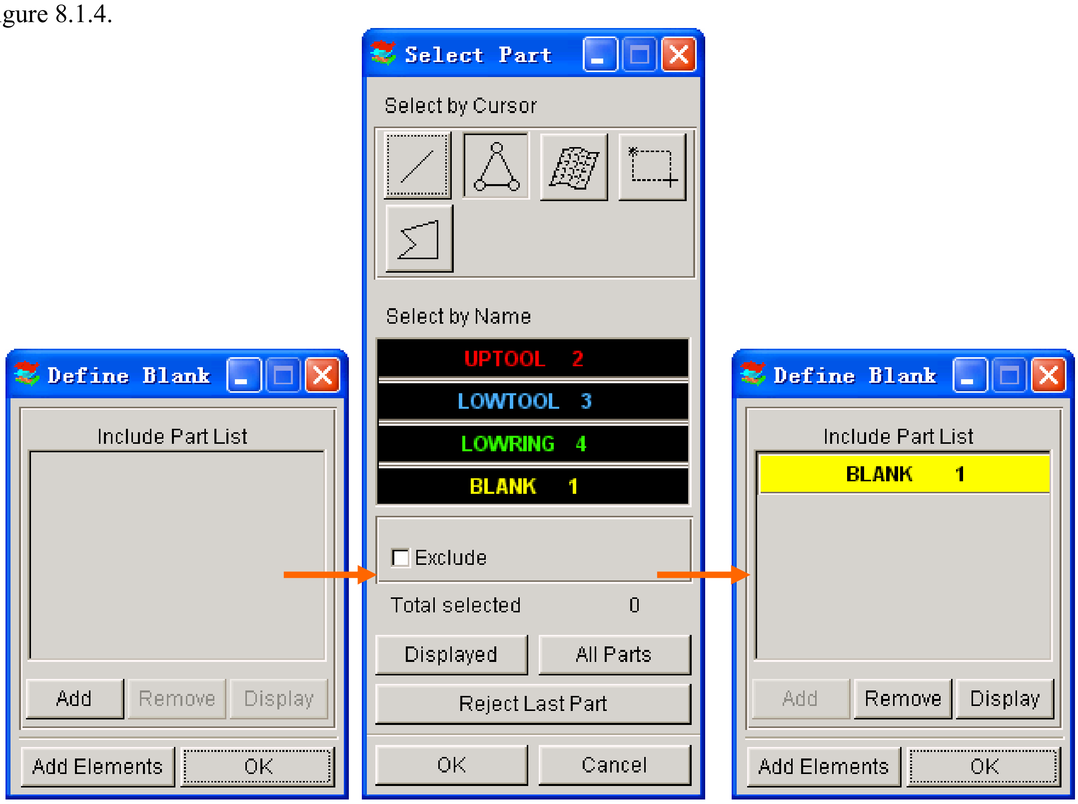


Figure 8.1.3 A schematic illustration showing the SELECT PART operation to define the Blank in QS.

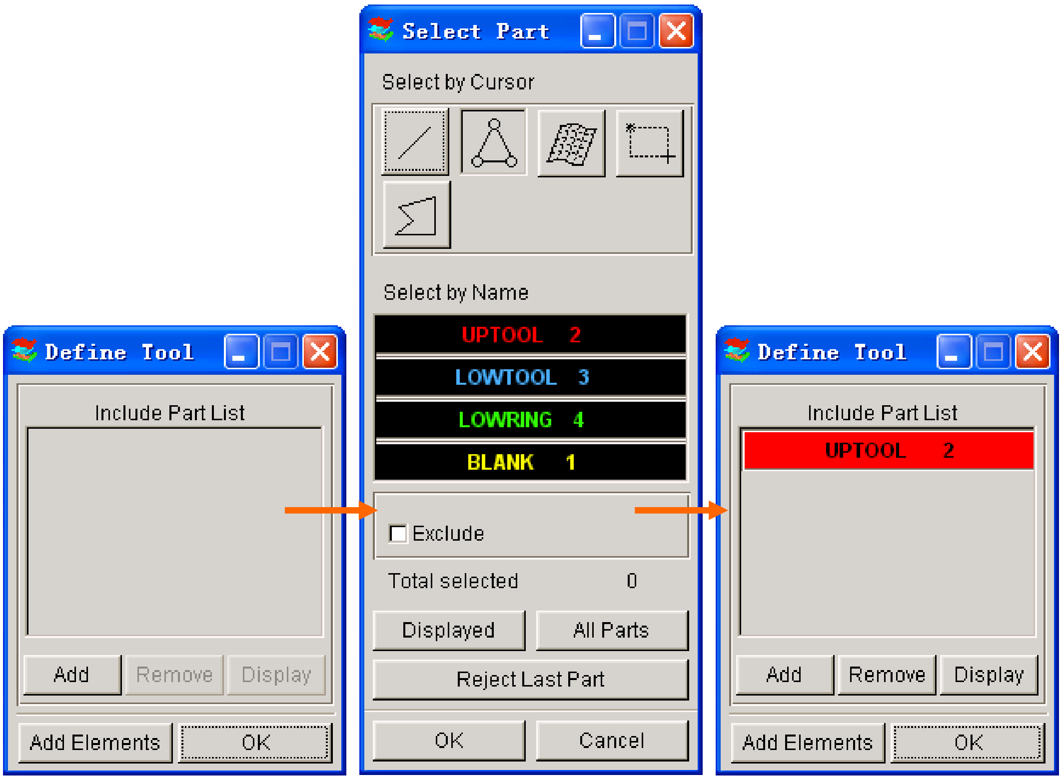


Figure 8.1.4 A schematic illustration showing the SELECT PART operation to define the Lower Tool in QS.

8.1.2 BLANK PARAMETERS

The user can define the blank material by selecting the **NONE** button as shown in Figure 8.1.1. It will display the MATERIAL dialog window as shown in Figure 8.1.5. For detailed information about defining the blank material, refer to Section 9.8. The **NONE** button is disabled if no blank is defined in the tool GUI.

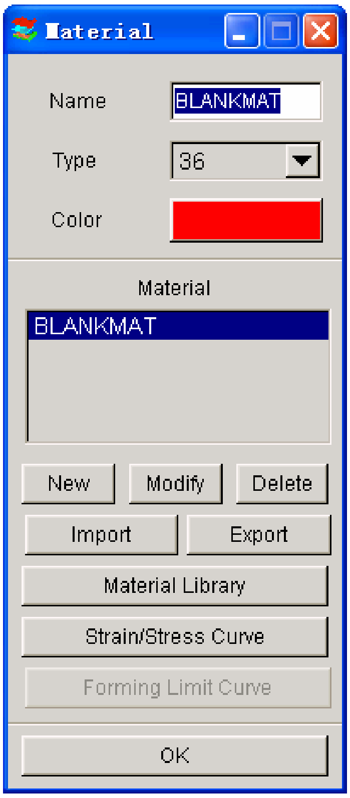


Figure 8.1.5 The Material dialog window.

8.1.3 QUICKSETUP/GRAVITY LOADING PROCEDURE

Follow the procedures listed below to set up the Single Action gravity loading simulation.

1. Select QUICKSETUP/GRAVITY LOADING.
2. Select DRAW TYPE.
3. Click on the BLANK/BINDER/LOWER TOOL button from 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 blank material. If blank material is not defined, NONE will be shown on the button. Click on the NONE button and proceed to select blank material from the embedded material library or user defined material.
5. Key in blank thickness. The default blank thickness is set as 1.00 (mm).
6. Select the solver options.
7. Click on APPLY and eta/DYNAFORM will proceed with auto-positioning between the tooling and blank.
8. Click on SUBMIT JOB to submit the analysis. Refer to Section 13.1.

8.2 DRAW

The Quick Setup/Draw GUI leads the user through the setup procedure for drawing simulation. 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 8.2.1.

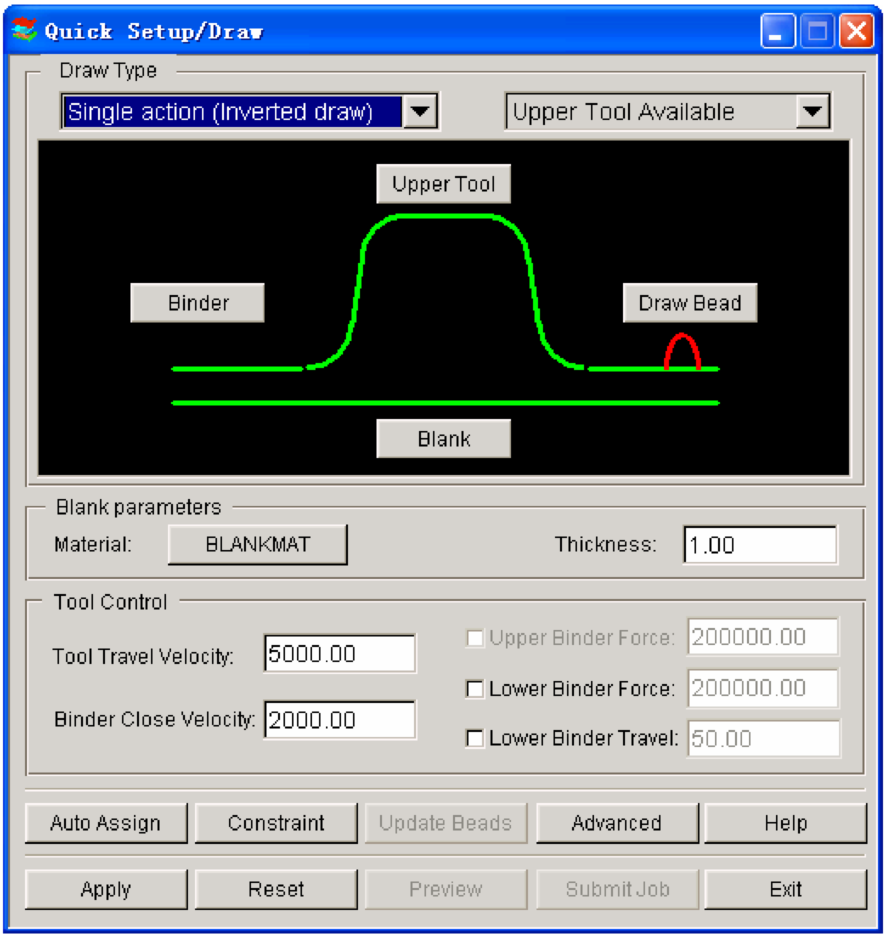


Figure 8.2.1 A typical Draw GUI for Single Action draw operation.

Draw Type

- Enables the user to select one of the four standard draw operations provided in QS GUI. In addition, the user can select either lower or upper tool in the setup depending on the type of tool surfaces (top or bottom).

Tool definition GUI

- Defines the tool, blank and drawbead for draw simulation.

Blank Parameters
- Defines blank material and properties. Refer to Section 8.1.2.

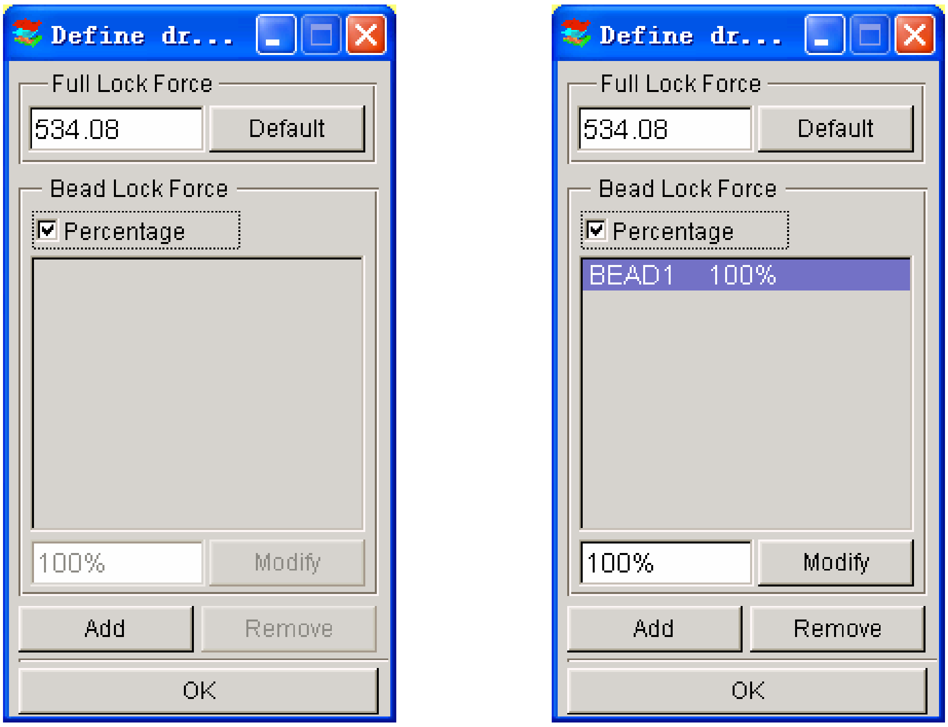
Tool Control
- Enables the user to set tool motion.

Advanced
- Allows the user to edit some of the default QS settings.

8.2.1 TOOL DEFINITION GUI

The tool definition GUI allows the user to assign the tooling and blank mesh required for setting up a particular draw simulation. Refer to Section 8.1.1.

In some draw simulations, drawbead utilization is required to provide better metal flow and prevent excessive wrinkling conditions. The QS/Draw enables the user to model drawbeads from the tool GUI as shown in Figure 8.2.1.



(a) (b)

Figure 8.2.2 The Define Drawbead dialog window.

After clicking the DRAWBEAD button (Figure 8.2.1), the user can define drawbeads from the DEFINE DRAWBEAD dialog window (Figure 8.2.2a). The user can click on ADD to select drawbead line data. If no drawbead line data is available in the database, the user can generate drawbead lines by using the POINT/NODE option.

The default full lock force is calculated based on material thickness and tensile strength. The user must define the blank material and thickness accordingly prior to generating drawbeads. As shown in Figure 8.2.2n, the user can modify the percentage of drawbead lock force after the drawbeads are defined. In addition, the user can simply add or remove drawbeads.

8.2.2 TOOL CONTROL

As shown in Figure 8.2.3, the tool control interface is utilized to set primary travel velocity and binder force. The binder force control is activated by selecting the Binder Force check box.

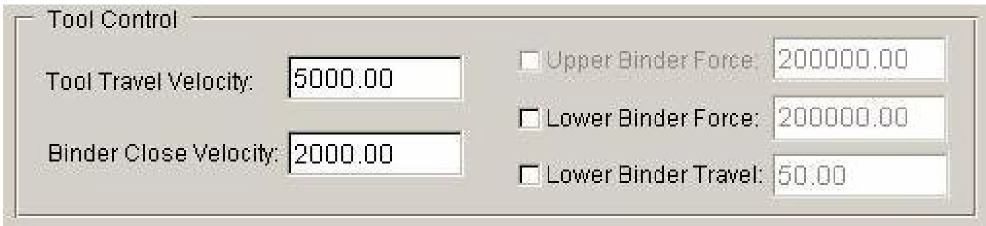


Figure 8.2.3 The Tool Control interface.

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

Crash Form	Upper Tool with defined tool velocity
Inverted Draw (Single Action)	Upper Tool is control by velocity Binder is controlled by either velocity or force
Toggle Draw (Double Action)	Upper Tool is controlled by velocity Binder is controlled by velocity or force
Four Piece Draw	Upper Tool is controlled by velocity Upper Binder is controlled by velocity or force Lower Binder is controlled by velocity or force

If default unit is selected, the unit of velocity and force are in **mm** and **Newton**, respectively.

Note:

1. In the QS interface, all force controls are implemented after the binder is closed using velocity control. That is, the velocity control is switched to force after binder closing.

2. In the QS interface, the force applied to the lower binder (in Four-piece draw) is controlled by velocity or force with the 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 binder with the force control:

a. Provide punch travel velocity (default is 5000mm).
b. Provide binder velocity (default is 2000 mm).
c. Provide binder force (default is 200,000Newton).

8.2.3 ADVANCED

This feature allows the user to edit the default Quick Setup setting, including the gap, load curve, output frames and drawbead control. The dialog window is shown in Figure 8.2.4.

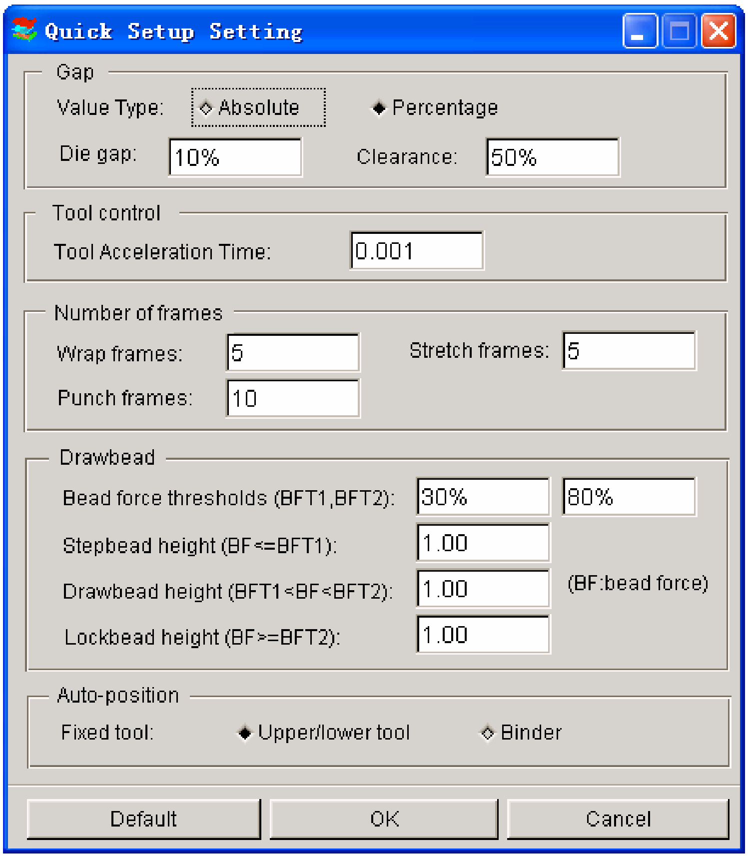


Figure 8.2.4 Quick Setup Setting dialog window.

The gap control allows the user to set the tooling gap and clearance between tooling and blank. The user can select either 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 will be 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 can 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 force threshold and drawbead height. As shown in Figure 8.2.4, the default drawbead threshold for BFT1 and BFT2 are set as 30% and 80%, respectively. The drawbead height will be determine by the BFT1 and BFT2. For example, if the drawbead force is less than 30%, the stepbead height will be implemented in the draw simulation.

Auto-Position enables the user to set the fixed tool after user click the apply button to do the auto-position operation etc.

8.2.4 THE QUICKSETUP/DRAW PROCEDURE

The following is the procedure for setting up a draw simulation using QS/Draw:

1. Select draw type and tooling Option
2. Define tooling. Refer to Section 8.1.1
3. Define blank parameters. Refer to Section 8.1.2
4. Define tool motion. Refer to Section 8.2.2
5. Define drawbeads. Refer to Section 8.2.1
6. Apply symmetry condition. The user can define symmetry boundary condition by clicking on CONSTRAINT. Refer to Section 5.7.2 for detailed information about SPC options.
7. Select APPLY and SUBMIT JOB. After selecting APPLY, eta/DYNAFORM will create the matching tool, automatically adjust tooling position, generate travel curve and offset contact interface. The SUBMIT JOB button allows the user to run the simulation via the ANALYSIS dialog window (refer to Section 13.1).

8.3 SPRINGBACK

The QS/Springback GUI (as shown in Figure 8.3.1) allows the user to easily setup a springback analysis.

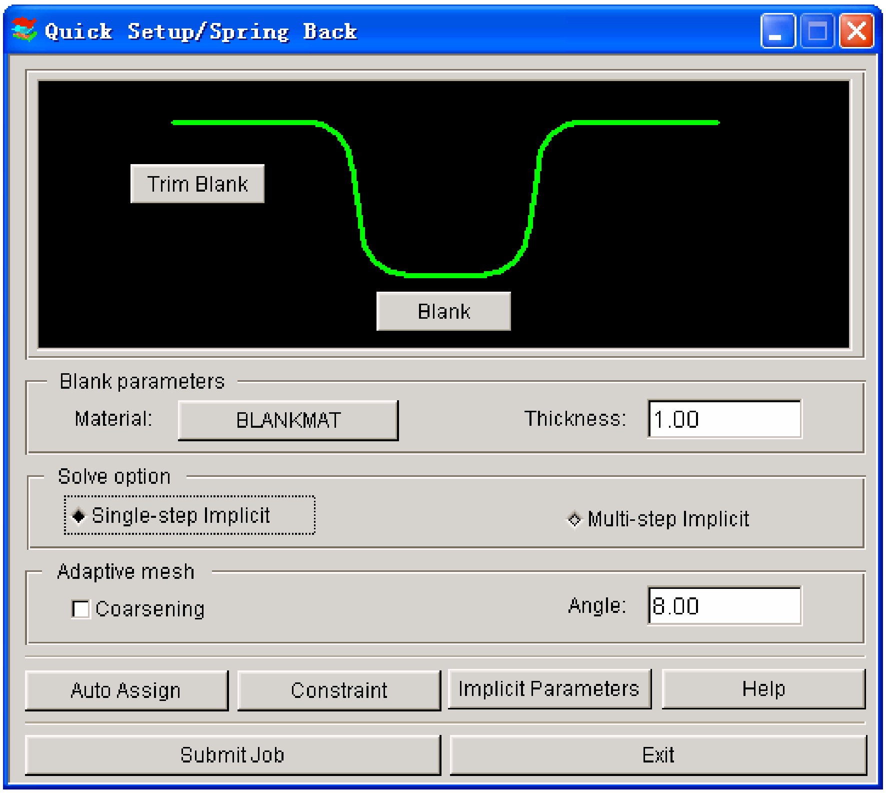


Figure 8.3.1 The QS Springback GUI.

Tool definition GUI

- Allows the user to define deformed blank. Selecting the TRIM BLANK button carries out the trim operation. The user has to read in the trim curve. By default, Blank will be assigned automatically after the deformed blank (in DYNAIN format) is read into the eta/DYNAFORM database.

Blank Parameters

- Defines 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 can select the multi-step implicit method for the springback analysis.

- The coarsen control can be activated to combine the refined mesh prior to springback analysis. The function allows the eta/DYNAFORM solver to combine neighbouring elements that have an angle of less than a certain degree (default is 8 degrees). Coarsening blank mesh can reduce computational noise, and therefore helps the springback model to converge.

Constraint

- Enables the user to select proper clamp points to prevent rigid body motion during springback analysis.

Implicit Parameters

- Defines parameters for springback analysis. The dialog window is shown in Figure 8.3.2. For detailed information about each parameter, refer to LS-DYNA User's Manual.

Figure 8.3.2 The Implicit Parameters dialog window.

8.3.1 The QUICK SETUP/SPRINGBACK PROCEDURE

The 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 DYNAIN file after each draw simulation. The user can read in the DYNAIN file by selecting BLANK from the QS/Springback tool GUI and then selecting IMPORT MESH from the DEFINE BLANK dialog window.

2. Trim the blank
   If no trim operation is involved, the user can skip this step and continue to Step 3. Refer to Section 9.7.5 for information about trimming blank.

3. Define blank parameters

The user needs to define the initial blank material and thickness. The blank parameters are needed for the solver to perform initialization prior to springback analysis. These blank parameters should be similar to the blank parameters used in draw simulation.. After the initialization, the thickness, stress and strain recorded in DYNAIN file will override the initial blank parameters.

4. Select implicit method and coarsen options.

5. Define Implicit parameters
   There are no appropriate default implicit parameters which are good for all springback applictions. 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 window (Figure 8.3.2). eta/DYNAFORM will prompt the user to select the contrained nodes.

For most applications, it is recommended to select 3 constraint nodes 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 this 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.

CHAPTER 9
TOOL DEFINITION

The functions provided in this menu are shown in Figure 9.1. The user can set up tool definition, define materials and properties, generate load curves, define contact interface, position tools, define draw bead, view the movement of defined tools and modify deformed blank shapes.

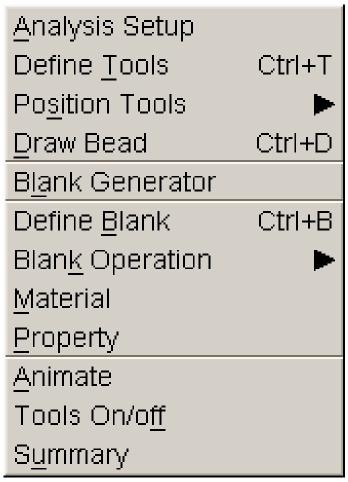


Figure 9.1 Tool menu.

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

9.1 ANALYSIS SETUP

This function enables the user to define forming parameters for the LS-DYNA ANALYSIS. The dialog window is shown in Figure 9.1.1.

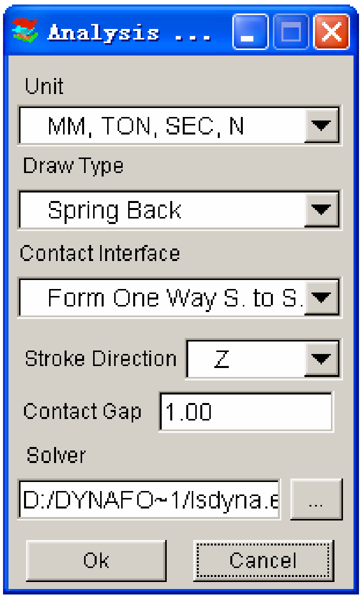


Figure 9.1.1 The Analysis Setup dialog window

Unit
The user can select a UNIT system by clicking the drop down menu of UNIT. As shown in Figure 9.1.2, four types of UNIT systems are available. The default UNIT system is set as <MM, TON, SEC, N>.

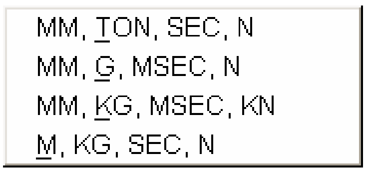


Figure 9.1.2 The options provided in the Unit System drop down menu.

Draw Type
The user can select a draw type, as shown in Figure 9.1.3, by clicking on the drop down menu of DRAW TYPE.

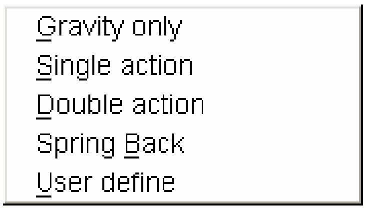


Figure 9.1.3 The options provided in the Draw Types drop down menu

- GRAVITY ONLY
  No punch and die is used as the blank is deformed by gravity.

- SINGLE ACTION
  The punch (male piece) is above the blank.

- DOUBLE ACTION
  The punch (male piece) is under the blank.

- SPRING BACK
  This is read in DYNAIN file

- USER DEFINE
  The DRAW TYPE is user defined

Note: The defined DRAW TYPE works in conjunction with the BLANKOPERATION/AUTO POSITION and TOOL/AUTOPOSITION

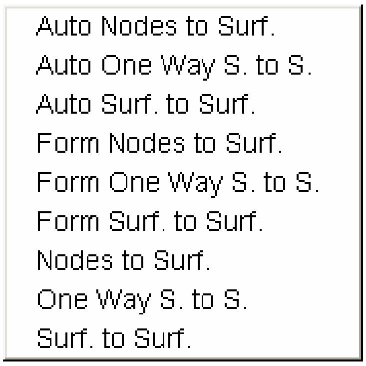


Figure 9.1.4 Types of Contact Interface for stamping simulation.

Contact Interface

There are several types of contact interface available for an LS-DYNA stamping simulation as listed in Figure 9.1.4. The user can set the type of contact interface between all tooling and blank. The default contact interface between blank and all tooling is set as <Form One Way S. to S.>.

Note: Please refer to the *LS-DYNA User's Manual*, keyword *CONTACT, for more detailed information about each of the contact interfaces.

Stroke Direction

The user can define stroke direction in X, Y, or Z.

Contact Gap

The contact gap is the minimum distance between the tooling and blank after auto-positioning. The default contact gap is set to 1.0. The contact gap will be overwritten if blank thickness is defined.

Solver

The user can click the button to locate the location of the LS-DYNA solver from the SELECT SOLVER window as shown in Figure 9.1.5. After clicking OK, the entire directory will be displayed in the field as in Figure 9.1.1.

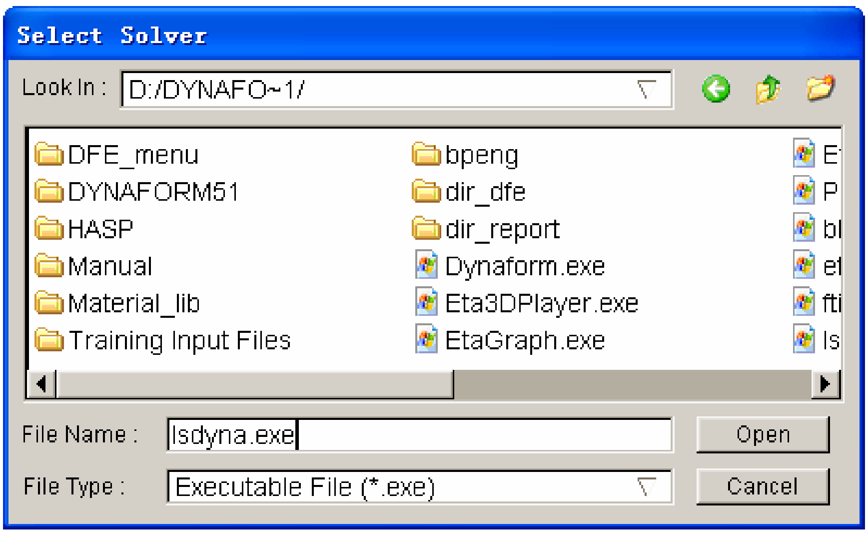


Figure 9.1.5 Select Solver window.

9.2 DEFINE TOOLS (CTRL+T)

In eta/DYNAFORM, there are three standard tools: DIE, PUNCH and BINDER. The user can also create as many tools as necessary for any stamping simulation. The top two toggle switches in the dialog window, as shown in Figure 9.2.1, provide options for using STANDARD TOOLS setup or USER DEFINED TOOLS setup.

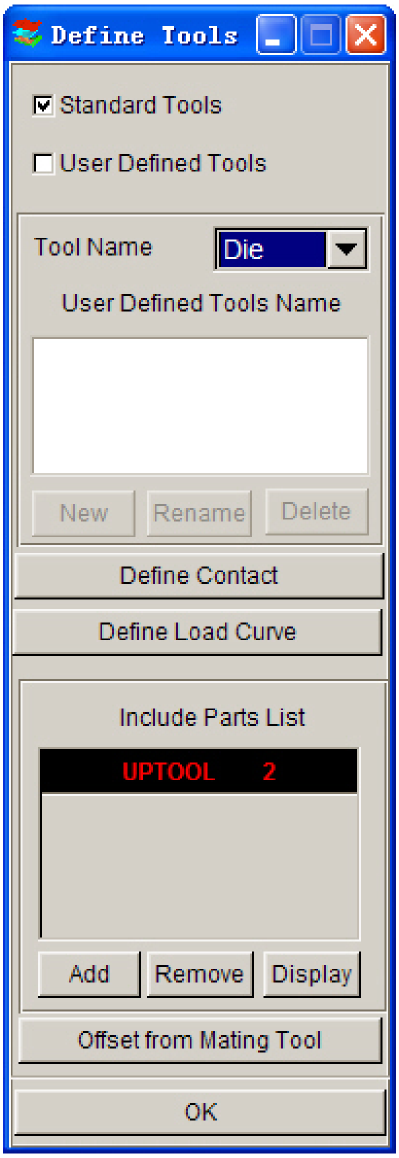


Figure 9.2.1 Define Tools dialog window

9.2.1 TOOLS SETUP

When the STANDARD TOOLS check box is toggled ON, the user can define the standard tools (DIE, PUNCH and BINDER) by selecting from the TOOL NAME drop down menu as shown in Figure 9.2.1. When the USER DEFINED TOOLS check box is toggled ON, the user can define and/or edit any user-defined tool by using the following functions:

NEW
Creates a new tool name.

Figure 9.2.2 User Defined Tool Name window.

RENAME
Renames new tools.

DELETE
Removes tools from the USER DEFINED TOOL list.

9.2.2 ADD (PART)

In general, a tool consists of a set of parts. This function enables the user to add parts to the current tool. To add a part, the user needs to click on the ADD button and then choose a part from the part list. See Section 4.5, PART/CURRENT PART, for more information. The selected parts will be listed in the INCLUDE PARTS LIST window as shown in Figure 9.2.1.

9.2.3 REMOVE (PART)

This function allows the user to remove parts from the current tool. To remove the part, the user needs to select/highlight a part from the INCLUDE PARTS LIST window prior to clicking the REMOVE button.

9.2.4 DISPLAY (TOOL)

This function is used to display all the parts included in the current tool definition in the display area. To display the part's current tool, the user needs to click the DISPLAY button.

9.2.5 DEFINE CONTACT

This function is used to define contact interface between the blank and a tool. In most cases, the default setting, *CONTACT_FORMING_ONE_WAT_SURFACE_TO SURFACE, is utilized for all tooling contact. The user can modify the contact interface for individual tools.

To change the default contact interface settings, click on the DEFINE CONTACT button as shown in Figure 9.2.1. The TOOLS CONTACT dialog window (Figure 9.2.3) appears.

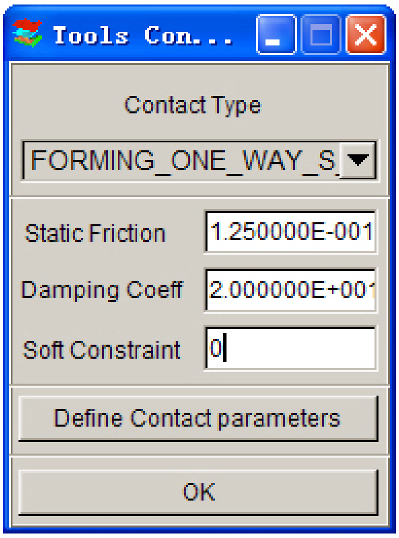


Figure 9.2.3 Define Contact dialog window.

CONTACT TYPE

Select the CONTACT TYPE combination list box, and select the desired type.

NOTE: Nine commonly used contact interfaces in sheet metal forming are offered in eta/DYNAFORM. They are:

1. AUTO-NODES-TO-SURF
2. AUTO-ONE-WAY-SURF-TO-SURF
3. AUTO-SURF-TO-SURF
4. FORMING-NODES-TO-SURF
5. FORMING-ONE-WAY-SURF-TO-SURF
6. FORMING-SURF-TO-SURF
7. NODES-TO-SURF
8. ONE-WAY-SURF-TO-SURF
9. SURFACE-TO-SURFACE

DEFINE CONTACT PARAMETERS

This function allows the user to set the contact parameters such as static friction, damping coefficient, soft constraint, etc.

Note: Refer to the *LS-DYNA User's Manual* for detailed information about each contact interface and its associated parameters.

9.2.6 DEFINE LOAD CURVE

This function allows the user to define, read in, or modify MOTION and/or FORCE load curves. As shown in figure 9.2.4, the user can define the curve type by toggling either MOTION or FORCE. The MOTION curve allows the user to define the tool movement via velocity or displacement control, while the FORCE curve allows user to define a given force applied onto a tool. If both the MOTION and FORCE curves are defined for a tool, the MOTION curve will overwrite the FORCE curve control. However, the force control can be activated if a death time is set for the MOTION curve (the default death time is the termination time of the MOTION curve).

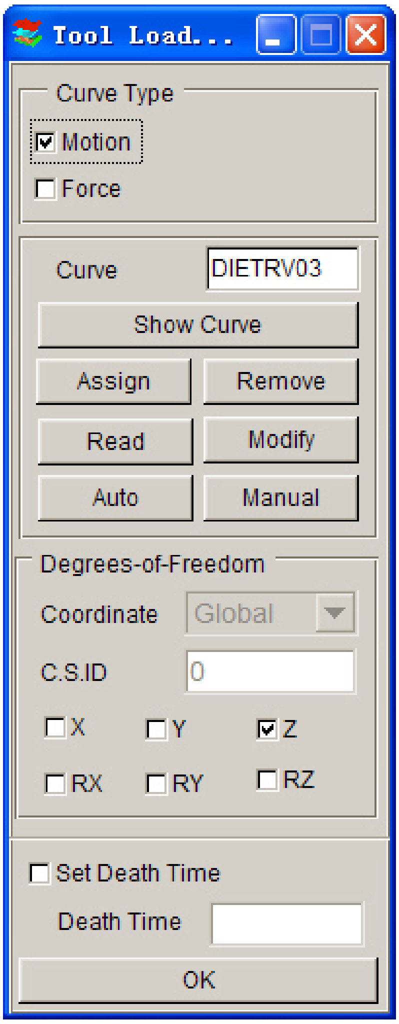


Figure 9.2.4 Tool Load Curve dialog window.

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 window. As shown in Figure 9.1.5, the user can select a particular load curve from the list. Clicking OK confirms the selection and completes the ASSIGN CURVE function.

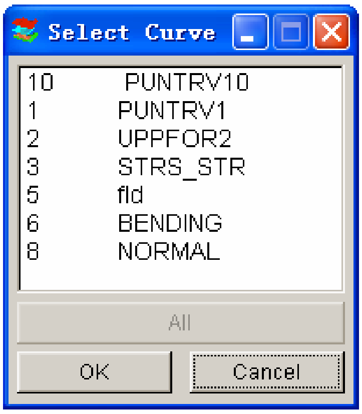


Figure 9.2.5 List Curve

READ

Allows the user to read in an external load curve using the window in Figure 9.1.6.

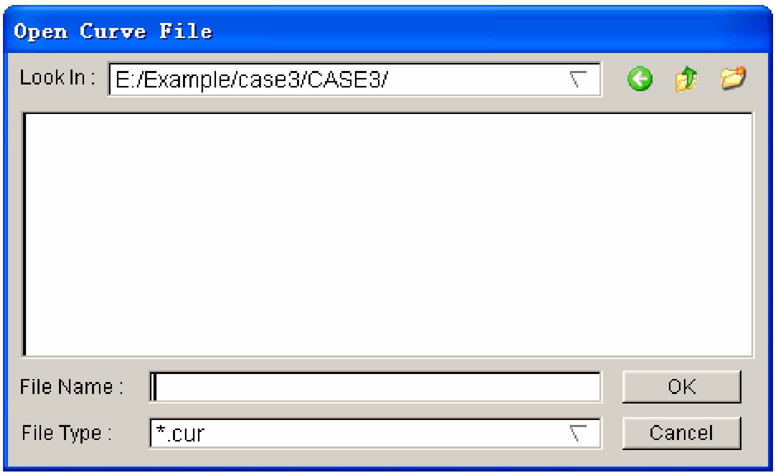


Figure 9.2.6 Read Curve File

AUTO

This function is used to generate a velocity/displacement/force load curve according to the begin time, velocity and stroke distance as shown in Figure 9.2.7. There are 3 different curve shapes: Trapezoidal, Sinusoidal and Sinusoidal with Hold. If MOTION CURVE is checked, as shown in Figure 9.2.4, the MOTION CURVE dialog window is displayed as in Figure 9.2.7.

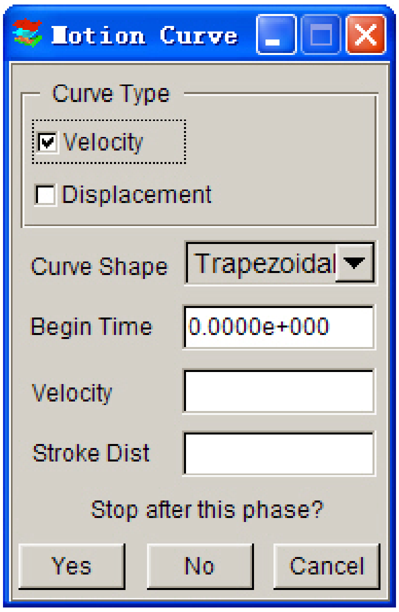


Figure 9.2.7 Auto Motion Curve dialog window.

As shown in Figure 9.2.7, the MOTION CURVE dialog window enables the user to define either the velocity or the displacement load curve. If the VELOCITY check box is toggled ON, the load curve is plotted as Velocity versus Time. DISPLACEMENT enables the user to plot a Displacement versus Time 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.

- Input values in the fields of VELOCITY, DISTANCE, and BEGIN TIME.
- If there is more than one travel phase in the curve, select NO, and input types and values for a second travel phase.
- Select YES to create the motion curve. A dialog window showing the motion curve will show up. Refer to Section 11.9.8, UTILITIES/LOADCURVE/SHOW LOAD CURVE. Select NO, and the

AUTO CURVE window will be refreshed with the begin time, starting with the end time of the first travel.

If FORCE CURVE is checked, the FORCE CURVE dialog window (as shown in Figure 9.2.8) is displayed.

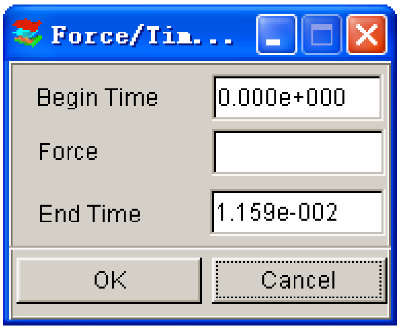


Figure 9.2.8 Auto Force Curve dialog window

- Input values in the fields of BEGIN, FORCE, and END TIME. The default end time is the end time of all motion curves.
- Select OK to create the force curve. A dialog window showing the force curve will show up. Refer to Section 9.9.8, UTILITIES/LOAD CURVES/SHOW LOAD CURVE.

REMOVE

This function is used to remove the applied load curve. If the REMOVE button is clicked, a warning message (as shown in Figure 9.1.9) will ask the user to confirm or deny the operation.

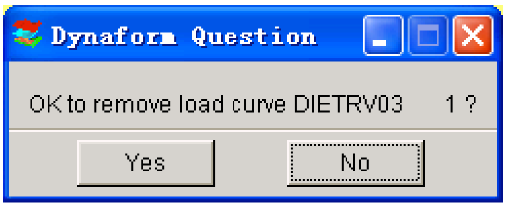


Figure 9.2.9 Remove Warning

MODIFY

This function is used to edit the current curve. Refer to Section 11.9.4 for more details.

MANUAL

This function is used to manually create a curve by input data points. The dialog window is shown in Figure 9.2.10.

- Select VELOCITY or DISPLACEMENT.
- Input TIME and VALUE (velocity or displacement).
- Select ADD POINT, and then input the next data points.
- Select OK to end curve definition.

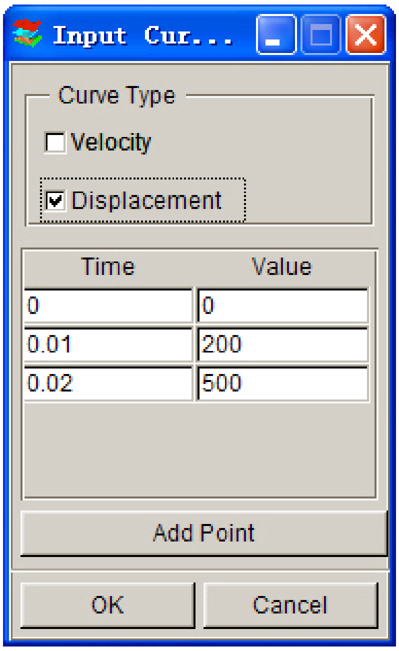


Figure 9.2.10 The Input Curve dialog window.

9.2.7 GENERATE A TOOL FROM MATING TOOLS

This function is used to generate the tool mesh from an existing tool. It combines the operations of copying or offsetting elements, creating parts, and adding parts to the current tool definition.

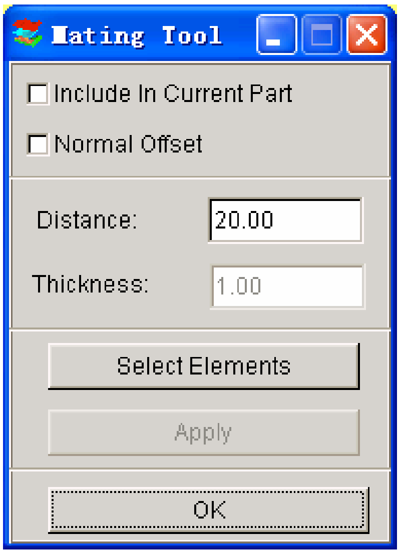


Figure 9.2.11 The Mating Tool dialog window.

- Select OFFSET FROM MATING TOOL and the MATING TOOL dialog window (as shown in Figure 9.2.11) is displayed.

- Toggle the INCLUDE IN CURRENT PART function ON/OFF. If the function is ON, the new copied or offset elements will be included in the current part, and the current parts will be included in the current tool definition. If the function is OFF, a new part will be automatically created and added to the current tool definition.

- The NORMAL OFFSET toggle 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 new tool along stroke direction by inputting DISTANCE. The distance is measured from the original tool mesh.

- To offset or copy elements, select APPLY.

- Select OK to exit the dialog window.

9.3 POSITION TOOLS

The POSITION TOOLS menu contains a submenu that offers the following operations:

- Auto Position
- Move Tool
- Min. Distance

9.3.1 AUTO POSITION

This function enables the user to automatically reposition the defined tools according to the user-defined draw type, tool clearance and the stroke direction. Refer to Section 9.1, ANALYSIS SETUP, to change the default setting.

9.3.2 MOVE TOOL

This function allows the user to translate any tool in any current C.S (Global or User Defined) direction. The dialog window is shown in Figures 9.3.1 and 9.3.2. To move the tool, the user selects the target tool from the tool list, selects the direction, and keys in the distance for translation. Selecting APPLY completes the translation. The user can select REVERSE to translate the tool in the opposite direction.

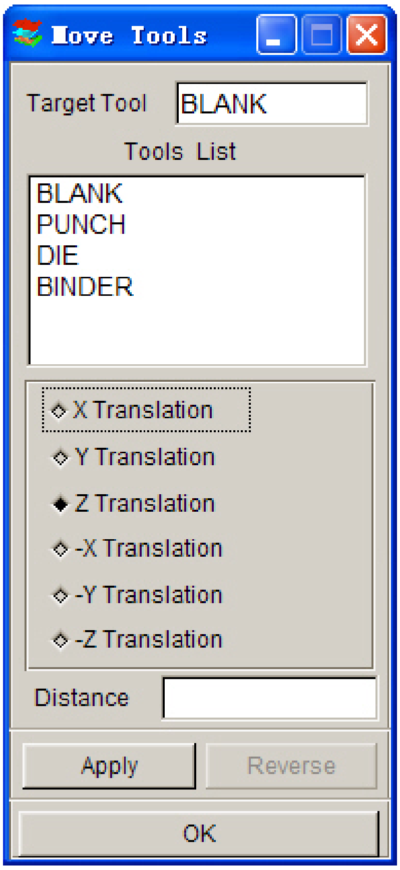


Figure 9.3.1 Global C.S is Current C.S

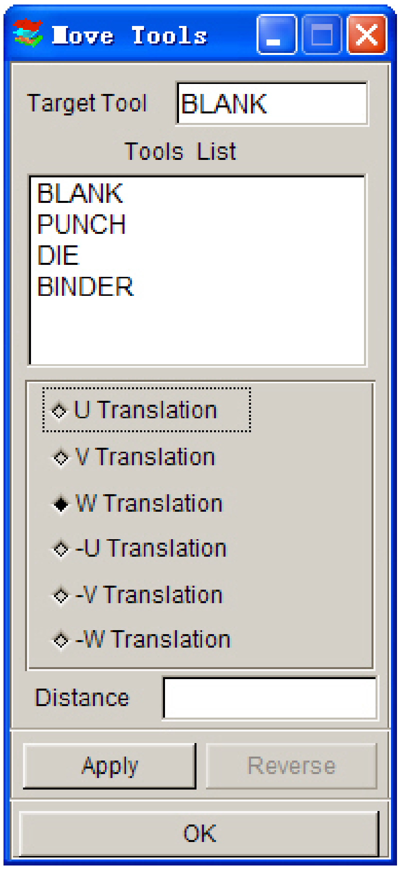


Figure 9.3.2 Local C.S is Current C.S

9.3.3 MIN (IMUM) DISTANCE

This function allows the user to measure the minimum distance between two selected tools in the selected direction. The dialog window is shown in Figure 9.3.1. To measure the minimum distance, the user has to select the MASTER and SLAVE TOOLS as shown in Figure 9.3.1.

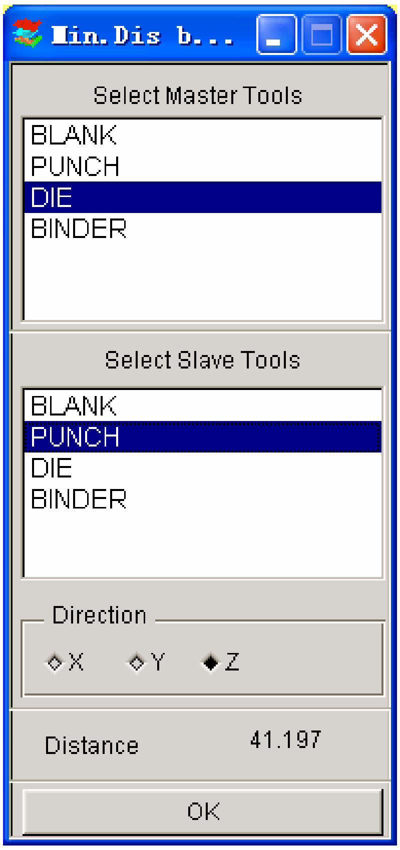


Figure 9.3.3 The Min. Distance dialog window.

9.4 DRAW BEAD (CTRL+D)

The functions in the DRAW BEAD menu are used to create, modify, and assign draw beads. The dialog window is shown in Figure 9.4.1.

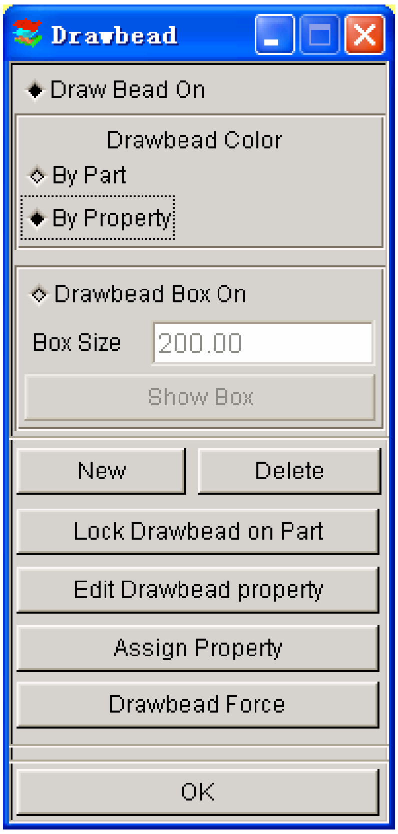


Figure 9.4.1 The Define Draw bead dialog window.

9.4.1 DRAWBEAD ON

If toggled ON, the drawbead lines are displayed.

9.4.2 DRAWBEAD COLOR

This feature allows the user to display the drawbead lines using either part or property color.

9.4.3 DEFINE BOX AND SHOW BOX

The DEFINE BOX function is used to define a specific box-shaped volume around the drawbead. The box contains the drawbead nodes and elements between the bead and the outer edge of the blank. It is implemented to limit the size of the master surface for the drawbead contact. The BOX SIZE is the linear distance between the drawbead line and the surfaces of box. The SHOW BOX function is used to graphically display the draw bead boxes that have been defined.

To define/show the drawbead box, the user has to toggle ON the DRAWBEAD BOX, then key in the box size and click the SHOW BOX button to display the drawbead box on the display area.

9.4.4 NEW (DRAWBEAD)

This function allows the user to create new drawbead and its properties. In eta/DYNAFORM, a drawbead is a line consisting of a series of continuous nodes with consistent node identification numbers. The user can create a new drawbead by following steps listed below:

1. Select NEW from the DRAWBEAD dialog window as shown in Figure 9.4.1.
2. Select the property color from the DRAWBEAD PROPERTY dialog window as shown in Figure 9.4.5
3. Click NEW to create new drawbead properties. The DRAWBEAD PROPERTIES table is shown in Figure 9.4.6. It allows the user to set the drawbead restraining force, normal force, depth, percentage of lock force, etc.
4. Select OK to close the window. The CREATE DRAW BEAD dialog window, as shown in Figure 9.4.2, is displayed. It allows the user to create drawbead using line, point or node. If drawbead line data is available, select LINE, and pick the drawbead line data shown in the display area. Click OK to confirm the line selection and to generate the drawbead.

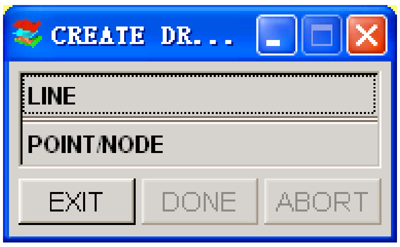


Figure 9.4.2 The Create Drawbead dialog window.

The user can also generate drawbead by selecting a series of points/nodes.

In order to create multiple drawbeads using a different property, select EXIT or DONE, and repeat steps 1 and 4.

9.4.5 DELETE (DRAWBEAD)

This function enables the user to remove drawbead(s) from the database using the options given in the SELECT DRAWBEAD dialog window as shown Figure 9.4.3.

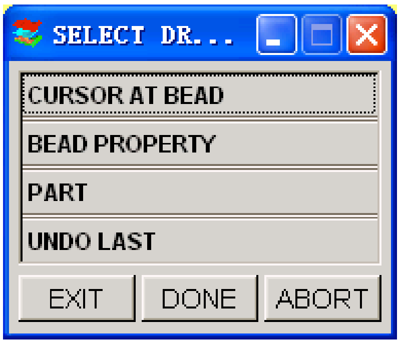


Figure 9.4.3 The Select Drawbead dialog window.

CURSOR AT BEAD

Allows the user to select drawbead using the mouse cursor

BEAD PROPERTY

Allows the user to select drawbead from the DRAWBEAD PROPERTY dialog window.

PART

Allows the user to select drawbead (locked on part) from the SELECT PART dialog window.

UNDO LAST

Allows the user to cancel the last drawbead selection.

9.4.6 LOCK DRAWBEAD ON PART

This function allows the user to assign a drawbead to be attached to a rigid body part. The user can select the rigid body part from the SELECT PART dialog window. After selecting the target part, the user selects drawbead(s) using the options given in the SELECT DRAWBEAD dialog window as shown in Figure 9.4.3. After accepting the selected drawbead(s), the DYNAFORM QUESTION window, as in Figure 9.3.4 , will be displayed. The user can select YES to project the drawbead(s) on the target part mesh. Selecting NO will assign the selected drawbead(s) to the target part.

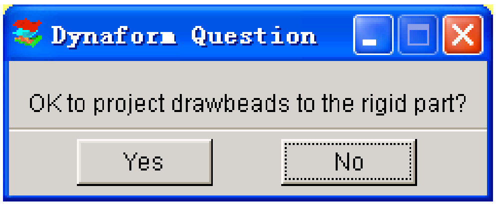


Figure 9.4.4 DYNAFORM Question Window

9.4.7 EDIT DRAWBEAD PROPERTY

The DRAWBEAD PROPERTY dialog window is shown in Figure 9.4.5. This function enables the user to create or edit drawbead properties. The functions of the buttons (NEW, MODIFY, DELETE) in this dialog window are similar to those described in the MATERIAL and PROPERTY sections (9.6 and 9.7). The table of DRAWBEAD PROPERTIES is shown in Figure 9.4.6.

Figure 9.4.5 The Drawbead Property dialog window.

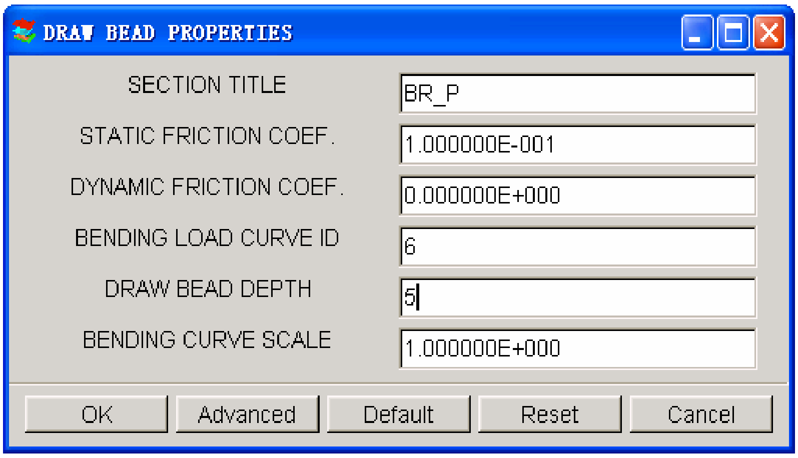


Figure 9.4.6 The Drawbead Properties Table.

9.4.8 ASSIGN PROPERTY

This function allows the user to reassign the property of a defined drawbead. The steps for reassigning a drawbead property are similar to those of creating a draw bead.

- Select ASSIGN PROPERTY from the DRAWBEAD dialog window to display the SELECT BEAD PROPERTY dialog window as shown in Figure 9.4.7.

Figure 9.4.7 The Select Drawbead Property dialog window

- Select PROPERTY NAME and pick a drawbead property from the DRAWBEAD PROPERTY dialog window. Click OK to confirm the selection.

- Select CURSOR AT DRAW BEAD and choose an existing draw bead. The property of the chosen draw bead is selected. Click DONE to complete the process.

9.4.9 DRAW BEAD FORCE

This function allows the user to define the drawbead bending and normal forces. The dialog window is shown in Figure 9.4.8.

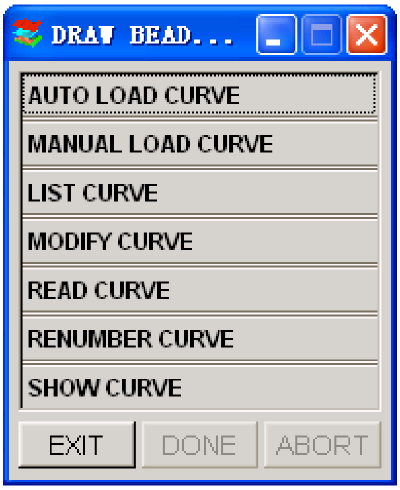


Figure 9.4.8 The Drawbead Force dialog window.

AUTO LOAD CURVE

This function enables the user to calculate the drawbead resistive force based on the input of the drawbead geometry and the mechanical properties of blank. As shown in Figure 9.4.9, the user can select the drawbead type and input different parameters.

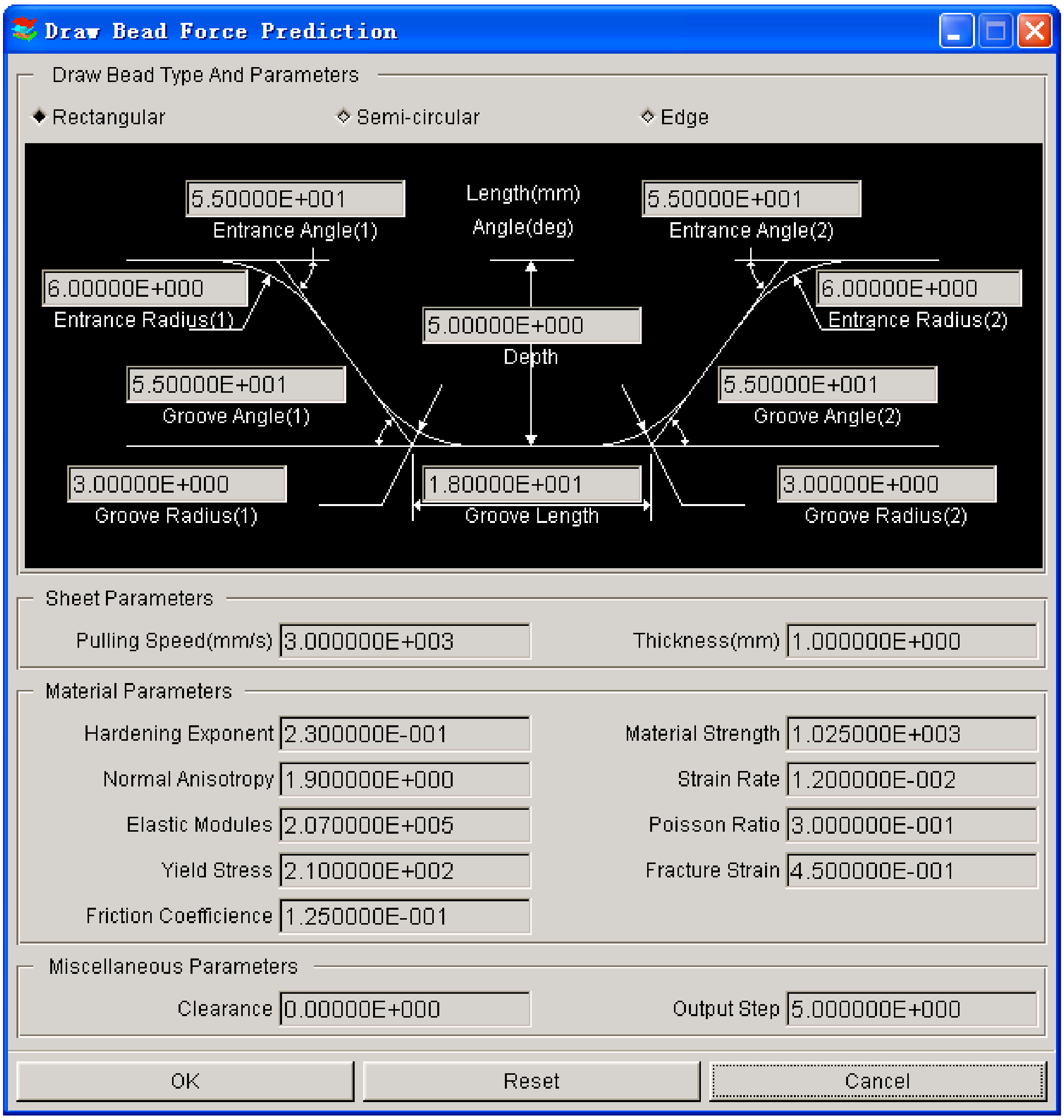


Figure 9.4.9 The Drawbead Force Prediction dialog window.

MANUAL LOAD CURVE

The user can manually create drawbead bending and normal force curves using the INPUT CURVE dialog window as shown in Figure 9.4.9. Refer to Section 11.11.1.

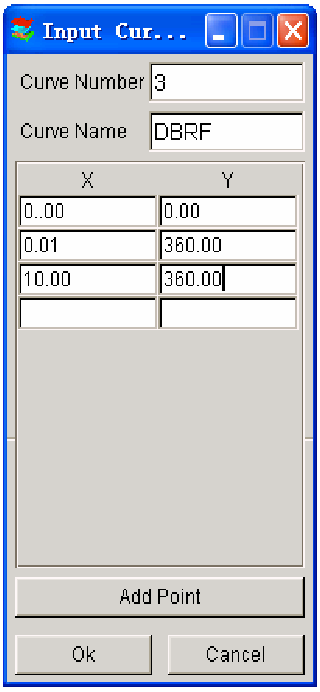


Figure 9.4.9 The Input Curve dialog window.

LIST CURVE
Lists a selected load curve or all load curves in the database.

MODIFY CURVE
Enables the user to edit the load curve data points.

READ CURVE
Allows the user to read an external load curve into the database. Refer to Section 11.11.5 for the format of curve data.

RENUMBER CURVE
Enables the user to edit the load curve identification number of a selected load curve. Refer to Section 11.11.7 for the procedure to renumber load curve.

SHOW CURVE
The user can show a curve by cursor selection at the draw bead or by property name. The rest of the options in this menu are similar to those described in Section 11.11.8.

9.5 BLANK GENERATOR

This function is developed mainly for meshing flat blank. The dialog window is shown in Figure 9.5.1.

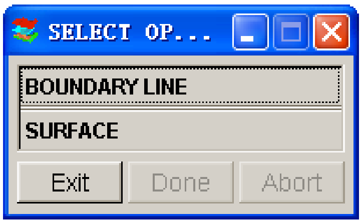


Figure 9.5.1 The Select Option for Blank Generator.

BOUNDARY LINE

Allows the user to create flat blank mesh using boundary lines

SURFACE

Allows the user to create flat blank mesh using existing surface

After selecting boundary lines or surface, the user needs to key in the concerned tool radii from the INPUT window. The concerned tool radii are usually the minimum die radii. These will determine the initial blank element size. A DYNAFORM QUESTION window appears, asking the user to select the options as listed below:

YES – Accept the blank mesh.
NO – Cancel the operation
REMESH – The CONCERN TOOL RADII window will be displayed again and the user can adjust the radius.

9.6 DEFINE BLANK (CTRL+B)

This function allows the user to define the blank material and properties of the stamping simulation. The DEFINE BLANK dialog window is shown in Figure 9.6.

Figure 9.6 The Define Blank dialog window.

SINGLE SURFACE

This option allows the user to activate the single surface contact interface for blank. Activating the single surface contact interface is recommended in order to accurately model the blank folding condition during the stamping simulation.

ADD PART

This function allows the user to add parts to the blank definition. It is similar to adding parts to tool definition as in Section 9.1.1.

DEFINE BLANK MATERIAL

Allows the user to define the mechanical properties of blank. Refer to Section 9.8.

DEFINE BLANK PROPERTY

Allows the user to define the shell element properties and thickness of blank. Refer to Section 9.9.

9.7 BLANK OPERATION

This menu contains eight different submenus which are used to manipulate the blank geometry, view blank results, map blank results, etc. The submenus are listed below:

- BLANK AUTO POSITION
- BLANK MAPPING
- RESULT MAPPING
- DYNAMIC CONTOUR
- TRIM
- TIP
- TAILOR WELDED
- LANCING

9.7.1 BLANK AUTO POSITION

This function is aimed to automatically reposition the blank with reference to the selected tool and given contact gap. The blank can be positioned along X, Y or Z coordinates in either a global or local coordinate system. The dialog window is shown in Figure 9.7.1.

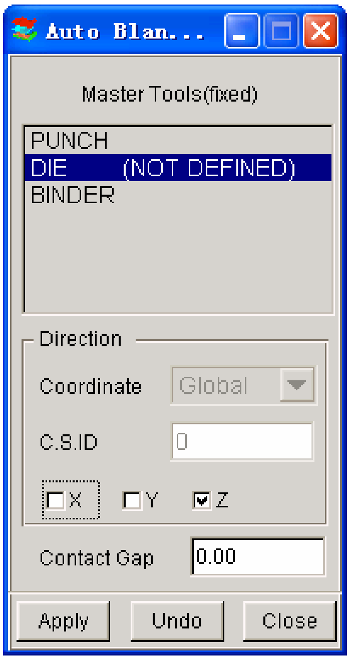


Figure 9.7.1 The Auto Blank Position dialog window.

9.7.2 BLANK MAPPING

This function is utilized to project a flat blank with refined mesh onto a deformed blank with coarser mesh. This application is useful for converting the coarse blank mesh (from gravity loading simulation) to refined blank mesh needed for binder wrap and/or drawing simulation.

If BLANK MAPPING is selected, the eta/DYNAFORM will prompt the user to select the flat blank with coarse mesh. The message is printed in the prompt message window as:

SELECT COARSE MESH FOR THE FLAT BLANK

The user selects the coarse blank mesh from the SELECT PART dialog window. The flat blank with coarse mesh is the initial blank mesh used in gravity loading simulation. After the selection of the flat blank with the coarse mesh, eta/DYNAFORM will prompt the following message:

SELECT COARSE MESH FOR THE DEFORMED BLANK

The user selects the deformed blank with coarse mesh that is yielded by gravity loading simulation. After the selection of the deformed blank with the coarse mesh from the SELECT PART dialog window, eta/DYNAFORM will prompt the following message:

SELECT FINE MESH FOR THE FLAT BLANK

The user selects the flat blank with the refined mesh. Both flat blanks share identical dimensions. eta/DYNAFORM will automatically map the refined blank mesh to the deformed blank shape. The deformed blank with finer mesh is then used for subsequent binder wrap and/or drawing simulation.

9.7.3 RESULT MAPPING

The purpose of this function is two-fold:

- To map forming results to the car position part mesh for structure analysis.
- To tip a blank without interfering with the thickness or strain/stress information.

Before using the RESULT MAPPING function, the user must import the desired (targeted) part mesh in Nastran format into the eta/DYNAFORM database. The deformed blank (forming results) in DYNAIN format should then be read into the database. The user will continue to carry out the RESULT MAPPING operation using the dialog window as shown in Figure 9.7.2. If there is no DYNAIN file read into the current database, the RESULT MAPPING function is disabled.

The procedure for RESULT MAPPING is as follows:

1. Click both the FROM and TO buttons to select the deformed blank and the desired (targeted) part mesh.

2. Toggle ON COORDINATE TRANSFORM, then define (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 INCREMENTAL ADJUSTMENT 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 the (+) and (-) buttons to manually adjust the deformed blank position. After aligning the two parts, select MAP to complete the RESULT MAPPING operation.

4. Select RESET to return the deformed part to the original position, and select EXIT to dismiss the RESULT MAPPING dialog window without mapping the results.

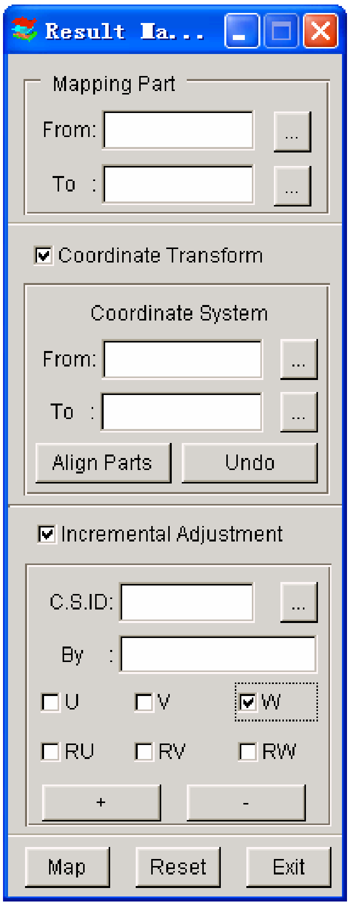


Figure 9.7.2 The Result Mapping dialog window.

9.7.4 DYNAIN CONTOUR

This function allows the user to plot thickness, stress and strain distribution after the blank results (in DYNAIN format) are read into eta/DYNAFORM database. If there is no blank result available in the eta/DYNAFORM database, this function is automatically disabled.

9.7.5 TRIM

This function is used to trim a blank using a closed curve trim line(s). The trimming will be processed based on a defined vector/direction. The user can decide to keep the elements inside/outside of the trim line. These elements are included in a new part called TRIMOUT.

After a trim line is selected, the user has to define a vector (direction of projection) for the trim curve, that is, W-direction or global Z-axis, from the LSC dialog window (as described in Section 2.9). The user can then set the trimming tolerance from the dialog window as shown in Figure 9.7.3. The trimming tolerance ranges between 0 and 1. It limits the size of the small elements generated during the trimming operation. A larger tolerance produces large elements, and a smaller tolerance gives small elements and more detail in the trim line. The default value is set to 0.30.

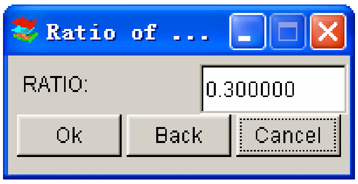


Figure 9.7.3 The Tolerance input window.

Note: Prior to the trimming operation, the user must define the blank properties.

9.7.6 TIP

This function allows the user to tip the blank according to a user-defined coordinate system. The dialog window is shown in Figure 9.7.4.

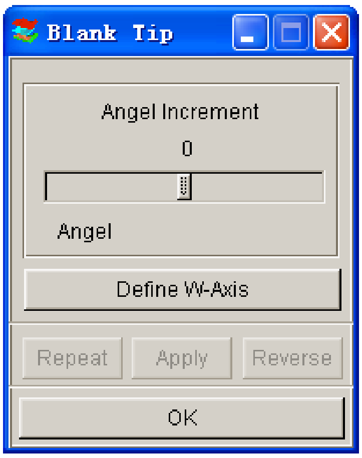


Figure 9.7.4 The Blank Tipping dialog window.

As shown in Figure 9.7.4, the user can define the coordinate system by clicking the DEFINE W-AXIS button. After a coordinate system is set, the user can tip the blank using the REPEAT and REVERSE buttons. The slider bar is used to adjust the tip the angle increment.

9.7.7 TAILOR WELDED

This function allows the user to weld together parts of different material and/or thickness.

If the TAILOR WELDED function is selected, eta/DYNAFORM displays the SELECT PART dialog window, as shown in Figure 4.9.1, and prompts:

SELECT TARGET PART FOR TAILOR

After the target part is selected, eta/DYNAFORM displays the SELECT NODE dialog window, as shown Figure 6.4.2, and prompts:

SELECT NODES TO TAILOR WELD

After the nodes are chosen, the PROMPT window displays the number of spotweld nodes created. The welded part will be shown in the display area.

9.7.8 LANCING

This function allows user to carry out lancing operations by splitting the selected elements along a given path.

If this function is selected, the user must select the elements using the SELECT ELEMENT dialog window. The SELECT METHOD dialog window, as shown in Figure 9.7.5, will appear after the user confirms the element selection.

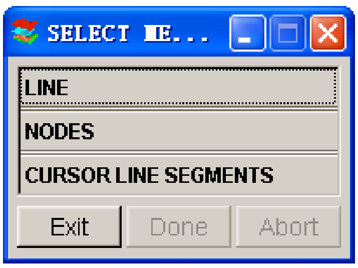


Figure 9.7.5 Select Method

LINE
Allows the user to select an existing line as a splitting path.

NODES

Allows the user to create a line segment by selecting the nodes. The line segment is projected on the selected elements along the defined W-AXIS. It will be used as the splitting path.

CURSOR LINE SEGMENTS
Generates the splitting path using line segments.

9.8 MATERIAL

This dialog window allows the user to create, modify, and delete material definition. In addition, the user can import and export any defined materials. The dialog window is shown in Figure 9.8.1.

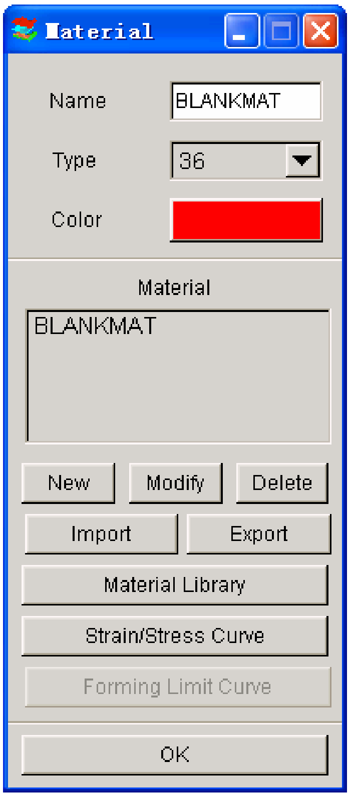


Figure 9.8.1 The Material dialog window.

The user can select the type of material model by clicking the drop down menu as shown in Figure 9.8.1. The default material is set to type #36. Refer to the LS-DYNA User's Manual for detailed information about each material model.

9.8.1 NEW (MATERIAL)

Enables the user to create a new material model and define the associated parameters from the table as shown in Figure 9.8.2.

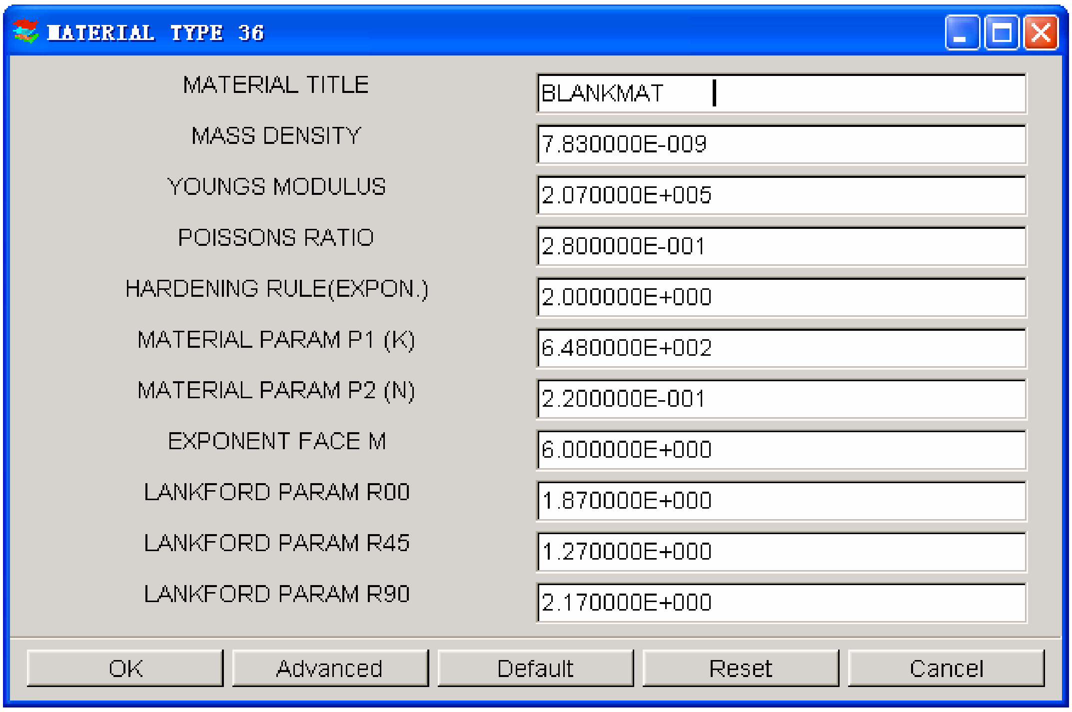


Figure 9.8.2 The parameter table of material model.

9.8.2 MODIFY (MATERIAL)

Allows the user to edit the parameters of a selected material model by using the table as shown in Figure 9.8.2.

9.8.3 DELETE (MATERIAL)

This function removes a defined material model from the database.

9.8.4 EXPORT

This function allows the user to save any defined material in a file with *.mat extension. The EXPORT button will display the EXPORT MATERIAL window as shown in Figure 9.8.3.

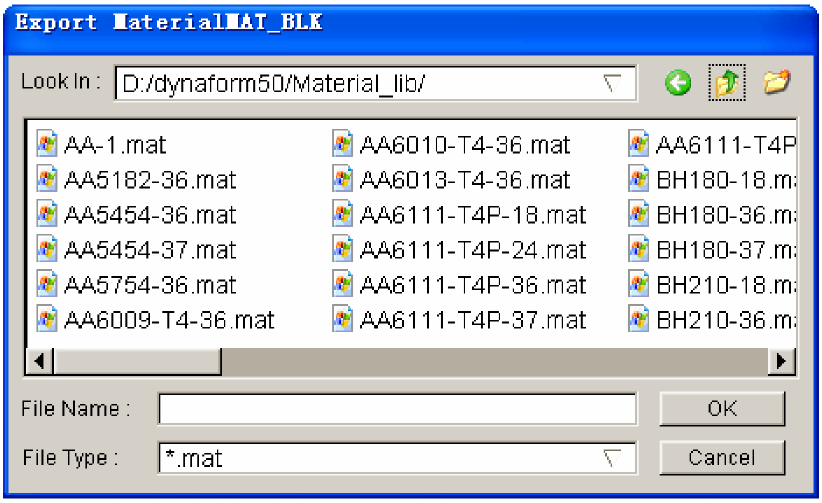


Figure 9.8.3 The Export Material window.

9.8.5 IMPORT

This function allows the user to read in a material from the file with the suffix "mat". Choose a mat file and select OK to close the IMPORT MATERIAL window. The imported material will be displayed in the MATERIAL window.

9.8.6 MATERIAL LIBRARY

This function allows user to select a material directly from the generic material library embedded in eta/DYNAFORM. The MATERIAL LIBRARY is a folder located in the installation directory, "./installation directory/material_lib/…". It consists of *.mat files and DF_MATERIAL_INDEX.dat file. Every *.mat file is one of the LS-DYNA material models. The DF_MATERIAL_INDEX.dat is the index of all *.mat files.

As shown in Figure 9.8.4, the user can select any button from the window, and the relative parameters are included in the MATERIAL PARAMETERS table.

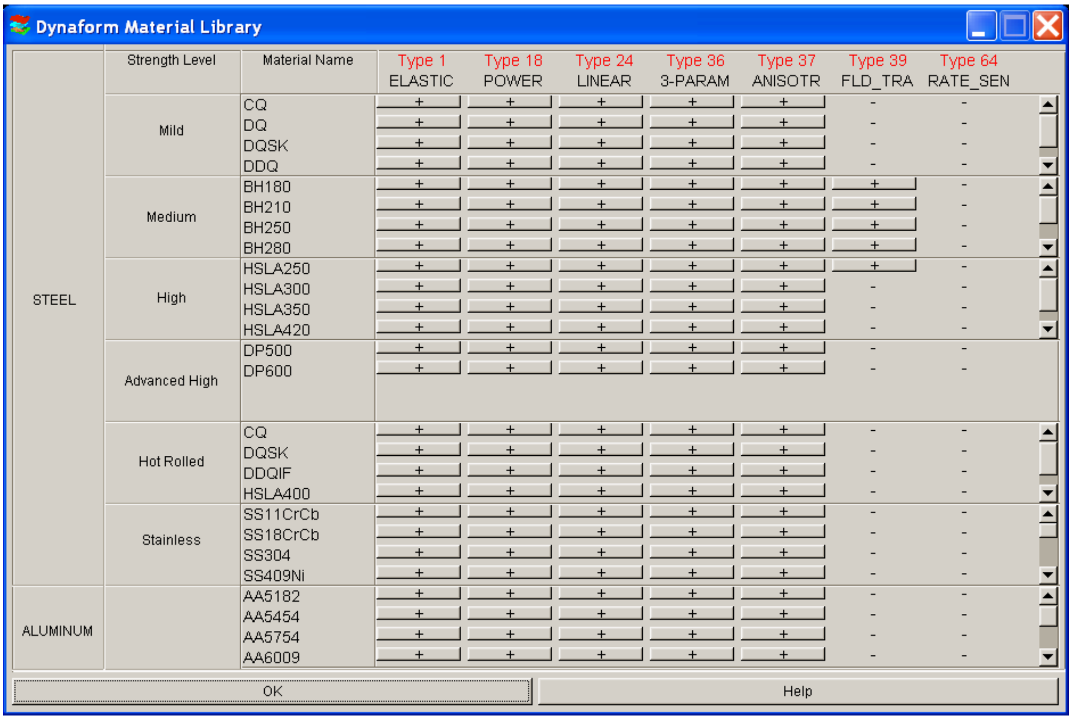


Figure 9.8.4 The Material Library.

9.8.7 STRAIN/STRESS CURVE

The functions in this menu are used to create, read, and display the stress/strain curves of materials. There are six options in this menu, as shown in Figure 9.8.5. The function of each option is similar to those described in Section 12.9, UTILITIES/LOAD CURVE.

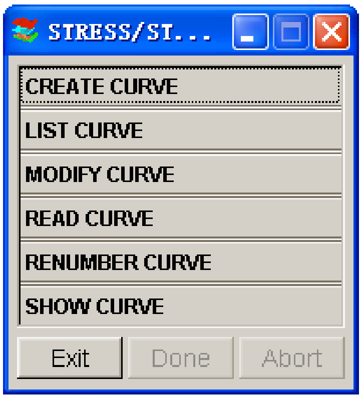


Figure 9.8.5 The Stress/Strain Curve dialog window

9.8.8 FORMING LIMIT CURVE

The functions in this menu are used to create, read, and display the FLD curve of the materials. The dialog window is shown in Figure 9.8.6. These options are similar to those described in Section 12.9, UTILITIES/LOAD CURVE.

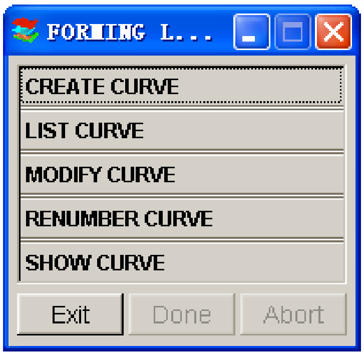


Figure 9.8.6 The Forming Limit Curve dialog window.

Note: There are three different curve formats that are supported by eta/DYNAFORM, i.e, these curve formats can be read into eta/DYNAFORM database.

FORMAT 1: FEMB read in format

```
$ THIS IS THE FEMB FORMAT LOAD CURVE
$ FEMB LOAD CURVE (TITLE LINE HAVE BEEN EXIST)
$CURVE, CURVE ID, TYPE, CURVE NAME (A9,I9,I9,1X,A8)
CURVE    9    0 CURVE9
$CURVE DATA (X,Y) (F9.0,F9.0)
0.0000E+000.1349E+03
0.900E-09.2018E+03
0.3000E-09.2699E+03
0.9000E-09.3017E+03
0.7000E-09.3282E+03
0.9000E-09.3494E+03
0.190E+000.3674E+03
0.1300E+000.3831E+03
0.1900E+000.3970E+03
0.1700E+000.4097E+03
0.1900E+000.4212E+03
0.290E+000.4319E+03
0.2300E+000.4418E+03
0.2900E+000.4911E+03
0.2700E+000.4999E+03
0.2900E+000.4682E+03
0.9990E+09.4722E+02
END
```

FORMAT 2: FEMB write out format (can be read in also)

```
1   (CURVE NUMBER, I9)
    27 'fld' 90 (num. of points, 'curve name', curve id)(4x,i,'a8',i4)
  -4.94296E-01  9.00160E-01
  -4.46287E-01  8.92777E-01
  -4.00478E-01  8.06437E-01
  -3.96679E-01  7.61400E-01
  -3.14711E-01  7.17994E-01
  -2.74437E-01  6.76406E-01
  -2.39722E-01  6.37089E-01
  -1.98491E-01  6.00334E-01
  -1.62919E-01  9.66903E-01
  -1.27833E-01  9.39946E-01
  -9.43970E-02  9.09004E-01
  -6.18794E-02  4.86003E-01
  -3.04992E-02  4.67236E-01
   0.00000E+00  4.92994E-01
   2.99988E-02  4.71172E-01
   9.82689E-02  4.86212E-01
   8.61777E-02  4.98662E-01
   1.13329E-01  9.08994E-01
   1.39762E-01  9.17989E-01
   1.69914E-01  9.24738E-01
   1.90620E-01  9.30703E-01
   2.19111E-01  9.39683E-01
   2.39017E-01  9.39843E-01
   2.62364E-01  9.43322E-01
   2.89179E-01  9.46232E-01
   3.07489E-01  9.48669E-01
   3.29304E-01  9.90709E-01
```

FORMAT 3: (DYNA format)

```
*KEYWORD  (must)
*DEFINE_CURVE
$CURVENAME      ABC
$    LCID     SIDR     SCLA     SCLO     OFFA     OFFO
      200        0
$              A1                O1
     .000000000E+00     .000000000E+00
     .476209790E+00     .770189471E+02
     .992419979E+00     .122399673E+03
     .142862940E+01     .149092189E+03
     .190483916E+01     .164766199E+03
     .238948920E+01     .174017044E+03
     .289729880E+01     .179463074E+03
     .333346844E+01     .182669189E+03
     .380967832E+01     .184996629E+03
     .428988820E+01     .189667770E+03
     .476209784E+01     .186321899E+03
*END
(Here the *END is optional)
```

9.9 PROPERTY

The functions in this menu are used to define and modify the physical properties of the blank. As shown in Figure 9.9.1, the PROPERTY dialog window 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 9.9.1 Property dialog window.

9.9.1 NEW

This function allows the user to create a new blank property. The user can define the name of new blank property, thickness and through thickness integration points from the PROPERTY table.

The functions MODIFY and DELETE, are similar to those described in Section 9.6, MATERIAL.

9.10 ANIMATE

This function is used to show the motion of all defined tools with either velocity or displacement load curve. The user will not be able to view the motion of tools with force load curve. The animation dialog window is shown in Figure 9.10.1.

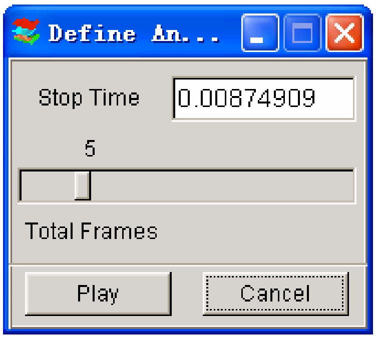


Figure 9.10.1 Define Animation dialog window

STOP TIME

The default stop time is equal to the termination time of the simulation. The user can also set a stop time that is less than the termination time.

SLIDER

Allows the user to set the number of frames for the animation

PLAY

Allow the user to animate all frames. It will also bring up the ANIMATE dialog window as shown in Figure 9.10.2.

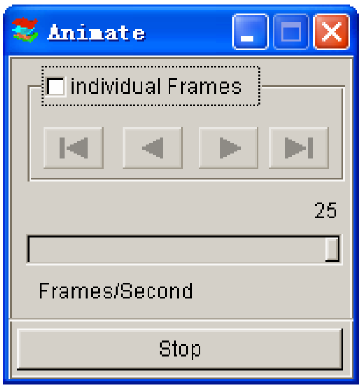


Figure 9.10.2 Animate dialog window.

As shown in Figure 9.10.2, the user can animate the tool motion using INDIVIDUAL FRAMES. When the function is toggled ON, the user can select the FORWARD and REVERSE buttons to view the animation. The slider bar is used to adjust the speed of animation.

9.11 TOOLS ON/OFF

This function allows the user to toggle ON/OFF each individual or all tools. The TOOLS ON/OFF dialog window is shown in Figure 9.11.1.

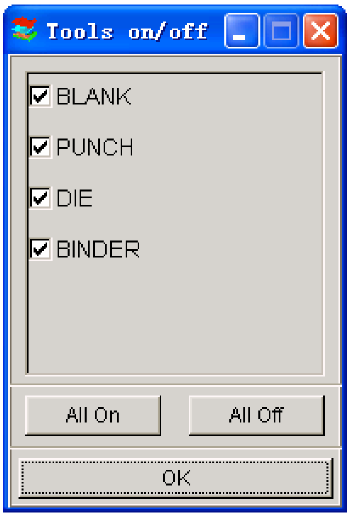


Figure 9.11.1 Tools On/Off dialog window.

9.12 SUMMARY

This function enables the user to review the statistical information of each tool from the SELECT TOOLS dialog window.

After a tool is selected, a message window with tool statistics is displayed. An example of the message window is shown in Figure 9.12.1.

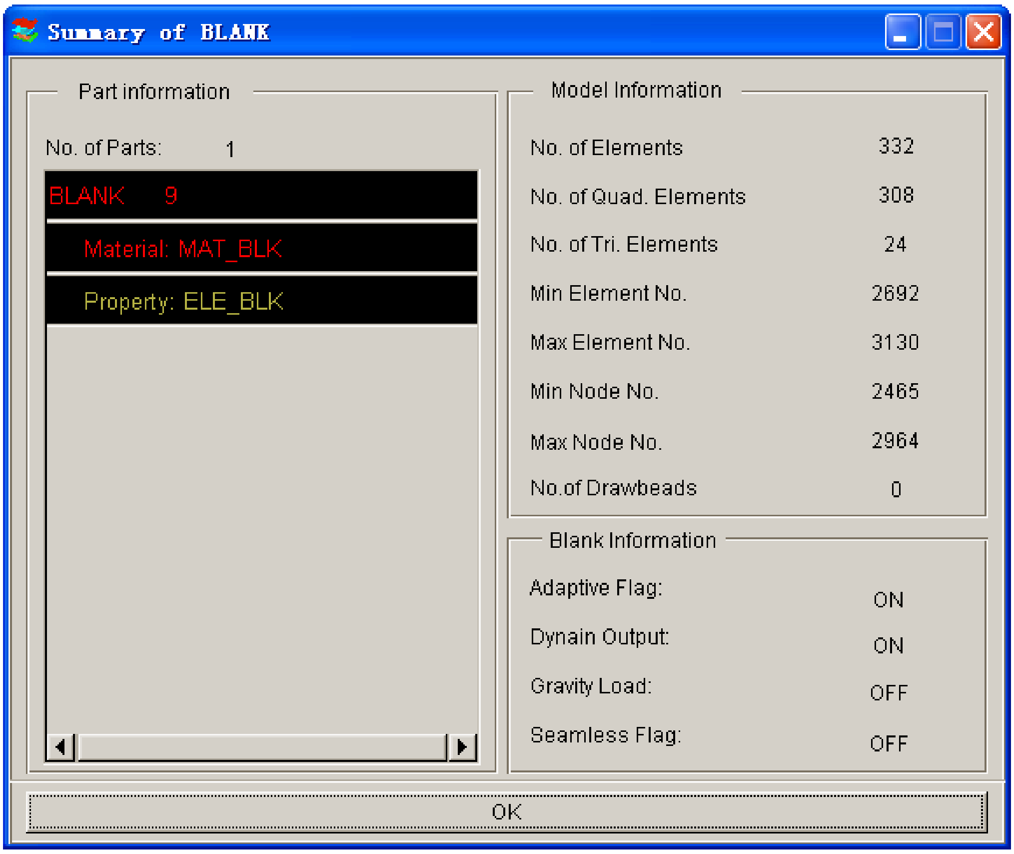


Figure 9.12.1 A sample of tool statistic message window.

CHAPTER 10

OPTION MENU

This menu consists of various preprocessing utilities that are toggle-switch activated as shown in Figure 10.1.

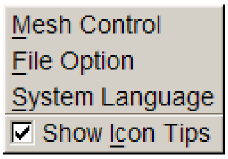


Figure 10.1 Option menu.

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

10.1 MESH CONTROL

The options in this dialog window, Figure 10.1.1, are used to control mesh generation.

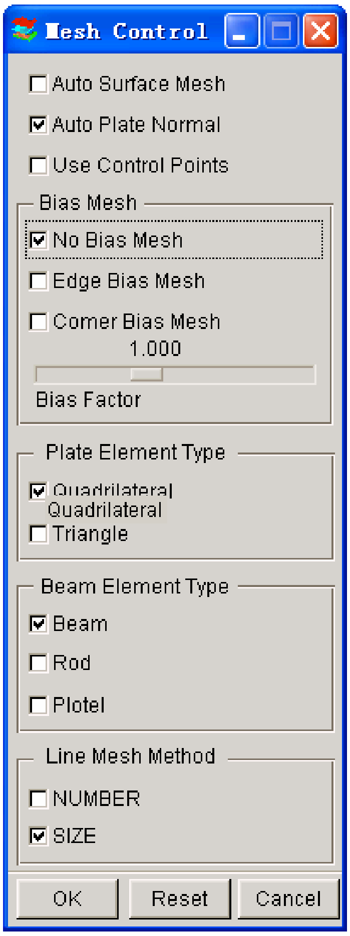


Figure 10.1.1 Mesh Control dialog window.

AUTO SURFACE MESH (toggle)
This function enables the user to mesh the displayed surfaces directly using the SURFACE MESH function.

AUTO PLATE NORMAL (toggle)
This function enables the user to mesh the part with consistent Plate Element normal.

10.1.1 CONTROL POINT (toggle)

This function enables the user to pre-select the points on a line for the desired node locations associated with the 2L, 3L, 4L, 6S, 10S, 9S, and 12S mesh function. It is used in conjunction with the options in Section 6.3.4.

10.1.2 EDGE BIAS MESH

The EDGE BIAS MESH command allows the user to enter an element bias factor (0.625 to 1.6) relative to the selected edge of the line data that is to be modeled. This function multiplies the size of each adjacent element that is created from the selected edge by the bias factor.

If the EDGE BIAS MESH function as shown in Figure 10.1.1 is toggled on, the user can define the bias factor by dragging the slider. An example of mesh with edge bias mesh control is shown in Figure 10.1.2b.

Note: A bias factor greater than 1.0 generates proportionately larger elements from the selected edge. A bias factor less than 1.0 generates proportionately smaller elements from the selected edge. This function can only be implemented with the 2L, 3L, 4L, 6S, 10S, 9S, and 12S meshing function.

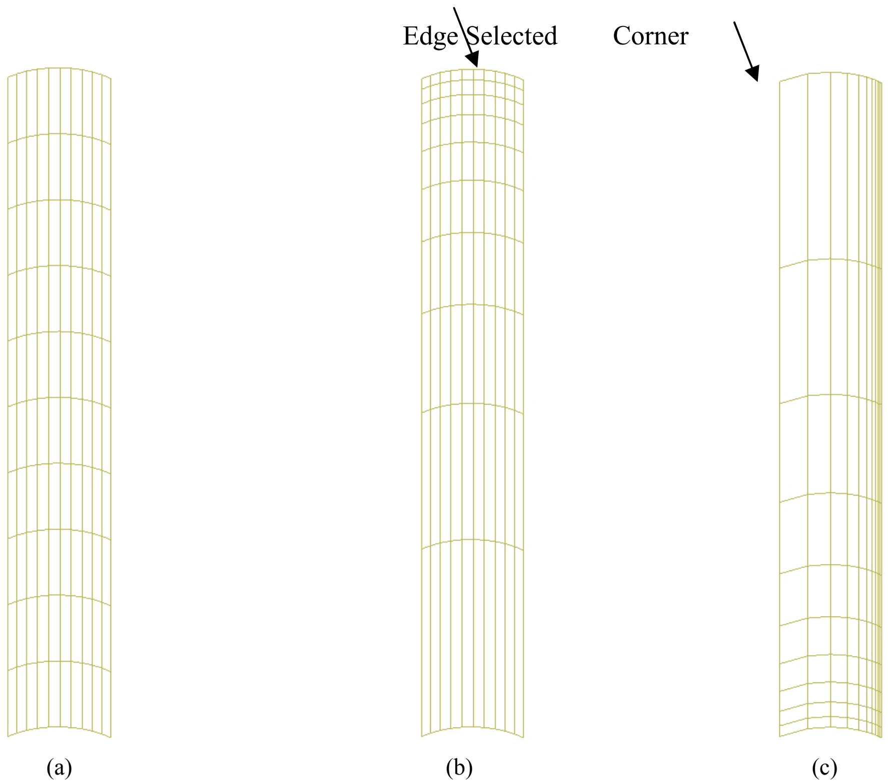


Figure 10.1.2 An example of mesh with bias mesh control: (a) No Bias (b) Edge Bias (c) Corner Bias (Bias Factor=1.765).

10.1.3 CORNER BIAS MESH

The CORNER BIAS MESH control allows the user to enter an element bias factor (0.625 to 1.6) relative to the selected corner of the line data that is to be modeled. This function multiplies the size of each adjacent element that is created from the selected corner by the bias factor.

If the CORNER BIAS MESH function as shown in Figure 10.1.1 is toggled on, the user can define the bias factor by dragging the slider. An example of mesh with edge bias mesh control is shown in Figure 10.1.2c.

Note: A bias factor greater than 1.0 generates proportionately larger elements from the selected 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.

10.1.4 PLATE ELEMENT TYPE

This function enables the user to control the type of mesh generated by eta/DYNAFORM. If the QUADRILATERAL function as shown in Figure 10.1.1 is toggled checked, 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 is checked, the whole plate mesh generated is triangular elements dominant.

10.1.5 LINE MESH METHOD

Please refer to SECTION 6.3.5 for detailed information about LINE MESH METHOD.

10.2 FILE OPTION

This function enables the user to set the FILE DIALOG STYLE (WINDOWS OR UNIX) and set the AUTO BACKUP file saving function. The dialog window is shown in Figure 10.2.1.

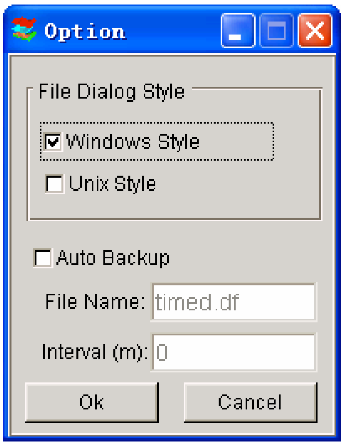


Figure 10.2.1 The Option dialog window.

If WINDOWS STYLE is checked, the file dialog style is shown as in Figure 10.2.2

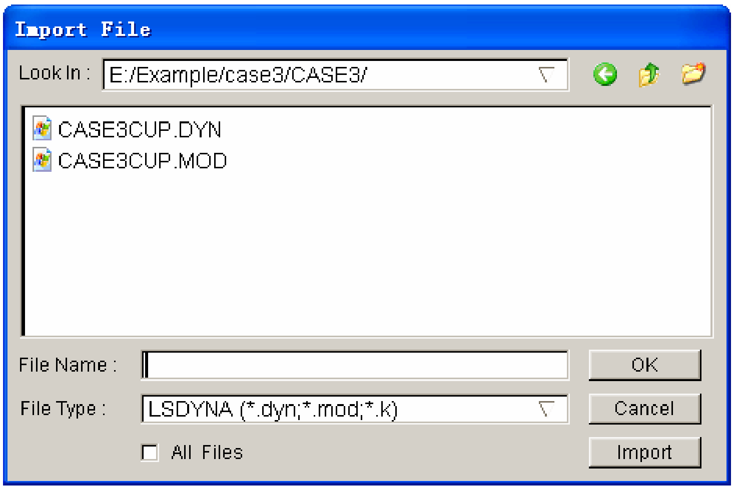


Figure 10.2.2 Windows Dialog Style

If UNIX STYLE is checked, the file dialog style is shown as in Figure 10.2.3.

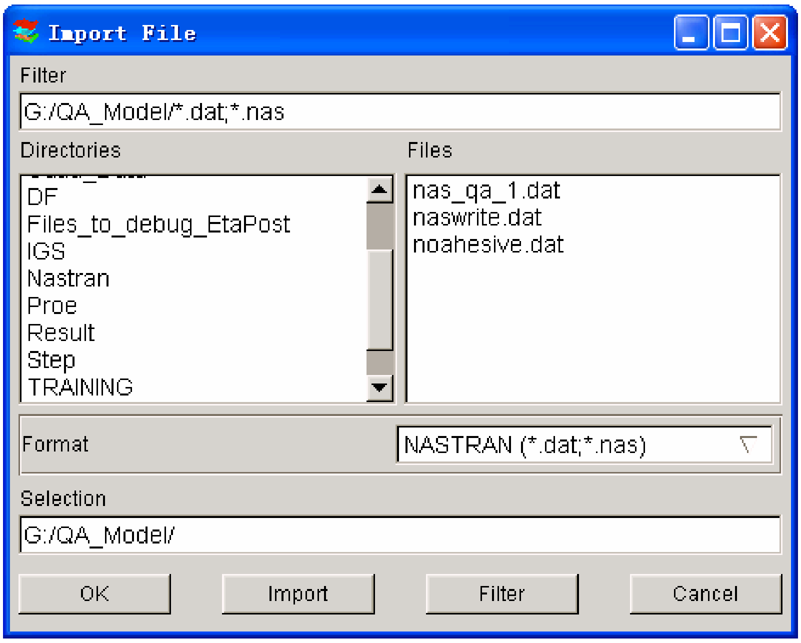


Figure 10.2.3 Unix Dialog Style

10.3 SYSTEM LANGUAGE

This function allows the user to change the language in the menu, prompt and icon tips displayed on the screen. The program will display a dialog window as shown in Figure 10.3.1.

Figure 10.3.1 Select language dialog window

Click the drop down list to select the language. Currently DYNAFORM supports four languages: English, Chinese, Japanese and Korean. More language can be easily customized by a Resource Compiler utility software.

After a language is selected, the program will display a DYNAFORM question window as shown in Figure 10.3.2.

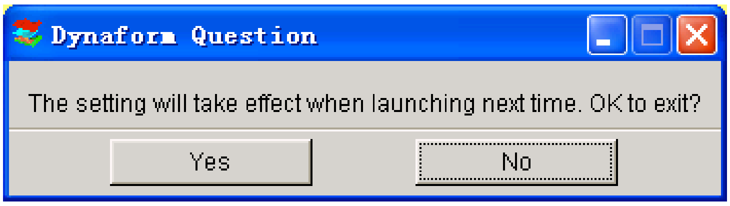


Figure 10.3.1 Select language confirmation window

If **YES** is selected, DYNAFORM will use the selected language as default language in the future DYNAFORM session. **NO** will keep the current language.

Note: The user may also use the System Language setting in the configuration file dynaform.ini to set the default language.

10.4 SHOW ICON TIPS (toggle)

This function allows the user to read/identify the name of each icon on the menu bar and dialog window.

CHAPTER 11
UTILITIES

The functions in the UTILITIES menu make up eta/DYNAFORM's tool kit. Many of these functions are also found in other menus, but the UTILITIES menu (Figure 11.1) provides the user with a convenient way to access these functions.

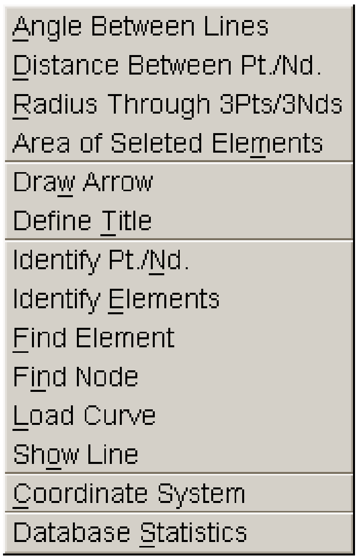


Figure 11.1 The Utilities menu.

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

11.1 ANGLE BETWEEN LINES

This function enables the user to measure the angle (in degrees) between two selected vectors.

After selecting the function, the INPUT COORDINATE window prompts the user to define two vectors by selecting 3 points. eta/DYNAFORM displays the measurement of four angles in the PROMPT MESSAGE window in the following format,

ANGLE=69.90, TOP:72.49, SIDE:3.41, REAR:124.78

11.2 DISTANCE BETWEEN POINTS/NODES

This function enables the user to measure the distance (in mm) between two points, two nodes, or a node and a point. This function is also provided in the PREPROCESS/NODE dialog window.

The user can select two points/nodes by using options in the INPUT COORDINATE dialog window. The distance and the differences of the three components are given in the PROMPT MESSAGE window in the following format:

D=180.000, DX=180.000, DY=0.000, DX=0.000

11.3 RADIUS THROUGH 3PTS/3NDS

This function is used to measure the radius (in mm) using 3 points or nodes.

The user can select three points/nodes by using options in the INPUT COORDINATE dialog window. The result will be displayed in the PROMPT MESSAGE window in the following format:

R=20.000 CENTER AT, X=-200.000, Y=0.000, Z=0.000

11.4 AREA OF SELECTED ELEMENTS

This function is used to calculate the area of the selected elements. The area and the mass center are printed in the PROMPT MESSAGE window in the following format:

AREA:1000.0000, MASS CENTER:1000.00000, 0.00000, 0.00000

11.5 DRAW ARROW

This function allows the user to draw arrows about a specific image on the display screen.

The user selects two locations in the screen following the message in the PROMPT window. An arrow will be drawn from the first location to the second location.
Note: This function is used in conjunction with DEFINE TITLE. The arrow and the title will be removed using the CLEAR HIGHLIGHT icon.

11.6 DEFINE TITLE

This function allows the user to enter a title or text at any location of the displayed screen. The dialog window is shown in Figure 11.6.

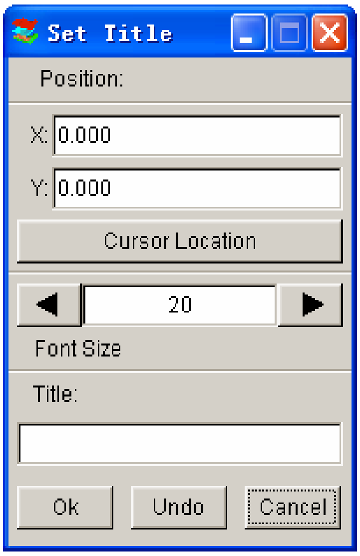


Figure 11.6 The Define Title dialog window.

- Type in the title in the field.
- Click the arrow buttons to set the font size of the title.
- Click OK to display the title on the display area.

The user can define the position of the title by entering the screen coordinates of the X, Y values or by selecting CURSOR LOCATION.

11.7 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 window.

11.8 IDENTIFY ELEMENT

This function allows the user to identify an element identification number and its associated nodes' identification number. It is also found in the PREPROCESS/ELEMENT dialog window.

11.9 FIND ELEMENT

This function allows the user to find or identify an element by keying in its element identification number. This function is also found in the PREPROCESS/ELEMENT dialog window.

11.10 FIND NODE

This function allows the user to find the X, Y, Z coordinates of the specified node by keying in the node identification number. This function is also found in the PREPROCESS/NODE dialog window.

11.11 LOAD CURVE

The options in this menu are utilized to generate or modify the load curves. The options are shown in Figure 11.11.1.

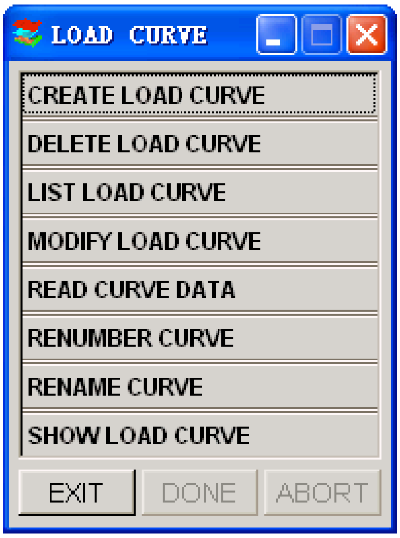


Figure 11.11.1 Load Curve Options

11.11.1 CREATE CURVE

This function allows the user to manually create a load curve. As shown in Figure 11.11.2, the user can type in both the curve number and name in the CREATE CURVE dialog window. A warning message will appear if the curve number is being used by another load curve or there is only one point listed in the curve. The user must then click on the ADD POINT button and key in the curve data manually.

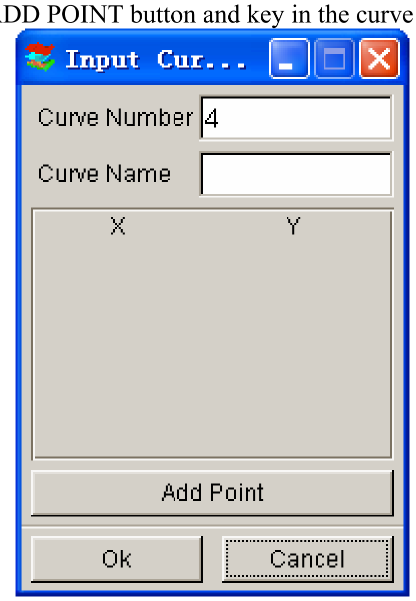


Figure 11.11.2 The Create Curve dialog window

11.11.2 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 window as shown in Figure 11.11.3. 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 and then clicking on the OK button to confirm the selection.

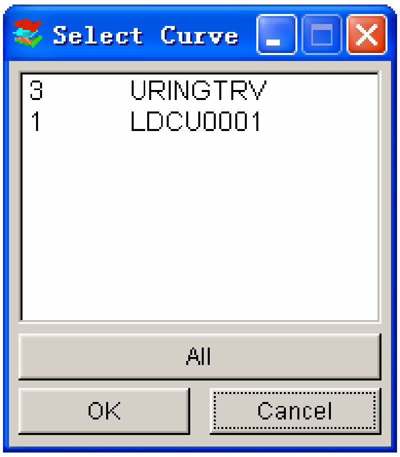


Figure 11.1.3 The Select Curve dialog window.

- A DYNAFORM QUESTION window, as shown in Figure 11.11.4, appears.

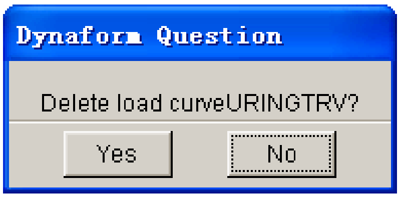


Figure 11.11.4 A DYNAFORM Question window.

- Select YES to delete the selected curve(s).
- Select NO to reject the choice.

11.11.3 LIST LOAD CURVE

This function allows user to list the existing load curves in the eta/DYNAFORM database.

The user can select one or multiple load curves from the LIST CURVE dialog window as shown in Figure 11.11.5.

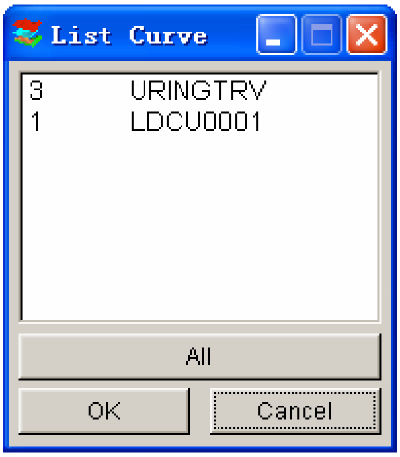


Figure 11.11.5 List Curve

11.11.4 MODIFY LOAD CURVE

This function allows the user to modify data points of a selected load curve.

After selecting a curve in the SELECT CURVE dialog window, the SELECT OPTION dialog window, Figure 11.11.6, is displayed.

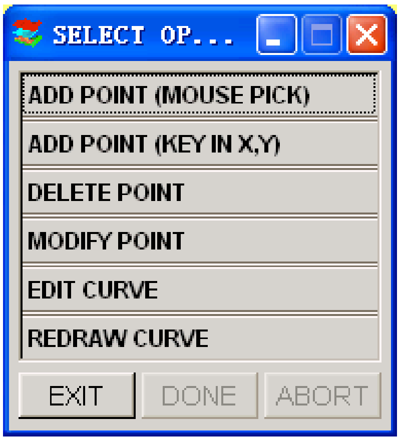


Figure 11.11.6 The Modify Curve dialog window.

1. **ADD POINT (MOUSE PICK)**

   Allows the user to select any location from the graph. The selected point will be included in the curve.

2. **ADD POINT (KEY IN X, Y)**

   Allows the user to add curve data points by keying in the X, Y values.

3. **DELETE POINT**

   Deletes selected data points automatically.

4. **MODIFY POINT**

   There are several options to modify data points as shown in Figure 11.11.7.

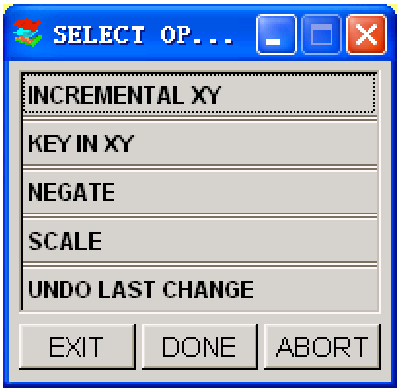


Figure 11.11.7 Modify Point Options

- **INCREMENTAL XY**
  Enter the X, Y increments, and select OK in the next dialog window. Select point(s) to move in the curve screen.
- **KEY IN XY**
  Select a point in the curve screen. The X, Y values will be shown in the next dialog window. Change the X, Y values, and select OK. The points on the curve will be 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.

5. **EDIT CURVE**

   This function enables the user to edit the.X and Y value of the curve.

6. **REDRAW CURVE**

   This function redraws the modified curve on the display area.

11.11.5 READ CURVE DATA

This function enables the user to import an existing curve's data into the eta/DYNAFORM database.

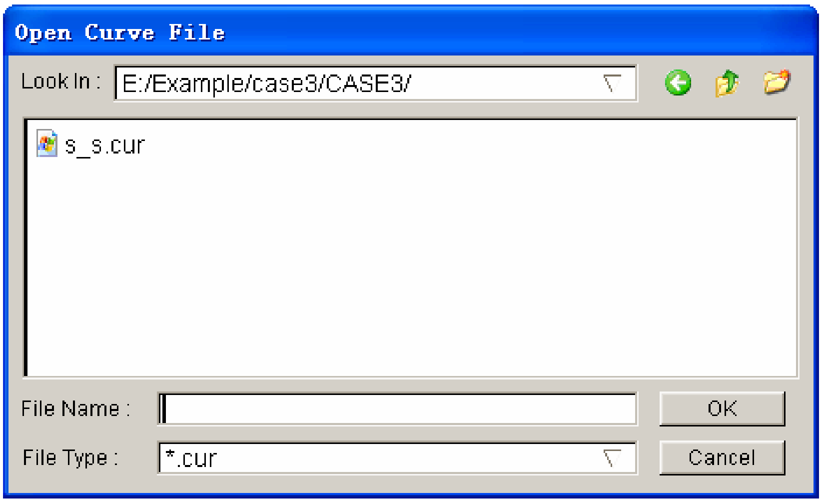


Figure 11.11.8 The Open Curve File window.

Note: There are three different curve formats supported by eta/DYNAFORM; i.e, all three curve formats can be imported into 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
  -2.744370E-01   6.76406E-01
  -2.357220E-01   6.37085E-01
  -1.118451E-01   6.00334E-01
  -1.625111E-01   5.66503E-01
  -1.278330E-01   5.35114E-01
  -11.43107E-02   5.01100E-01
  -6.187540E-02   4.86003E-01
  -3.045112E-02   4.67236E-01
   0.000000E+00   4.52115E-01
   2.115588E-02   4.71172E-01
   5.826811E-02   4.86212E-01
   8.617770E-02   4.11866E-01
   1.133211E-01   5.08111E-01
   1.311762E-01   5.17585E-01
   1.655140E-01   5.24738E-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

```
*KEYWORD (must)
*DEFINE_CURVE
$CURVENAME      ABC
$     LCID       SIDR      SCLA       SCLO      OFFA      OFFO
       200          0
$                  A1                   O1
      .000000000E+00     .0000000000E+00
      .476201171E+00     .7701854710E+02
      .115241115E+00     .1223511673E+03
      .142862114E+01     .1411052185E+03
      .111048311E+01     .1647661511E+03
      .238104811E+01     .1740170440E+03
      .285725880E+01     .1711463074E+03
      .333346844E+01     .1826611181E+03
      .380116783E+01     .1845566250E+03
      .428588820E+01     .1856677700E+03
      .476201178E+01     .1863218111E+03
*END(The *END is optional here)
```

The dialog window is shown in Figure 10.11.8. eta/DYNAFORM lists all the curve data files with the suffix **.cur** in the FILES field. The user may also read other curve data files by entering the file names.

11.11.6 RENUMBER LOAD CURVE

This function enables the user to renumber the ID of a selected curve.

- Select a curve from the SELECT CURVE dialog window as shown in Figure 11.11.9

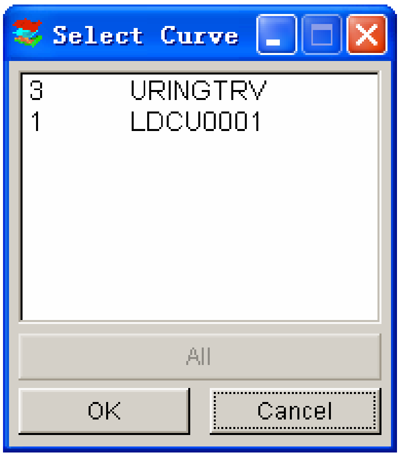


Figure 11.11.9 Select Curve for Renumber

- Enter the new number in the next dialog window.

11.11.7 RENAME CURVE

This function enables the user to rename the load curves individually.

- Select a load curve from the list in the SELECT CURVE dialog window.
- Enter the new name in the next dialog window.

11.11.8 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 window as shown in Figure 11.11.10.

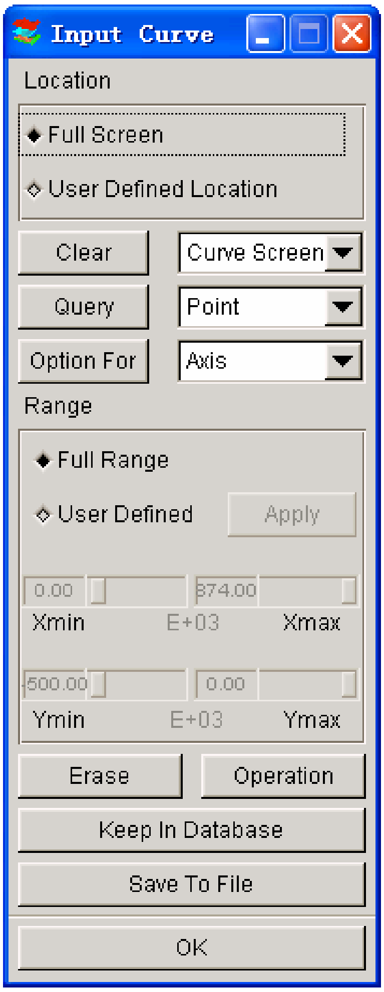


Figure 11.11.10 The Show Curve Options dialog window.

Select the load curve from the SELECT CURVE dialog window as shown in Figure 11.11.3.

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 on the display area.

- **MODEL**
  Clear the displayed model.

- **ETA LABEL**
  Clear the label at the left bottom of the display screen.

- **ALL**
  Clear all objects in the display screen.

Note: Click on CLEAR button to remove the 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 will be given in the 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 will be given in the PROMPT window.

- **GRAPH**

The name of the graph and the number of curves in the graph will be given in the PROMPT window.

OPTION FOR GRAPH DISPLAY

This allows the user to manipulate the display settings of AXIS, GRAPH, CURVE, GRID and LEGEND. See Figures 11.11.11 ~ 11.11.15.

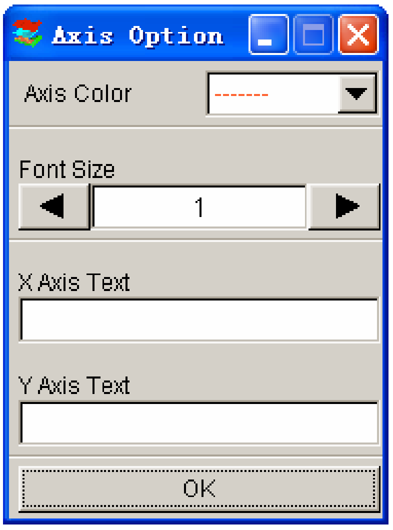


Figure 11.11.11 Axis Options

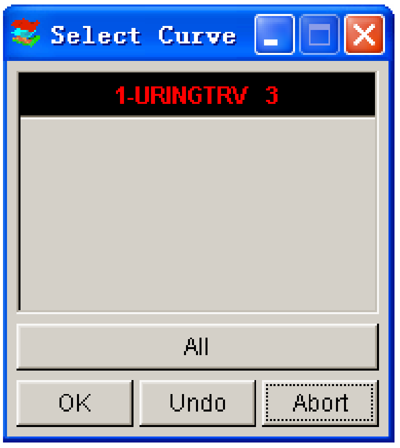


Figure 11.11.12 Select Curve

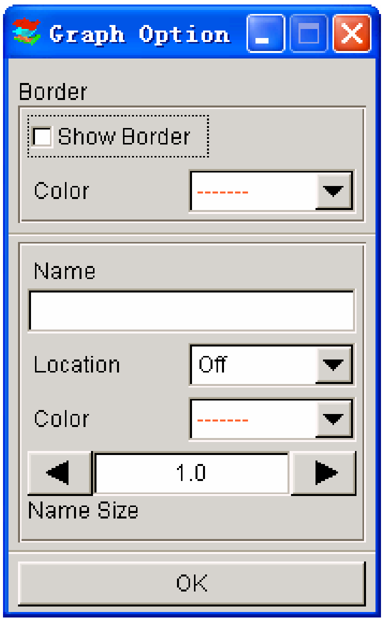


Figure 11.11.13 Graph Options

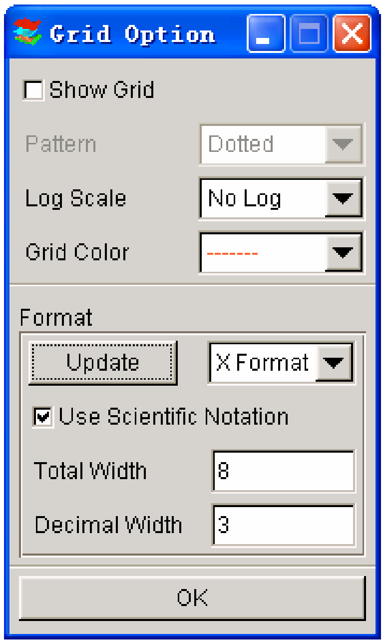


Figure 11.11.14 Grid Options

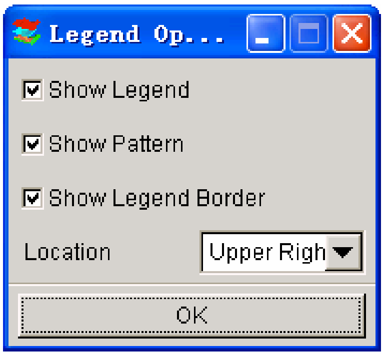


Figure 11.11.15 Legend Options

Select the button to the right of OPTION FOR to select between AXIS, CURVE, GRAPH, GRID and LEGEND. Each option displays its own dialog window.

DEFINE RANGE OF GRAPH

This function zooms in on a specific section of the graph.

- Toggle on FULL RANGE to show a curve in full range (default).

- Toggle on USER DEFINED to adjust the values of four sliders and then select APPLY to reset the range.

OPERATION

This function allows the user to apply mathematical operations to the displayed curve. The options are shown in Figure 11.11.16.

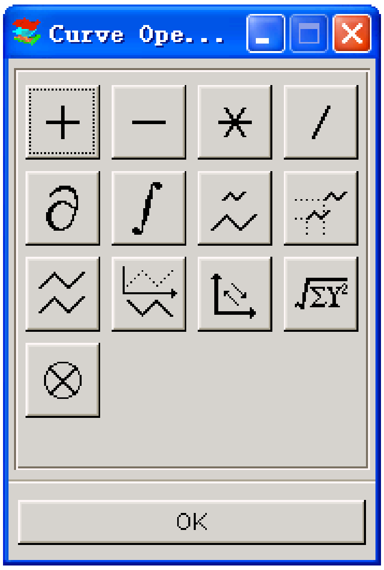


Figure 11.11.16 Curve Operation

After an operation is selected, the SELECT CURVE dialog window will appear for selecting curves for the operation as shown in Figure 11.11.17. 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 prompts:

LEFT BUTTON SELECT, MIDDLE BUTTON DESELECT

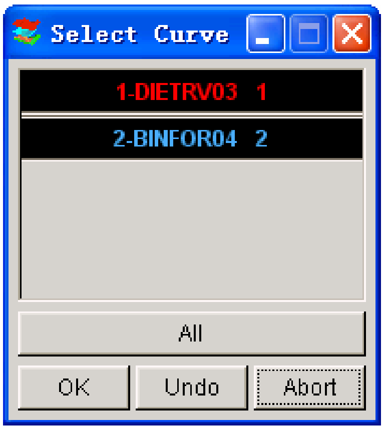
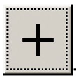
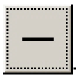
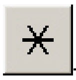
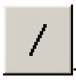


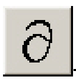
Figure 11.11.17 Select Curve

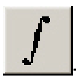
-- Creates a new curve in the curve screen of which the values of Y are the sums of the selected curves. At least two curves must be selected.

-- Creates a new curve of which the values are the differences of two selected curves.

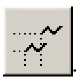
-- Creates a new curve of which the values are the product of two curves.

-- Creates a new curve of which the values are quotients of two selected curves.

-- Creates a new curve that is the derivative of the selected curve with respect to time.

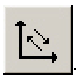
-- Creates a new curve that is the indefinite integral of the selected curve with respect to time.

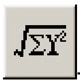
-- Scales the selected curve using magnification factors in the X, Y directions.

-- Moves the selected curve in the X, Y directions.

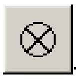
 -- Creates a new curve that is a copy of the selected curve at the same position and in different colors.

 -- Negates the values of the selected curve in the Y direction.

 -- Switches the X-axis and the Y-axis.



-- This function creates a new curve that is the resultant of the selected curve.

 -- Creates 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 loaded curves. The user needs to save the curve.**

ERASE
This function removes the selected curve from the displayed graph

KEEP IN DATABASE
This function saves the selected curve in the database with a name that begins with CUR.

SAVE TO FILE
This function saves the display graph as a text file in a directory. The SAVE CURVE window is shown in Figure 11.11.16

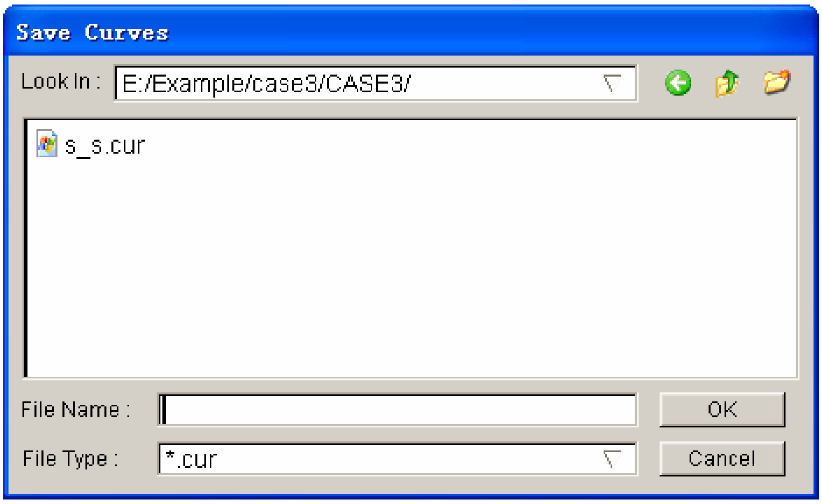


Figure 11.11.16 The Save Curves window.

11.12 SHOW LINE

This function is used to identify any existing line and its direction. See Section 6.1, PREPROCESS/LINE/POINT.

11.13 COORDINATE SYSTEM

The functions in this menu are used to create and modify local coordinate systems. The functions are shown in Figure 11.13.1.

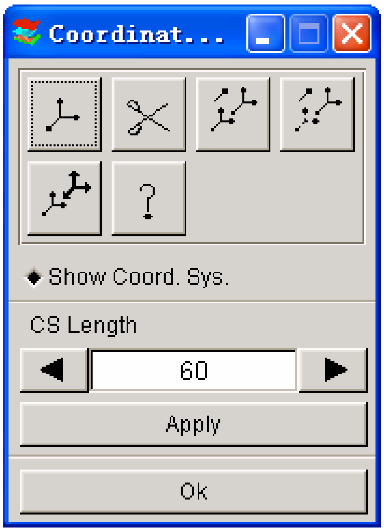


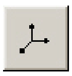
Figure 11.13.1 Coordinate System Options

- Click the horizontal scroll bar to changes the size of the displayed coordinate system. Select APPLY when finished.

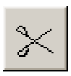
- **COORD. SYS. ON**

  Toggle all created coordinate systems ON/OFF.

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

11.13.1 CREATE COORDINATE SYSTEM 

This function is used to create a coordinate system. Please refer to SECTION 2.5, LOCAL COORDINATE SYSTEM, to get detailed information.

11.13.2 DELETE COORDINATE SYSTEM 

This function enables the user to delete a local coordinate system from the database.

- The SELECT LCS dialog window appears. See Figure 11.13.2.

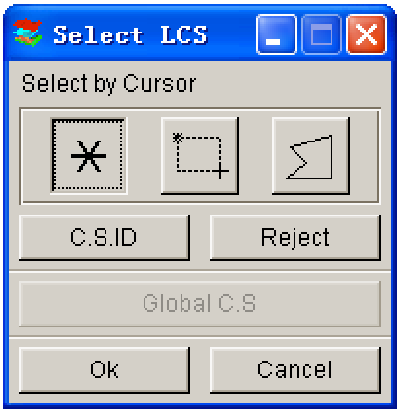
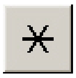
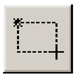
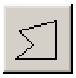


Figure 11.13.2 Select LCS

 -- Picks a coordinate system.

 -- Drags a window to include the coordinate system.

 - -Defines a multi-point region to include the coordinate system.

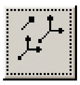
C.S. ID

Enter the starting and ending coordinate system ID and the increment in the next dialog window.

REJECT

Reject the last selection.

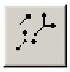
Note: To remove the local systems/definitions from individual nodes, the user must reassign the nodes to the GLOBAL COORDINATE SYSTEM.

11.13.3 COPY COORDINATE SYSTEM 

This function enables the user to copy and create a selected coordinate system in a new location.

- The SELECT LCS dialog window provides three options: 1) Pick the coordinate system displayed on the screen using the default mouse pick, 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 INPUT COORDINATE dialog window. The selected coordinate system will be copied automatically.

11.13.4 MODIFY COORDINATE SYSTEM 

This function is used to modify the definition of an existing coordinate system.

- After a coordinate system is selected in the SELECT LCS dialog window, the SELECT OPTION window as shown in Figure 11.13.3 is displayed.

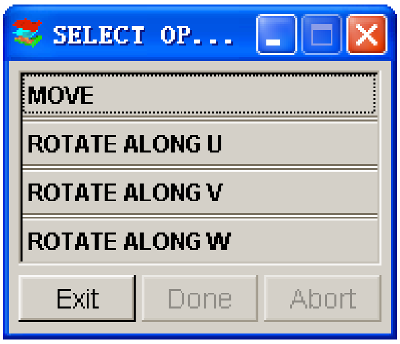
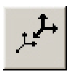


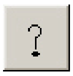
Figure 11.13.3 Modify C.S Options

- Select the MOVE function, and the INPUT COORDINATE dialog window will appear and prompt the user to define the new location of the coordinate system. The coordinate system will be moved automatically after the new location is defined.
- Select the ROTATE function, and the INPUT COORDINATE dialog window will appear and prompt the user to define the new location for the coordinate system. The coordinate system will be rotated automatically after the new location is defined.

11.13.5 CURRENT COORDINATE SYSTEM 

This function allows the user to assign a CURRENT status to a selected coordinate system.

- Select a coordinate system by cursor; the selected coordinate system will be highlighted and the last one will change its color.

11.13.6 IDENTIFY COORDINATE SYSTEM 

This function is used to identify the type and origin of a coordinate system.

- Select a coordinate system by cursor; and eta/DYNAFORM will prompt:

  C.S. #### [SYSTEM TYPE] ORIGIN (x y z)

  Where:
  #### is the coordinate system numbers.
  SYSTEM TYPE is the system type (rect., sphere, cylin.), and
  ORIGIN is the global location of the system origin.

11.14 DATABASE STATISTIC

This function enables the user to view the statistics of the eta/DYNAFORM database regarding geometry, models, materials and interfaces.

CHAPTER 12

VIEW OPTIONS

The functions provided in this menu adjust the display of the items shown in the display area of eta/DYNAFORM. The options are shown in Figure 12.1.

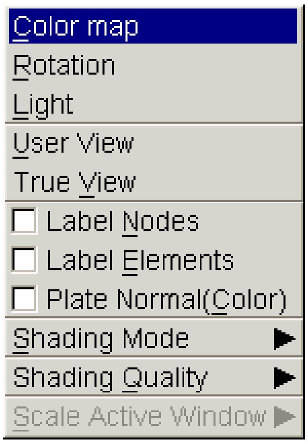


Figure 12.1 The View Options menu.

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

12.1 COLOR MAP

This function is used to change the default color in the color bar. See Figure12.1.1.

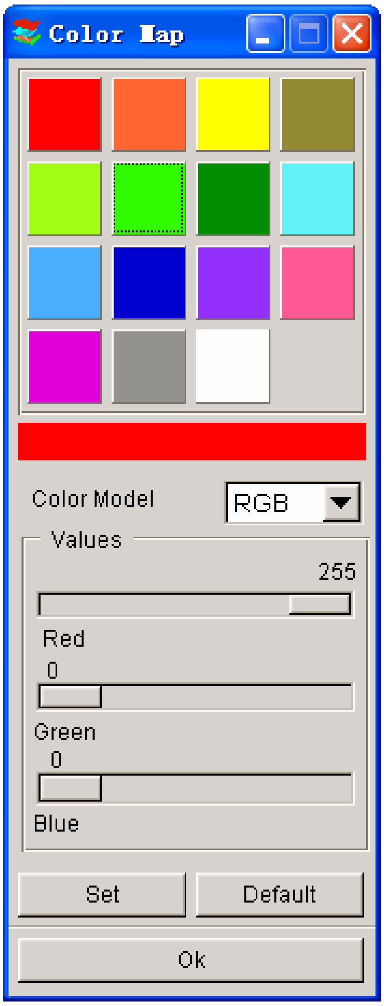


Figure 12.1.1 The Color Map dialog window

- 15 colors are available from the color palette. The selected color will be shown in the long rectangular bar.
- The Color Model provides two options, either RGB (Red, Green, and Blue) or HSV (Hue, Saturation, and Luminosity).

- The sliders enable the user to change the values of RGB and HSV.
- SET confirms the change of values of the color model.
- DEFAULT resets the values of color model.

12.2 ROTATION

This function enables the user to rotate an object in the display area about the global (or virtual) X, Y, and Z-axes. The options are shown in Figure 12.2.1.

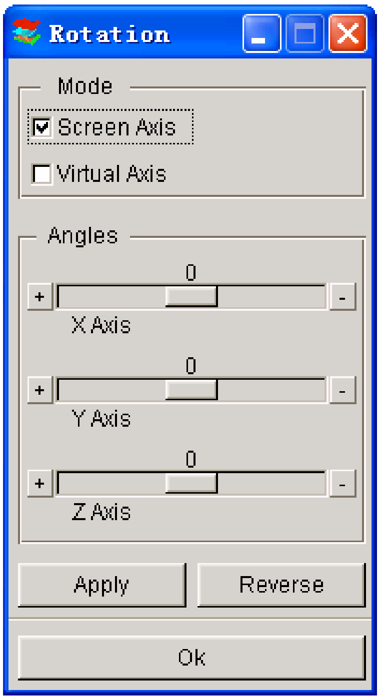


Figure 12.2.1 The Rotation dialog window.

- **SCREEN AXIS**
  Rotates an object in the display area according to an angle. The screen axes are fixed at all times and are defined as follows:

  1. SCREEN X
     Axis direction is from the left to right of the terminal screen.
  2. SCREEN Y
     Axis direction is from the bottom to the top of the terminal screen.

3. SCREEN Z
   Axis direction is from the screen to the user.

- **VIRTUAL AXIS**
  Rotates an object about global (or virtual) X, Y, and Z-axes according to an angle.

- The sliders enable the user to adjust rotation angles.

- APPLY completes the rotation. This function can be repeated as many times as desired.

- REVERSE reverses the rotation. This function can be repeated as many times as desired.

12.3 LIGHT

This function enables the user to move a directional light source uniformly with a consistent intensity. The dialog window is shown in Figure 12.3.1. The right and left arrow buttons move the light source in the positive and negative X-direction, and the up and down arrow buttons move the light source in the positive and negative Y-direction.

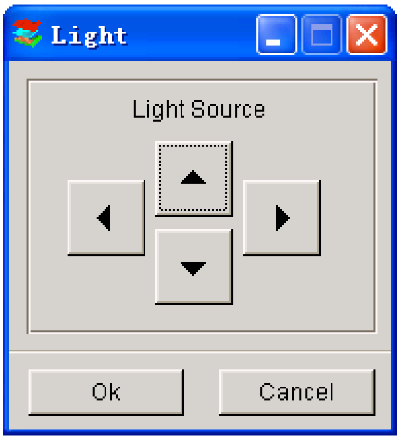


Figure 12.3.1 The Light Source dialog window.

12.4 USER VIEW

This function saves or stores a desired view in the current eta/DYNAFORM database and recalls any previously saved view. See Figure 12.4.1.

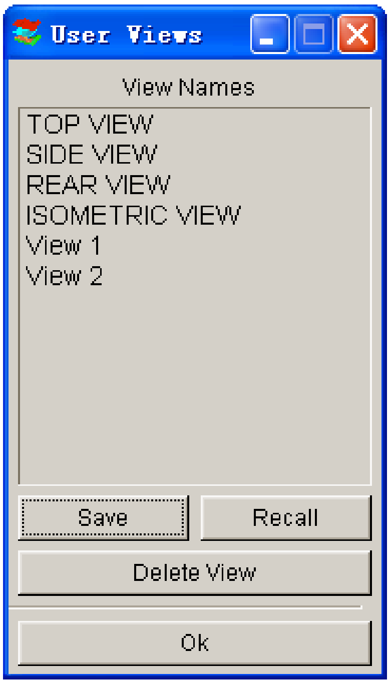


Figure 12.4.1 The User Views dialog window.

- **Save**
  Saves the current displayed view in the eta/DYNAFORM database. The saved view will be listed in the window. eta/DYNAFORM allows the user to save up to six user views plus the four pre-defined views (a total of 12 views).

- **Recall**
  Displays the selected view from the list.

- **Delete View**
  Deletes the selected view from the list.

12.5 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 window appears to define a local coordinate system.

- The DYNAFORM QUESTION dialog window appears as shown in Figure 12.5.1.

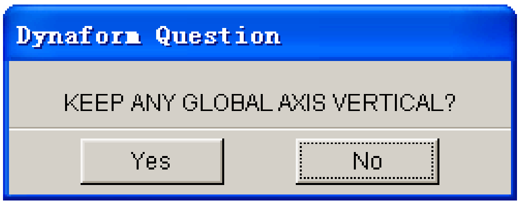


Figure 12.5.1 A eta/DYNAFORM Question dialog window

1. YES
   Allows 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
   Displays the object in its true view (perpendicular to the W-axis).

12.6 LABEL NODES (toggle)

This function enables the user to toggle the node labels on/off. The node number of all displayed nodes will be shown. This function is also found under Preprocess/Node**.**

12.7 LABEL ELEMENTS (toggle)

This function enables the user to toggle the element labels on/off. The element number of all displayed elements will be shown. This function is also found under Preprocess/Element**.**

12.8 PLATE NORMAL (Color)

This function enables the user to display the normal direction of element with color.

12.9 SHADING MODE

As shown in Figure 12.9.1, Shading Mode offers three options for shading a part or a model. The Gourand Shading mode, which is smooth shading, is the default setting for eta/DYNAFORM and its normal is based on nodes in algorithm. The Flat Shading is based on elements.

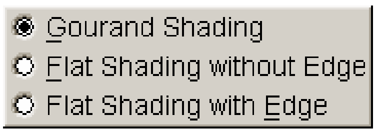


Figure 12.9.1 The submenu of Shade Mode.

12.10 SHADING QUALITY

As shown in Figure 12.10.1, Shading Quality offers three options for shading a part or a model. The High mode have the best shading quality but it will occur more CPU time. The Normal is the default setting for shading. The Low mode have an inferior shading quality but is the quickest mode when shading. User can adjust this parameter according the performance of the Workstation or the need of output high effect picture.

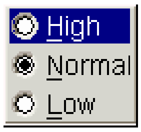


Figure 12.10.1 The submenu of Shade Quality.

12.11 SCALE ACTIVE WINDOW

This function enables the user to scale an active window display by the scale factor provided from the pull-down menu. This function is active only the active window function is performed.

CHAPTER 13

ANALYSIS

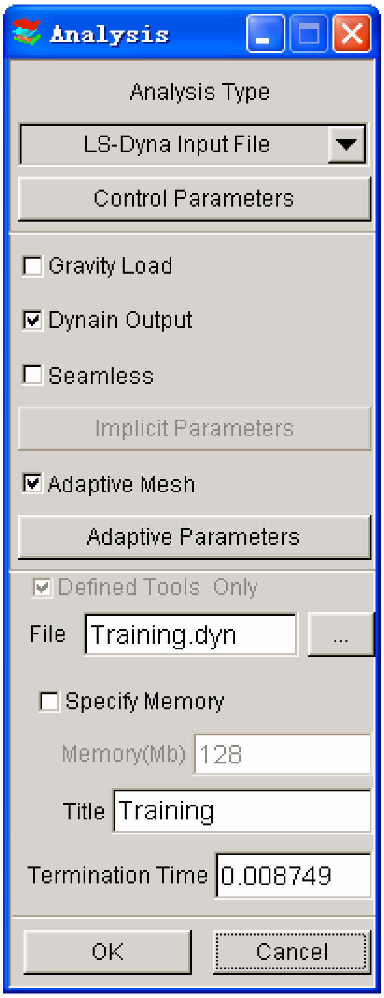
The functions provided in the ANALYSIS menu allow the user to submit an analysis or to generate an output file as shown in Figure 13.1.

Figure 13.1 Analysis

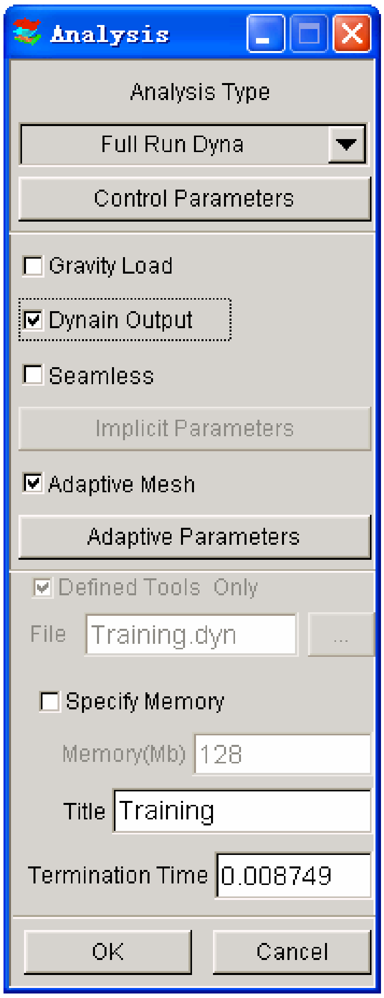
A detailed description of each function in the menu is given in the following sections.

13.1 LS-DYNA (CTRL+A)

If the draw type is not set as Spring Back or Gravity only (Tools/Analysis Setup/Draw type), the ANALYSIS dialog window is shown as in Figure 13.1.1. The **LS-DYNA** menu allows the user to set the parameters for running a stamping simulation, gravity loading and springback analysis. The parameters will be written into the analysis output. The default dialog window of this menu (for stamping simulation) is shown in Figure 13.1.1.



(a)



(b)

Figure 13.1.1 Analysis dialog window (a) LS-DYNA Input (b) Full Run DYNA

13.1.1 ANALYSIS TYPE

There are two types of analysis output:

1. LS-DYNA INPUT FILE
   This function outputs analysis results as an LS-DYNA input deck.

2. FULL RUN DYNA
   This function outputs an LS-DYNA input deck and submits it to LS-DYNA, which will be running in the background.

The user can select the type of analysis output by clicking on the toggle menu.

CONTROL PARAMETERS

This function allows the user to define the values of control parameters for running a stamping simulation using LS-DYNA. The user can edit the parameters by clicking on the CONTROL PARAMETERS icon from the ANALYSIS dialog window. The DYNA3D CONTROL PARAMETER dialog window is shown in Figure 13.1.2.

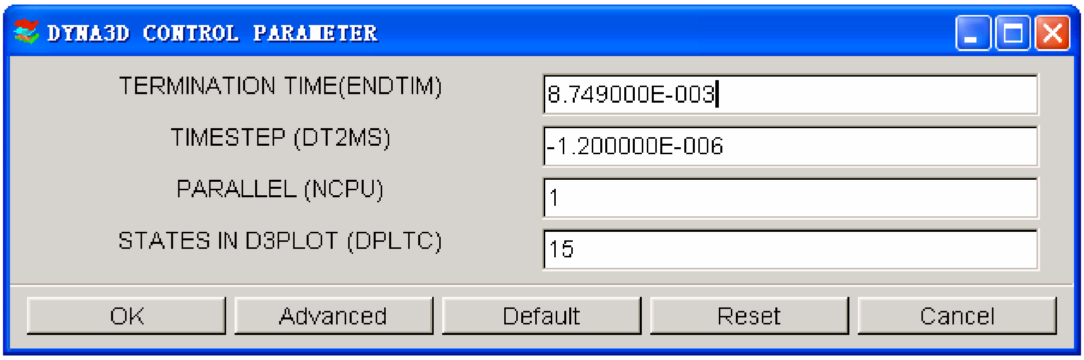


Figure 13.1.2 Control Parameters dialog window

OK - Accepts the control parameters

DEFAULT - Resets all values to their default settings

RESET - Resets the last defined setting to its previously defined value

ADVANCED - Extends the table to make the advanced control parameters show up

CANCEL - Aborts the table and resets all values to their previous setting

Note: Refer to the *LS-DYNA User's Manual* for detailed information about each control parameter.

13.1.2 GRAVITY LOAD (toggle)

This function toggles gravity force ON/OFF and writes out the relative control cards to the input deck.

13.1.3 DYNA OUTPUT (DYNAIN) (toggle)

This function automatically writes out the INTERFACE_SPRINGBACK_DYNA3D card to the input deck. Therefore, LS-DYNA will create a DYNAIN file at the end of the simulation run. The DYNAIN file contains the deformed blank results, including thickness, stress and strain.

13.1.4 SPRINGBACK (SEAMLESS)

This function toggles on/off the LS-DYNA seamless explicit/implicit switch. It will activate implicit springback analysis after the stamping simulation. If SEAMLESS is toggled on, user can set the springback constraint and define the IMPLICIT parameters as shown in Figure 13.1.3.

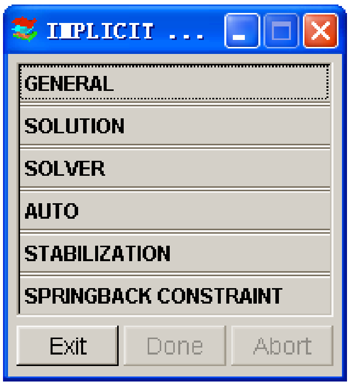


Figure 13.1.3 Implicit Parameters

***Note: Refer to the LS-DYNA User's Manual, keyword *CONTROL_IMPLICIT_OPTIONAL,* for more information.**

13.1.5 ADAPTIVE MESH (toggle)

If this function is toggled on, ADAPTIVE scheme is activated during the stamping simulation. The user can edit the adaptive parameters by clicking on the ADAPTIVE PARAMETERS button. The default ADAPTIVE CONTROL PARAMETERS dialog window is shown in Figure 13.1.4. The ADVANCE button enables the user to expand the dialog window for more adaptive parameters.

Note: Refer to the *LS-DYNA User's Manual*, keyword **CONTROL_ADAPTIVE*, for detailed information about each parameter.

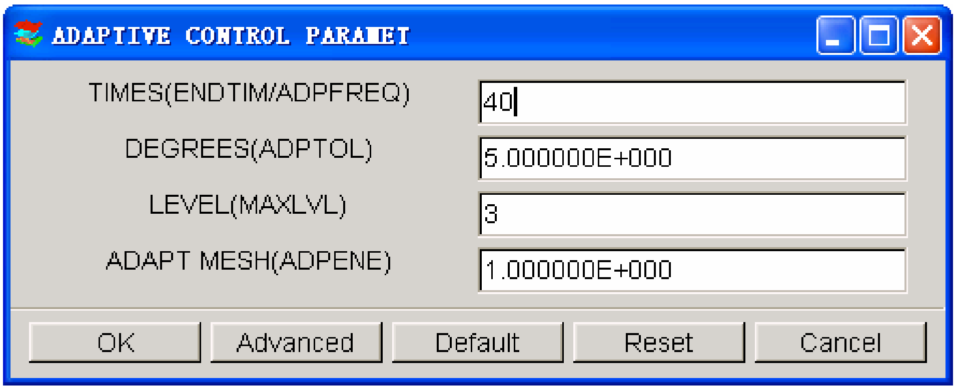


Figure 13.1.4 The Adaptive Control Parameters dialog window

13.1.6 DEFINED TOOLS ONLY (toggle)

If toggled ON, only parts defined as tools will be output to the input deck. If toggled OFF, all parts in the eta/DYNAFORM database will be output.

13.1.7 SPECIFY MEMORY (toggle)

If toggled ON, the user can allocate memory (in Mb) for running LS-DYNA. This function will automatically convert the input memory (in Mb) to memory (in words) and write it out to the *KEYWORD card in the input deck.

- The FILE NAME must be given. If the file name is not known, the button alongside the FILE NAME FIELD displays a selection dialog window from which the user can select the file name.
- The TITLE must be given. TERMINATE TIME is given automatically by the program according to the motion curve.
- After the above requirements are defined, select OK to start the process.

13.2 LS-DYNA (SPRINGBACK)

If the draw type is set as Spring Back (Tools/Analysis Setup/Draw type), the ANALYSIS dialog window is shown as in Figure 13.2.1.

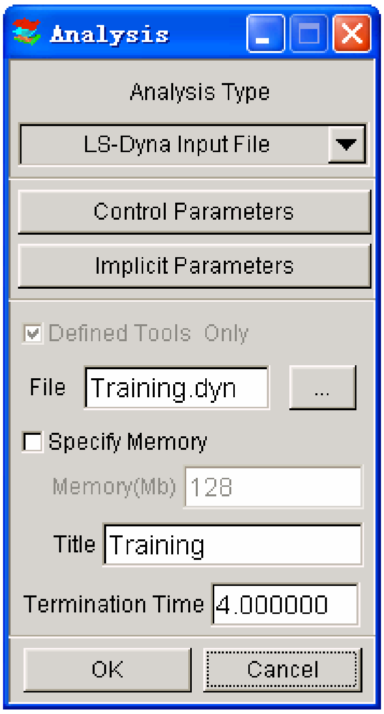


Figure 13.2.1 The Analysis dialog window for Springback analysis

The functions are discussed in SECTION 13.1, LS-DYNA.

Note: Refer to the *LS-DYNA User's Manual* for more information.

13.3 LS-DYNA (GRAVITY LOADING)

If the draw type is set as Gravity (Tools/Analysis Setup/Draw type), the ANALYSIS window is shown in Figure 13.3.1.

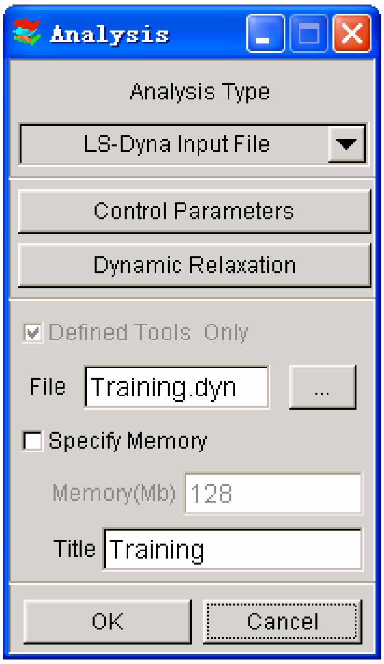


Figure 13.3.1 The Analysis dialog window for Gravity Loading

Note: Refer to the *LS-DYNA User's Manual*, keyword **CONTROL_DYNAMIC_RELAXATION*, for detailed information about each parameter.

13.4 MSTEP

The MSTEP is a new Modified-One-Step solver adopted by eta/DYNAFORM which is mainly used to quickly gain the formability access and the blank outline in the early stage of automobile design cycle.

Please refer to the section 7.2 for detailed information about the MSTEP.

13.5 OUTPUT NEW DYNAIN FILE

This function allows the user to output a new DYNAIN file (for example, after trimming).

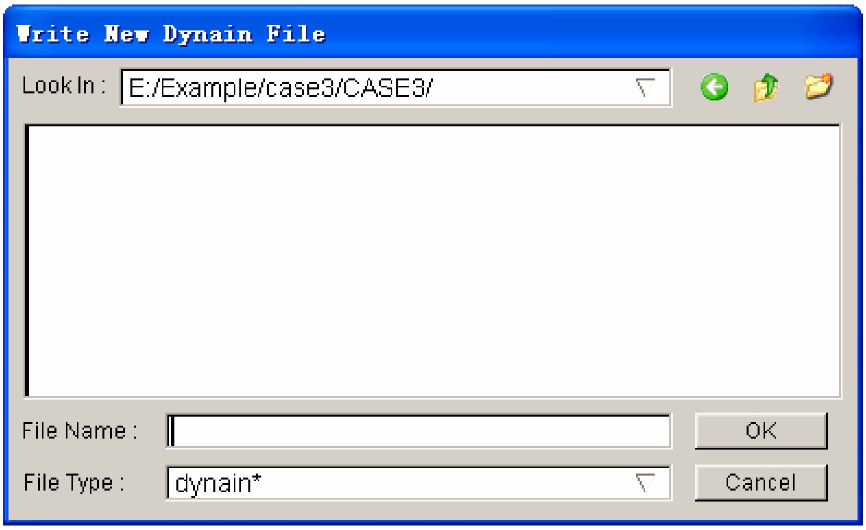


Figure 13.5.1 Write out DYNAIN File

APPENDIX A

HARDWARE AND SOFTWARE REQUIREMENTS

UNIX

PLATFORM	OS VERSION	GRAPHICS CARD	DISK SPACE MB
HP	HP-UX 11+	Minimum CRX 8 Plane	200
IBM	AIX 4.2+	Minimum 24 Plane Graphics	200
SGI	IRIX 6.5+	All Graphics Boards Supported	200
SUN	SunOS 5.8+	Minimum Creator 3D	200

LINUX

The RedHat operating system version 7.3 and above is supported. Dynaform must run under KDE environment. NVIDIA graphic cards are recommended.

PC/WINDOWS

DYNAFORM-PC is compatible with LS-DYNA/PC 970. It will run in Windows 98, 2000, and XP environments. It is not recommended for use with earlier versions of Windows. The following are minimum requirements for proper operation of DYNAFORM-PC in a Windows environment:

Minimum Graphics Requirement: XGA(1024 x 768)

Graphics Card: OpenGL based (Nvidia chip set recommended)

Minimum Memory Requirement:
- Small models (10,000 - 20,000 elements): 256 megabytes RAM
- Medium models (20,000 - 50,000 elements): 384 megabytes RAM
- Large models (50,000 - 100,000 elements): 512 megabytes RAM
- Huge models (100,000 - 200,000 elements): 768+ megabytes RAM

Minimum Loadspace requirement: 256 megabytes
Recommended processor: Pentium 4

APPENDIX B

SUPPORTED IGES ENTITY TYPES

NAME	TYPE
Null Entity	0
Circular Arc Entity	100
Composite Curve Entity	102
Conic Arc Entity	104
Copious Data Entity	106
Plane Entity	108
Line Entity	110
Parametric Spline Curve Entity	112
Parametric Spline Surface Entity	114
Point Entity	116
Ruled Surface Entity	118
Surface of Revolution Entity	120
Tabulated Cylinder Entity	122
Transformation Matrix Entity	124
Rational B-Spline Curve Entity	126
Rational B-Spline Surface Entity	128
Offset Surface Entity	140
Boundary Entity	141
Curve on a Parametric Surface Entity	142
Bounded Surface Entity	143
Trimmed (Parametric) Surface Entity	144
Subfigure Definition Entity	308
Associativity Instance Entity	402
Property Entity	406
Singular Subfigure Instance Entity	408

APPENDIX C

CURVE FILE FORMAT EXAMPLE

LIST DIRECTED FORMAT

```
RECORD 1     :N
RECORD 2     :N1 'TITLE'
RECORD 3     :XDATA1    YDATA1
         .
         .
         .
RECORD 2 + N1       :NDATAN1    YDATAN1
RECORD 3 + N1       :N2 'TITLE FOR CURVE2'
RECORD 4 + N1       :XDATA1       YDATA1
         .
         .
         .
etc.
```

```
Where:

N          = Number of curves, integer
N1         = Number of points on the first curve, integer
N2         = Number of points on the second curve, integer
 TITLE     = Title with a maximum of 20 characters, Starts and
 ends
with a quote ('). Separate with a space between N1 or N2.

XDATA1 = X value of the first part on the curve, real number

YDATA1 = Y value of the first part on the curve, real number.
Separate with a space between XDATA1.


etc.
```

FINAL NOTES

We at ETA would like to thank all those who helped in creating this manual. We have all tried to make this manual as accurate as possible. In an effort to keep future versions as error free as possible, we ask that you send us your suggestions and notify us of any errors that you come across. You can contact the ETA software support group at the Troy office via:

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The DYNAFORM Team
Engineering Technology Associates, Inc