eta/DYNAFORM
User's Manual

Version 3.3

TABLE OF CONTENTS

Table of Contents

INTRODUCTION

The *eta/DYNAFORM Version 3.3* 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 Version 960, with the streamlined pre- and post-processor functions of eta/FEMB. 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 the prototyping costs and cycle time for product development.

The primary components of eta/DYNAFORM's analysis engine are LS-DYNA and LS-NIKE3D, programs developed and currently supported by the Livermore Software Technology Corporation (LSTC) of Livermore, California. Both are general purpose, non-linear, dynamic, finite element analysis codes utilizing explicit and implicit solver approaches for fluid and solid structural problems. These codes have been developed for applications such as automobile crashworthiness, occupant safety, underwater explosion, sheet metal forming, etc.

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 cost required to produce high quality panels and stamped parts. Specifically, 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.

Tooling surface data is used to predict stamping performance 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, SOLARIS, and SGI; LINUX platform 6.22+.

Special features for eta/DYNAFORM Version 3.3 include:

MESH REPAIR TOOLBAR
The functions most often used for mesh repair have been integrated in one function option window eliminating the need to jump between menus when repairing mesh.

BLANK GENERATOR
The Blank Generator function automatically generates the blank mesh based on surface or line data; for irregular shapes, triangle elements are created on the outer edge automatically. The user can determine the initial blank element size by the smallest or concerned tool radii.

BLANK MAPPING AND RESULT MAPPING
Blank Mapping maps a new more uniform mesh to the deformed blank for crash-type analysis. Result Mapping re-meshes and repositions the deformed blank.

ENHANCED SELECTION FUNCTIONS
Selection functions such as Select by Free Region, Select by Spread , etc have been added to enhance the selection of surfaces, element, nodes, and points.

NEW CALCULATION FUNCTIONS (area, radius)

Area calculates the area of selected elements and Radius displays the radius through three selected points/nodes.

NEW SURFACE FUNCTIONS

Mirror Surface, Untrimmed Surface, Duplicate Surface Check, etc. have been added.

ENHANCED POSTPROCESSOR

The quality and velocity of graphics have been improved with a new graphic engine.

IMPORT/EXPORT UG PART FILE

Line and surface data can be imported from UG part file and exported to UG part file.

ENHANCED ADAPTIVE MESH

In addition to setting up the angle for the adaptive mesh criteria, the user can set up the penetration tolerance and the absolute thickness of the blank.

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 a header-section S-channel using LS-DYNA. This example describes the process of the residual stress/strain effect on the stamped part during a loading simulation and illustrates the use of LS-NIKE3D for predicting spring back effects.

A step-by-step procedure is provided to guide users through the complete process and flow of the tasks. In most cases, new users can complete the training program in less than two hours and be ready to begin a production project.

The **eta/DYNAFORM Applications Manual** discusses the basics of the sheet metal forming process and provides users with some fundamental knowledge of tooling design and related topics. 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 help 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 a simplified combination of both the LS-DYNA and DYNAFORM Pre/Post User's Manuals; several sections specific to sheet metal forming simulation are also included.

For further details on LS-DYNA and LS-NIKE3D, users are kindly asked to refer to the respective manuals for these codes published by the Livermore Software Technology Corporation.

CHAPTER 1

STRUCTURAL OVERVIEW

1.1   GENERAL

eta/DYNAFORM Version 3.3 is a complete graphic user interface package that is operated on LINUX and UNIX based workstations (W/S) including IBM, HP, DEC-Alpha, SGI and SUN-SOLARIS platforms using various popular operating systems. The model generation, input file preparation, and results processing activities are all done in a W/S environment. The solution, either explicit or implicit, can be executed on both local W/S and/or remote server systems.

The program is organized as a tree structure and is operated and controlled by a MOTIF Graphic User Interface (GUI). The user activates the functions via mouse pick.

1.2   DISPLAY WINDOW

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

1. DISPLAY AREA
    Models and graphs are displayed in this area.

2. MENU BAR
    Commands and command options are displayed in this area.

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 to the user.

Menu Bar
Icon Bar
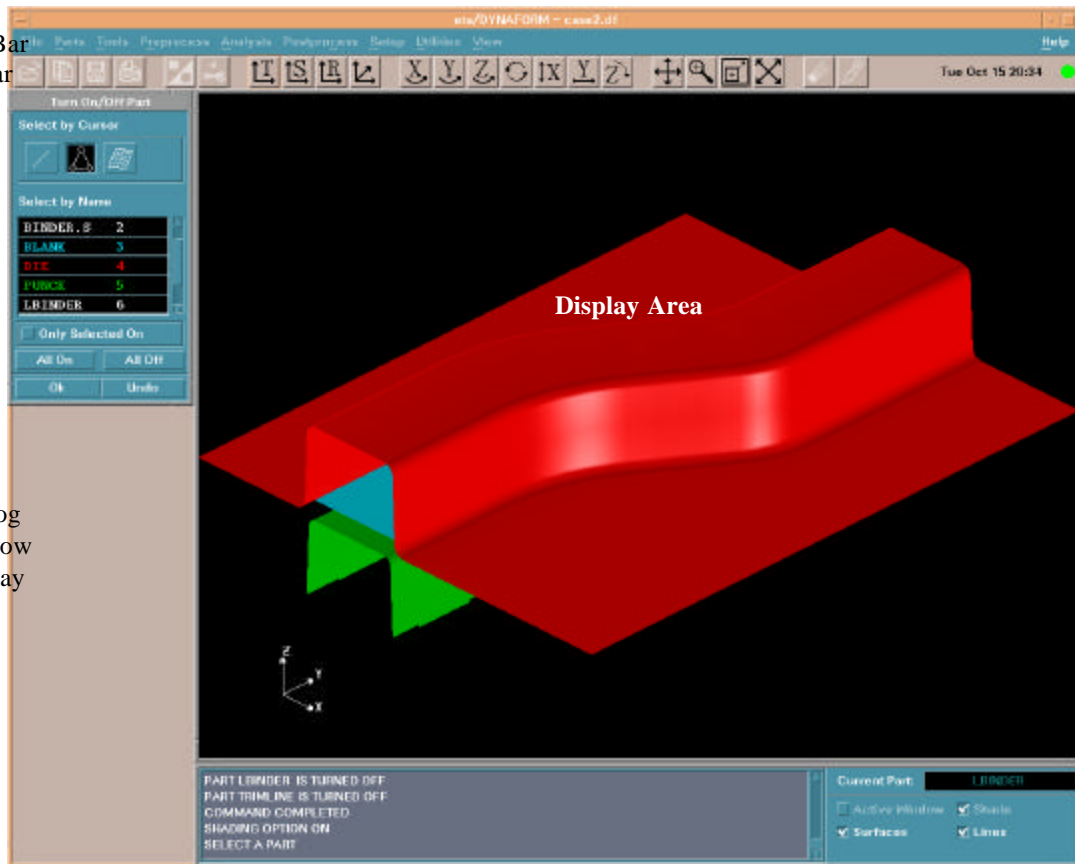
Dialog
Window
Display

Display Area

Prompt Area   Display Options

1.3   MENU BAR

Menus are selected by mouse pick and contain DYNAFORM's functions. Descriptions for these functions are located in their respective sections.

FILE MANAGER   Imports and exports data from eta/DYNAFORM.
PART CONTROL   Organizes lines, surfaces, and elements.
TOOL DEFINITION   Creates, defines, and modifies tool data.
PREPROCESS   Contains the menu of preprocessing functions.
ANALYSIS   Defines the parameters of the output.
POSTPROCESS   Provides options for viewing the results of an analysis.
SETUP MENU   Contains various preprocessing utilities that are
                      toggle switches activated.
UTILITIES   eta/DYNAFORM's "tool kit."
VIEW OPTIONS   Adjusts the display of the items on the screen.
HELP   Displays the information to contact eta/DYNAFORM Technical
                      Support.

1.4   ICON BAR

The icon bar is designed to give the user easy access to the most often used functions of eta/DYNAFORM. The user may simply click on these icons to activate the functions instead of browsing the various menus.

OPEN
This function allows the user to open a database.

Import
This function allows the user to import files, such as IGES, VDA and so on into the current database.

SAVE
This function updates the current database.

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.

PART ON/OFF
This function turns the selected parts on or off, the PART TURN ON/OFF dialog window is displayed once selected.

DELETE ALL UNREFERENCED NODES
This function deletes all nodes that are unreferenced nodes (nodes that are not part of an element)

TOP VIEW
This function automatically displays the model from the TOP or in the XY-plane.

SIDE VIEW
This function automatically displays the model from the SIDE or in the XZ-plane.

REAR VIEW
This function automatically displays the model from the REAR or in the YZ-plane.

ISOMETRIC VIEW
This function automatically displays the model from the isometric plane (60-degree isometric).

VIRTUAL X ROTATION
The displayed model will dynamically rotate about the global X-axis when the cursor is moved up or down.

VIRTUAL Y ROTATION

The displayed model will dynamically rotate about the global Y-axis when the cursor is moved up or down.

VIRTUAL Z ROTATION
The displayed model will dynamically rotate about the global 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.

SCREEN X ROTATION
The displayed model will dynamically rotate about the screen X-axis when the cursor is moved up or down.

SCREEN Y ROTATION
The displayed model will dynamically rotate about the screen Y-axis when the cursor is moved up or down.

SCREEN Z ROTATION
The displayed model will dynamically rotate about the screen Z-axis when the cursor is moved up or down.

PAN
This command translates the model by following the movement of the cursor. If the cursor is moved off the screen, the cursor reappears at the center of the screen. Clicking the left mouse button stops the pan.

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.

TRIM WINDOW
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. Release the left button, the section included in the window is displayed in full screen.

FILL
Rescales the model to include all entities belonging to parts that are currently turned on. FILL automatically zooms in or out until the model fits the viewing area of the screen.

CLEAR
This command removes the highlighted entities from the screen such as those that are made with the commands SHOW LINE, BOUNDARY CHECK, ID ELEMENTS, DEFINE TITLE, etc.

REDRAW

eta/DYNAFORM is currently designed to update the screen after each command. Occasionally, the use of specific commands requires that images be updated with an additional step (e.g., when the user operates the dynamic zoom at the same time that the element normals are displayed, activate the REDRAW command to resize the arrows that represent the element normals).

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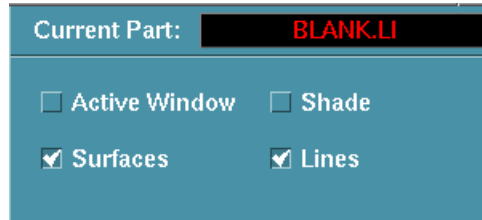
ACTIVE WINDOW (toggle)
This command allows the user to isolate a portion of the displayed geometry/model for more detailed viewing or editing. The user defines the region by dragging a window over the desired area. eta/DYNAFORM displays the elements, lines, and surfaces within the window (volume in space)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

SHADE (toggle)
This command 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.

SURFACE (toggle)
This function toggles all surfaces on and off.

LINE (toggle)
This function toggles all lines on and off.

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 depresses the left mouse button. The middle mouse button can be used as Enter or Complete within a function. The right mouse button displays a pull-down menu when clicked within the Graphics Display Window.

1.7   SPECIFICATIONS

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

        100,000   LINES
        200,000   POINTS

4,000	SURFACES
	-100,000 Edge Points (SURFACE)
	-150,000 Control Points (SURFACE)
250,000	GRIDS
250,000	ELEMENTS
1,000	PROPERTIES
1,000	PIDS

1.8   LINE DATA

eta/DYNAFORM directly reads in IGES, VDA and DYNAFORM/FEMB geometry data (line and surface).

1.9   RECOMMENDED NAMING CONVENTION
(.df, .his, .pac, .lin, .bin, etc.)

The protocol for naming files during an eta/DYNAFORM session includes attaching suffixes to the file names that specify the file types. 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**
4. eta/DYNAFORM model binary file name: *filename.***bin**
5. NASTRAN input file name: *filename.***dat**
6. LS-DYNA input file name: *filename.***dyn**
7. LS-DYNA model file name: *filename.***mod**
8. LS-NIKE3D input file name: *filename.***nik**
9. LS-DYNA/NIKE3D (etc.) history result file: *filename.***dyn**
10. eta/DYNAFORM binary result file: *filename.***pp**
11. UG part file   *filename.***prt**
12. VDA surfaces data file: *filename.***vda**

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.

This practice makes the file name selection convenient 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 allowing 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   TABLES

The tables in eta/DYNAFORM allow the user to enter properties for materials, elements, draw beads etc. The data fields contained in the table can be edited in several different ways. By single-clicking the mouse button the user 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.

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

ADVANCED - Activates advanced parameters.

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

Two configuration files (.DynaformDefault) are created to set all the control parameters. One is, by default, located in the DYNAFORM installation directory. This file requires super user privilege to modify. The other is provided to save the customized setting in the database or the user's home directory. It allows the user for modification. These two configuration files have the same control parameters. Each parameter will have a control flag to control the permission for modification. The format of each control parameter looks like the following:

       "Keyword   value   permission key".

For more information, refer to the description included in the configuration file.

CHAPTER 2
GETTING STARTED

2.1   GETTING STARTED

An eta/DYNAFORM session begins with an empty database with the default name UNTITILED.DF. The following scenarios are possible:

1. The user supplies CAD-IGES or VDA data that is translated and read directly into the eta/DYNAFORM database.

2. The user supplies model data (e.g. NASTRAN, LS-DYNA, etc.) which is read directly into the eta/DYNAFORM database.

3. The user generates model data from scratch or opens an existing database (*.df).

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

2.2   GETTING STARTED WITH CAD DATA

The user begins a session by entering the word "dynaform" at the UNIX prompt. Once the program is activated, the default database called UNTITLED.DF is created and the menu bar is displayed at the top of the screen. The user selects FILE/IMPORT and an IMPORT FILE dialog window will display.

From the window, select the button to the right of FILE TYPE to display a menu and select a file type (see Figure 2.2.1). eta/DYNAFORM supports CAD types including IGES, VDA, UG and LIN.

The options will be displayed in the menu as follows:
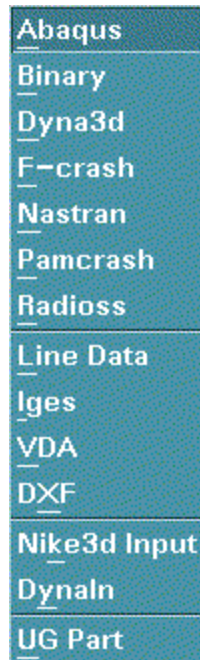
Figure 2.2.1 File Type

Once the user selects the appropriate file type and all files in the current directory with the suffix will display in the file list, select the desired file, then OK, the file will be import in the eta/DYNAFORM.

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

After the user starts eta/DYNAFORM with the default database,

1.   To read the LS-DYNA (or other analysis code) data file into your eta/DYNAFORM database, select FILE/IMPORT.

2.   A dialog window will display. Select the button to the right of FORMAT and choose the desired format. Please see the figure 2.2.1.

3.   Once the selection has been made, the program will read the model data 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

The intention of this section is to help users get 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. Activate the eta/DYNAFORM program.

    ● Select an eta/DYNAFORM database filename from the list or enter a new name in FILE/NEW.

2. 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.

3. 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 coordinate system or global system. See Section 2.5 LOCAL COORDINATE SYSTEM.

    ● Select 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 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 modeled, 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 automatically generate during the writing of the LS-DYNA file.

A DRAW BEAD can be easily 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 results can be viewed by selecting
FILE/OPEN in FILE to load the result files and activate the POSTPROCESS menu.

After the forming analysis is complete, the SPRING BACK menu in ANALYSIS can be used to read in the
NIKIN file and view the formed blank shape. A trimmed blank model can be written out into a NIKE3D
input file or the job can be submitted directly from the eta/DYNAFORM window. A blank may be trimmed
using the TRIM command of BLANK MISCELLANIES in the TOOL DEFINITION menu. Once the
analysis is complete, the results can be viewed by reading the NIKE N3PLOT files into the
POSTPROCESS menu.

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, LCS, or LAST.
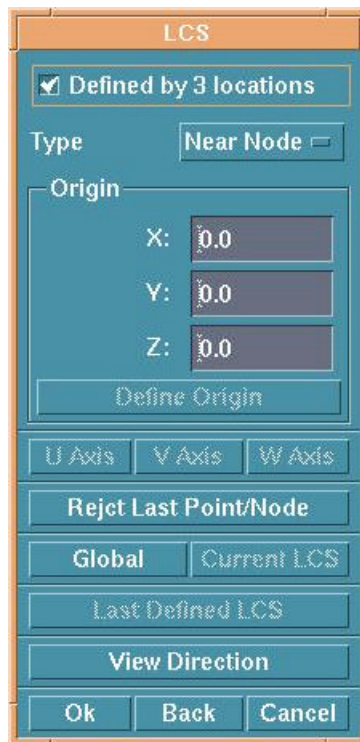
Figure 2.5.1 LCS

     GLOBAL

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

LCS

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

LAST

Defines the last coordinate system as 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 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 and the V-axis is determined by
        the cross product WXU.

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

    - Select DEFINE ORIGIN. The INPUT COORDINATE dialog appears. Select a point or node
      as the origin and 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 DONE.

- Select OK from the AXIS dialog.

- Select OK from the LCS dialog.

   Note: If the local system is not acceptable use the BACK button to reject and redefine.

- The coordinate system is displayed.

Figure 2.5.2 U Axis Definition

For more information on manipulating coordinate systems, refer to Section 2.5 LOCAL COORDINATE SYSTEM.

CHAPTER 3
FILE MANAGER

The options in this menu are used to new a file or open, save, import, export, restart and print current files.

Figure 3.1 File Manager

Please see Figure 3.1.

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.

- If a database file has been opened in eta/DYNAFORM, there is a warning message for saving the opened file before creating a new one, see Figure 3.1.1.
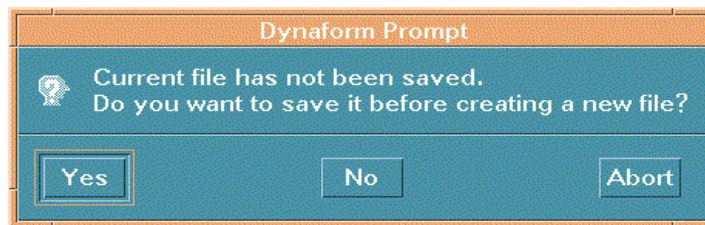
Figure 3.1.1 New File Prompt

● The NEW FILE dialog window will appear, see Figure 3.1.2.
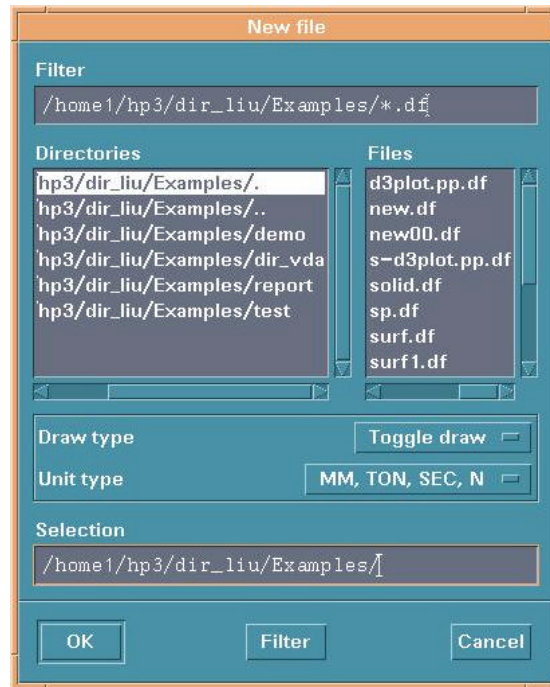
Figure 3.1.2 New File

● **FILTER**

Lists the current directory and file type. The default is *.df.

● **DIRECTORIES**

Lists all sub-directories in the current directory. The current directory is highlighted.

● **FILES**

Lists all files with the designated filter extension in the current directory. A file can be selected by double clicking or by selecting OK after picking.

● **DRAW TYPE**

Select the button beside DRAW TYPE. The five types will be shown, see Figure 3.1.3.

Figure 3.1.4 Unit Type

Figure 3.1.3 Draw Type

● **UNIT TYPE**
Select the button beside UNIT TYPE. The four types will be shown, see Figure 3.1.4.

● **SELECTION**
Lists the selected file and its path.

● **CANCEL**
Cancels the new file.

3.2   **OPEN** (CTRL + O)

This function allows the user to open a DYNAFORM database in the preprocessor and also a postprocessor database for post-processing by FORMAT. Please see the Figure 3.2.3

● If there is already a database opened, eta/DYNAFORM prompts the user to save the current file, see Figure 3.2.1.
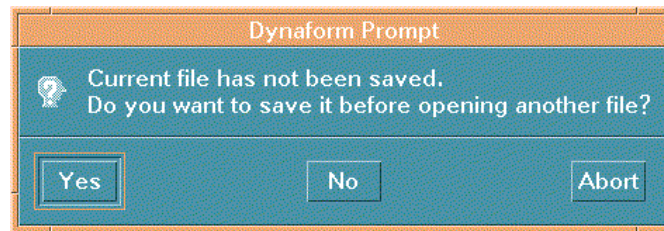
Figure 3.2.1 Open File Prompt

In order to open an existing database follow these steps (Refer to Figure 3.2.2):

1.   Pick the file type from the pull down menu in the format window.

2.   Choose a file in the file window that is in a corresponding format to the format chosen.

3.   The database will be opened.
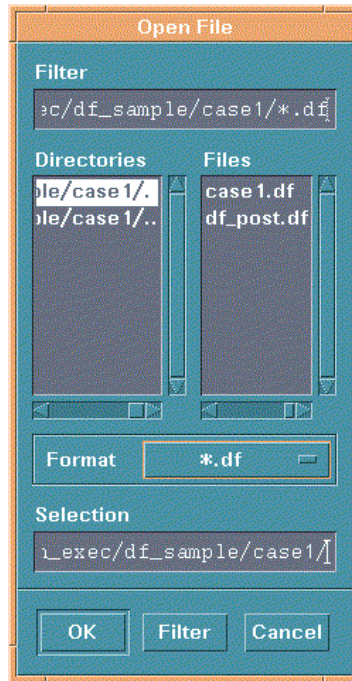
Figure 3.2.2 Open File
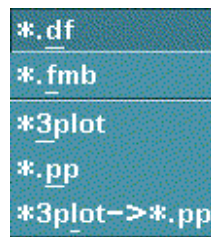
Figure 3.2.3 Database Format

3.3   RESTART

This function allows the user to restart the current database at the last saved point. eta/DYNAFORM will prompt the user to save the current file, see Figure 3.3.1.
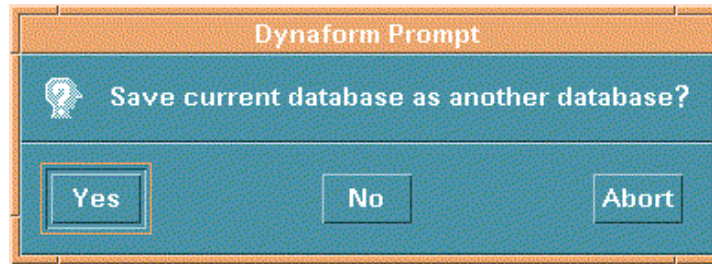
Figure 3.3.1 Save Current File

- Select YES to save the database as another name.

- Select NO and any unsaved data will be lost.

- Select ABORT to abort RESTART.

3.4   SAVE

This function updates the current database.

3.5   SAVE AS

This function saves the current database as a new file. The options are common with those in NEW, 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. Select the button APPLY to retain the dialog box for reading multiple files. The button OK will read only one file and close the dialog box.
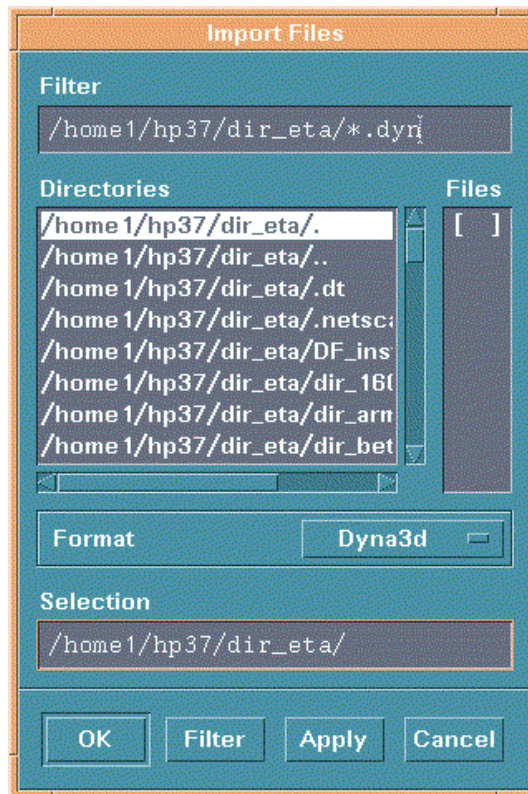
Figure 3.6.1 Import File

FORMAT

Select the button beside FORMAT to select the proper format. The available formats are:

MODEL DATA FORMAT:	Abaqus, Binary, Dyna3d, F-Crash, Nastran, Pamcrash, Radioss.
CAD DATA FILE:	Line data, IGES, VDA & UG.
SPRING BACK ANALYSIS FILE:	Nike3d Input.
MULTI-STAGE FILE:	Dynain .

Note: The binary model file contains only the finite element model data written by eta/DYNAFORM or data converted from a finite element application program such as ABAQUS. Any number of binary files may be read in at any time during the eta/DYNAFORM session. eta/DYNAFORM allows the user to offset the node/element numbers if the numbers have already been used in the database.

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 above. The formats are:

BINARY

Output elements and nodes only.

NASTRAN

Entire or partial model data.

LINE DATA

Lines and surfaces in DYNAFORM format.

IGES

Lines and surfaces in IGES format.

VDA

Lines and surfaces in VDA format.

UG

Lines and surfaces in UG part file format.

3.8   GRAPH PLOT

This function allows the user to import a time history file.

3.9   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 in Section 3.10.

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.
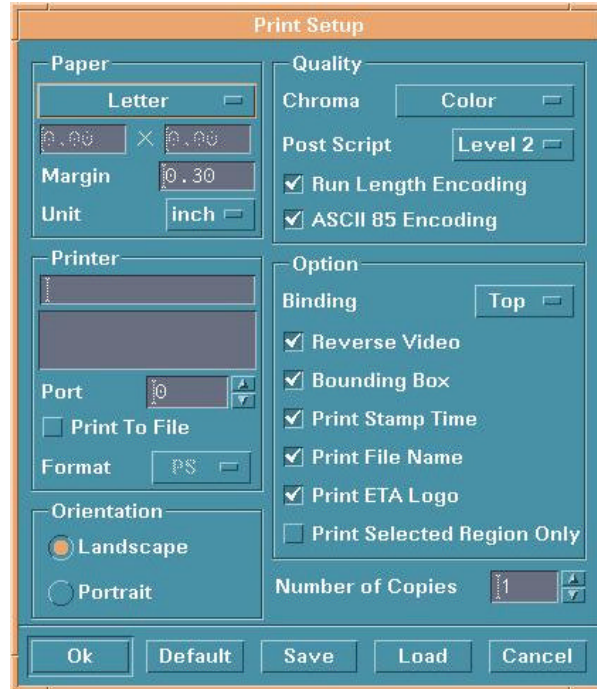
Figure 3.10.1 Printer Setup

3.10.1   PAPER

The user can specify a paper size and paper margin.

- 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 beside MARGIN to define the shorter of the paper's two margins (top and bottom). The program automatically determines the second margin in order to maintain the original aspect ratio of the picture. This feature can also be used to scale the picture.

- Select the button beside UNIT to choose a unit (inch or mm).

3.10.2   PRINTER

The user can select a printer or select 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 generally sets the port number.

- If PRINT TO FILE is toggled on, the user can choose a format by selecting the button beside FORMAT.

    Note: **When PRINT is selected, the user will be prompted to enter a filename instead of the postscript file being directly sent 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 button beside CHROMA to select a color type.

POSTSCRIPT

There are two levels, level 2 is the default setting and 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 dramatic (for PS Level 1 these are automatically turned off).

3.10.5   OPTION

These options are 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 at the lower right corner of the picture.

PRINT FILE NAME

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

PRINT ETA LOGO

This function prints eta/DYNAFORM at 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 in 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 will load the system defaults that are saved in a ".FembHardcopyDefault" file located in the eta/DYNAFORM executable directory.

3.10.8   SAVE

This function saves the user-modified hardcopy options to the ".FembHardcopyDefault" file located in the user's home directory.

3.10.9   LOAD

This function will load the user-defined defaults that have been saved to the file ".FembHardcopyDefault" located in the user's home directory.

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

3.11   **EXIT** (Ctrl + X)

This function allows the user to exit the program. eta/DYNAFORM will prompt to save current database.

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.
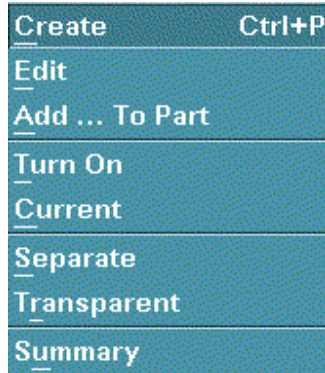
Figure 4.1 Part Control

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

4.1   CREATE NEW PART (Ctrl + P)

This function allows the user to make a new part. See Figure 4.1.1
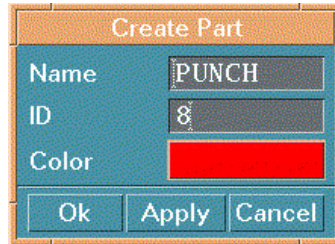
Figure 4.1.1 Create Part

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 and the SELECT COLOR window is displayed allowing the user to choose a color, see Figure 4.1.2.
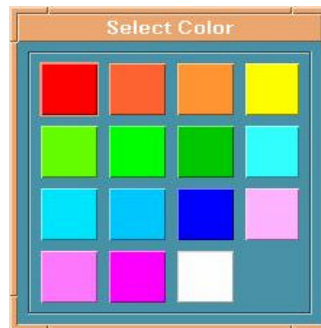
Figure 4.1.2 Select Color

4. Clicking the OK button will create a new part and exit the create part function. Clicking the APPLY button willl 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   PART EDIT

The functions in this menu allow 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 Part Edit

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.   The user has the option to change the part name, ID, and/or color.

3.   Select MODIFY and the part will be modified.

4.2.2   DELETE PART

Choose a part to remove and select DELETE. A DYNAFORM Question window appears which allows the user to confirm or cancel the delete.

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.
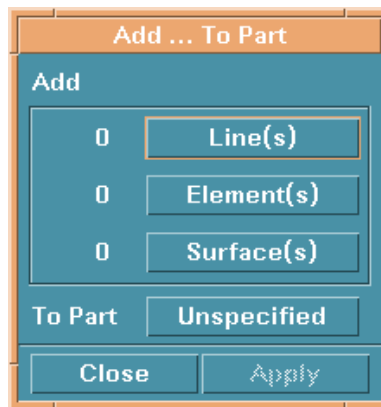
Figure 4.3.1 Part Add

4.3.1   ADD LINE

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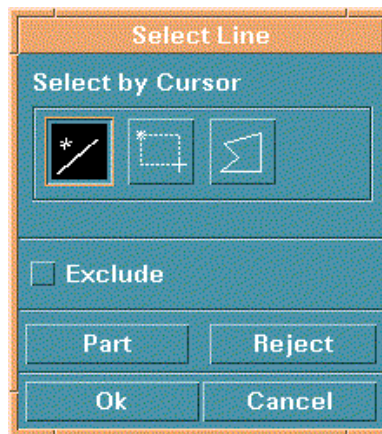
Figure 4.3.2 Select Line

- The SELECT LINE dialog window offers the user four ways to select a line. The color of the selected line is changed to white.

1. Select [icon] to pick a line by clicking its location.

2. Select [icon] to pick lines inside a rectangular window. The user can define the rectangle by selecting two points as its diagonal.

3. Select [icon] 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 PART to pick lines included in the part.

- Deselect lines.

    1. Toggle the EXCLUDE light ON. The user can exclude the selected lines using the above functions.

    2. Select REJECT and the user rejects the last selection.

- Select OK or CANCEL.

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

- Add a selected line to the part, 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.   There are

Figure 4.3.3 Select Element

two sets of functions depending on whether SELECT BY CURSOR or SELECT BY is ON.

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

Select [icon] to pick an element by its location.

Select [icon] to pick elements on a surface.

Select [icon] to pick elements inside a window.

Select [icon] to pick elements that the angle between adjacent elements is less than the determined value.

Select [icon] to pick elements inside a polygon.

● If SELECT BY is toggled ON, there are three other ways to select an element. The user may select by part, type, or range, see Figures 4.3.4 ~ 4.3.6. Select the combination box to the right of the SELECT BY button to choose one of them.
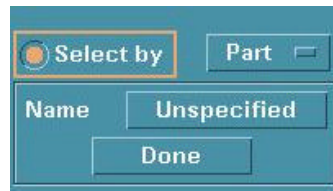
Figure 4.3.4 Select by Part

Figure 4.3.5 Select by Type
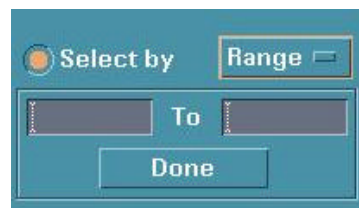
Figure 4.3.6 Select by Range

Figure 4.3.7
Element Type

1.   SELECT BY PART
      See Section 4.2 for choosing a part. All elements in the part will be selected when this function 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 displayed on the screen will be selected, see Figure 4.3.5.

> **NOTE:** **Refer to LS-DYNA user manual for a description of the nine types of elements. Deselecting elements and closing the dialog box are similar to those in Section 4.5.1.**

3. SELECT BY RANGE

    Type the numbers in the Pop-Up window. All elements within that range will be selected, see Figure 4.3.6.

●   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 the same functions as those in SELECT LINE.

●   Select OK to finish the selection of elements and go back to the PART ADD menu.

4.3.3   ADD SURFACE
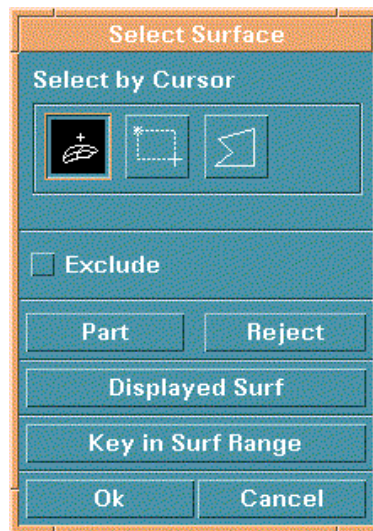
Figure 4.3.8 Select Surface

●   Select SURFACE(S) and the SELECT SURFACE dialog window appears, see Figure 4.3.8.

●   **DISPLAYED SURF**

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

●   **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.
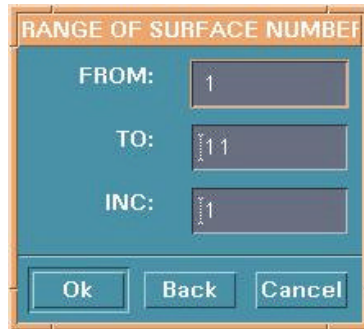
Figure 4.3.9 Range of Surface

Other functions are common with 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 your selection to the specified part when APPLY is selected. Select CANCEL to abort the addition to the part.

4.4   TURN ON

This function turns 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.

● A part can be turned off by selecting a line, element, or surface. It can also be toggled
    on/off by selecting its name in the SELECT BY NAME list.
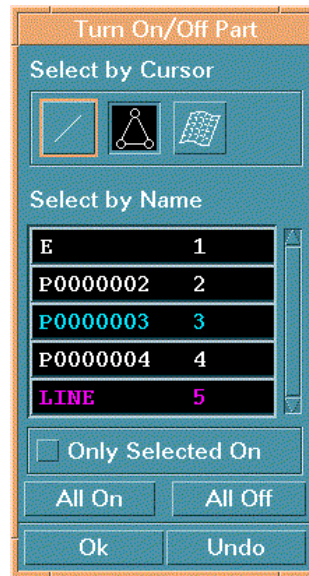
Figure 4.4.1 Turn On/Off

● Only one part can 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 changes 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 (the user can also click on this field to
change the current part). A SELECT PART dialog window is displayed, see Figure 4.5.1.
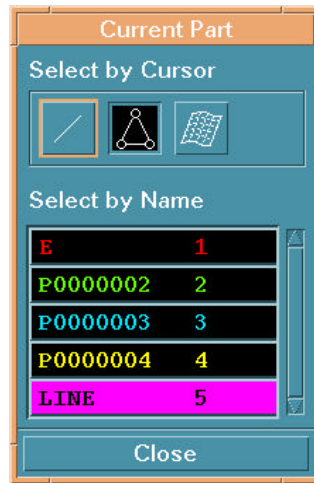
Figure 4.5.1 Select Part

● 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. This function also provides the option to create zero length rigid links between the separated parts. Once the parts are separated, each common node becomes several nodes, one for each part, at the same location.

● A DYNAFORM Question window appears, see Figure 4.6.1.
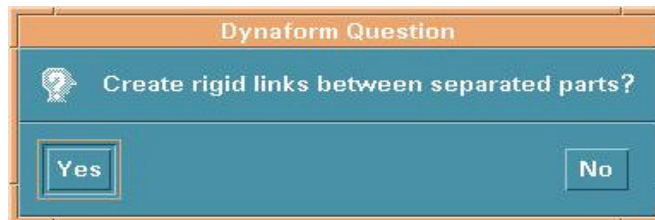
Figure 4.6.1   Dynaform Question

The functions in the SELECT PARTS dialog window are similar to those in Section 4.2 (see Figure 4.6.2). Select the ALL PARTS button for separating all parts or selected parts. If the answer to the DYNAFORM Question is YES, a part called ZEROBAR is automatically created and it contains all newly created nodal rigid bodies.

4.7   TRANSPARENT

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

Figure 4.6.2 Select Part to Separate

4.8   SUMMARY

This function displays statistics regarding the geometry, material, and interface information of the selected parts. Figure 4.8.1 is an example.
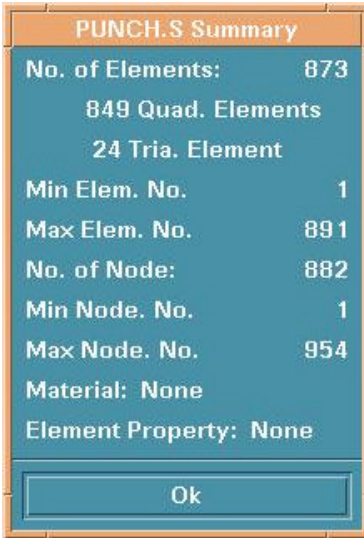
Figure 4.8.1 Part Summary

CHAPTER 5
TOOL DEFINITION

The functions in this menu are shown in Figure 5.1. The user can create tool pieces via CAD data, generate blank mesh, define materials, define properties, generate load curves, define interface contact parameters, position tool pieces, create draw beads, view movement of defined tools and modify deformed blank

Define Tools	Ctrl+T
Position Tools	▷
Draw Bead	Ctrl+D
Define Blank	Ctrl+B
Blank Operation	▷
Material	
Property	
Animate	
Tools On/Off	
Summary	

Figure 5.1 Tools Menu

shapes.

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

5.1   DEFINE TOOLS (Ctrl + T)

In eta/DYNAFORM, there are three standard tools: die, punch, and binder. The user can also create his own tools. To define a tool in detail, the tool must be current. The top two control groups in this dialog window (Figure 5.1.1) provide ways to create a tool and specify a tool as the current tool to be defined.

● Create a user-defined tool as follows:
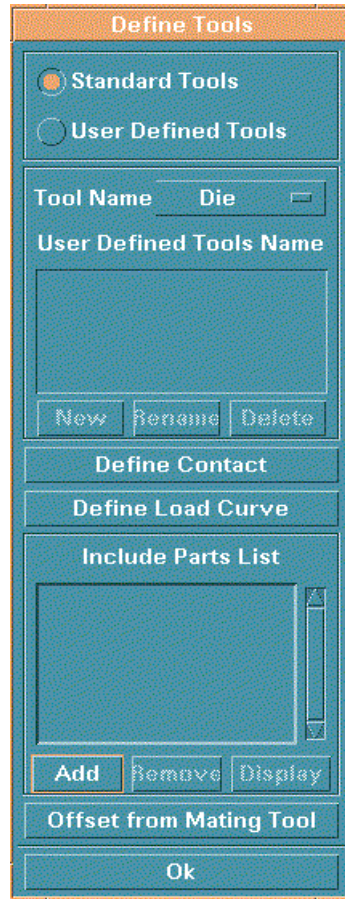
Figure 5.1.1 Define Tools

1. Select USER DEFINED TOOL.

2. Select NEW and the NAME OF NEW TOOL window displays, see Figure 5.1.2. The user can also rename or delete a tool by selecting the other buttons.

3. Type the name in the window.
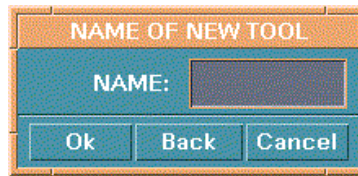
Figure 5.1.2 Name of New Tool

4. Select OK to close the window. The name will be listed in the USER DEFINED TOOLS window of the submenu.

● Specify a standard tool as a current tool:

1. Select the STANDARD TOOLS window.

2. Select a standard tool in the TOOL NAME combination window.

● Specify a user's tool:

1. Select the USER DEFINED TOOLS window.

2. Select a name in the USER DEFINED TOOLS NAME list window.

5.1.1   ADD PART

In general, a tool is a set of parts. This function will let the user put additional parts into the current tool.

● Select ADD, then choose a part (see Section 4.5, PART/CURRENT PART for more information). The selected parts will be listed in the INCLUDE PART LIST window.

5.1.2   REMOVE PART

● Select a part in the INCLUDE PART LIST. The part is removed from the current tool.

5.1.3   DISPLAY TOOL

● Select DISPLAY and all the parts included in the current tool will be displayed.

5.1.4   DEFINE CONTACT

This function is used to define contact interface parameters between the blank and a tool. In most cases, the user can just use the default setting.

Note: Refer to the *LS-DYNA User's Manual* for an explanation of each interface type and parameter.

Select DEFINE CONTACT to change the default settings. The TOOLS CONTACT dialog window appears, Please see Figure 5.1.3.

Figure 5.1.3 Define Contact

- **DEFINE CONTACT TYPE**

    Select the CONTACT TYPE combination list box and select the desired type.

    NOTE: For sheet metal forming, eta/DYNAFORM uses nine contact interface types listed below.

    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 MAJOR PARAMETERS**

The user can change the default values in the text boxes.

- ● **DEFINE MORE PARAMETERS**

    Select DEFINE CONTACT PARAMETERS. Change the values in the CONTACT PARAMETER dialog window and select OK, RESET, or CANCEL to close CONTACT PARAMETER. Select OK to finish defining contact.

5.1.5   DEFINE LOAD CURVE

This function is used to define, read in, or modify either a tool motion curve, or a tool force curve. As can be seen in figure 5.1.4, there are two toggles inside of the CURVE TYPE box. Here, you choose which type of curve you would like to create, and also the direction it will be in.

Figure 5.1.4 Define Load Curve

Select the MOTION (toggle), the user can define the travel of tools.
Select the FORCE (toggle), the user can define the force of tooles.

- • **SHOW CURVE**

    Shows a graph of the current curve, and also provides many options for viewing the graph. See Section 10.9.8 for more details.

- **ASSIGN CURVE**

    This function is used to assign a travel curve to a tool from a list of defined load curves, see Figure 5.1.5 Select a curve name and click OK to assign the curve to the current tool.
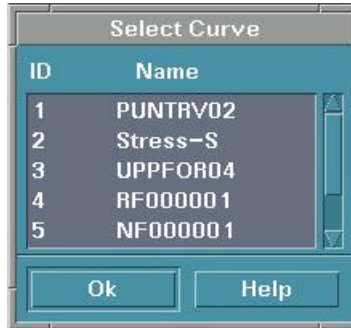
Figure 5.1.5 Assign Curve

- **READ**

    Allows you to read in a curve from another location. See figure 5.1.6 below.

Figure 5.1.6 Open Curve

- **AUTO**

    This function is used to automatically generate a curve according to the input values. For Motion curve see Figure 5.1.7; for Force Curve see Figure 5.1.8. To automatically create the

curve, input values for Begin Time, Force, and End Time, then click OK to create a curve. Define Auto Curve for Motion as follows.

Figure 5.1.7 Auto Curve for Motion   Figure 5.1.8 Auto Curve for Force
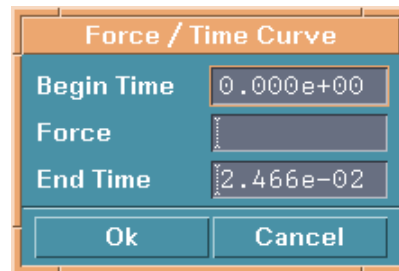
1. Select a Curve Type; Velocity is the curve of velocity versus Time; Displacement is the curve of displacement versus time.

2. Select a type in the CURVE TYPE window. There are 3 types of load curve that can be generated automatically by AUTO LOAD CURVE:

*   Trapezoidal – simplified analytical load curve.

*   Sinusoidal – used to simulate the cam action for the tool.

*   Sinusoidal with Hold – same as sinusoidal except the maximum velocity will be held constant. This is the default curve.

3. Input values for Begin Time, Velocity, and Stroke Distance.

4. If the phase of the curve is more than one, the user can select NO and input types and values for a second travel.

5. Select YES and a curve is created. A dialog window appears showing the motion curve, see Section 10.9.8 UTILITIES/LOADCURVE/SHOW LOAD CURVE.

*   **REMOVE CURVE**

This function removes the applied load curve that defines the motion. A warning dialog window appears, see figure 5.1.9.
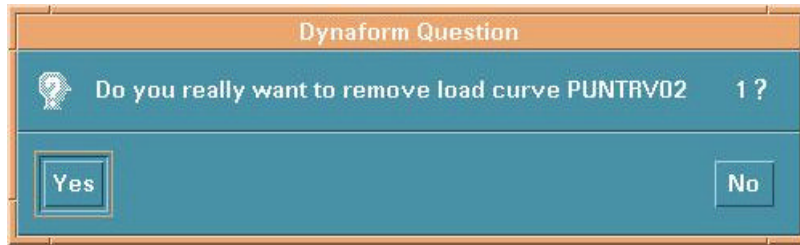
Figure 5.1.9 Remove Warning

Select YES to accept or NO to reject.

- **MODIFY**
  This function is used to make alterations to the current curve. See section 10.9.4 for details.

- **MANUAL**
  This function is used to create a curve by point definition. If the curve is a Motion Curve, please see Figure 5.1.10. If the curve is a Force Curve, see Figure 5.1.11.
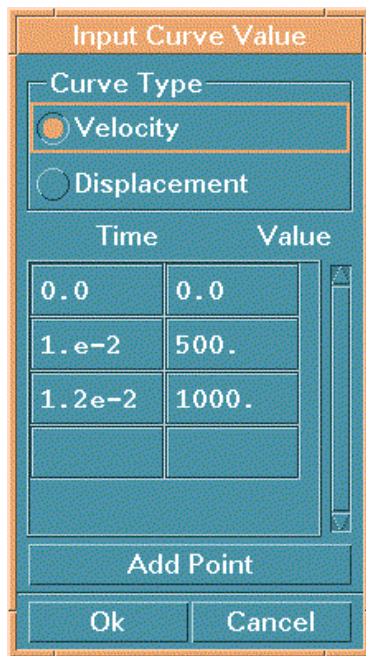
Figure 5.1.10 Input Curve Value for Motion   Figure 5.1.11 Input Curve Value for Force
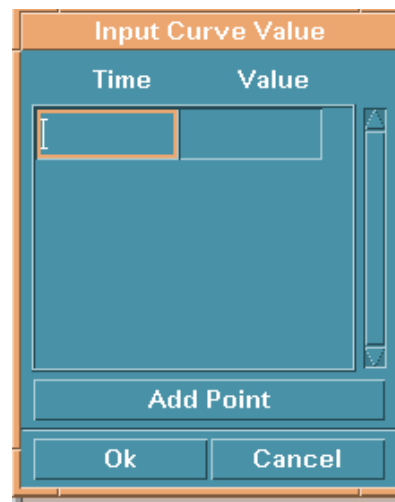
Define a motion curve as follows:

1.   Select VELOCITY/TIME or DISPLACEMENT/TIME.

2.   Type a pair of numbers in the field below TIME and VALUE.

3.   Select ADD POINT and then enter the next points.

4.   Select OK to end curve definition.

5.1.6   GENERATE A TOOL FROM MATING TOOLS

This function is used to generate the current tool from an existing tool. It combines the operations of copying or offsetting elements, creating parts, and adding parts to the current tool.

1.   Select OFFSET FROM MATING TOOL and the TOOL OFFSET dialog window displays, see Figure 5.1.12.

2.   Select SELECT ELEMENTS. The user can select elements in the dialog window, see Section 4.3.2, PART/ADD ELEMENT for more information.

3.   Toggle the INCLUDE IN CURRENT PART function on/off. If the function is ON, the new copied or offset elements will be in the current part and the current parts will be included in the current tool. Otherwise, a new part will be automatically created. It contains all new elements and is contained in the current tool.

4.   Toggle the function NORMAL OFFSET on/off. If it is ON, the new elements will be offset at the location based on input values of distance and thickness. Otherwise, the new elements will be copied at the location based on the values of distance.

5.   Toggle the function SHOW NORMAL on/off. If on, the displayed parts will be shown normal with their normals.

6.   Type the value in the DISTANCE field for moving the new elements in the stroke direction. It is always better to leave the value at zero.

7.   Type the value in the THICKNESS field for offsetting elements in the element normal direction.

8.   To apply the offset to the element, select APPLY.

9.   Select OK to exit the window.
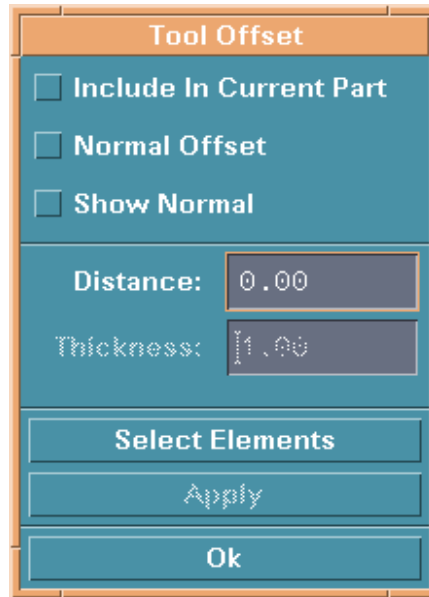
Figure 5.1.12 Tool Offset

5.2   POSITION TOOLS

There is a submenu indicator on this menu, the three submenus are:

1. Auto Position

2. Move Tool

3. Min. Distance

5.2.1   AUTO POSITION

This function automatically repositions the tools according to the user-defined draw type tool clearance and the stroke direction. Please see Figure 5.2.1 The master tool is the stationary tool. The movement of each slave tool by the AUTO POSITION function is saved and listed by the LIST MOVEMENT function. See Section 9.1 SETUP/ANALYSIS PARAMETER to change the setting.

Figure 5.2.1 Auto Position

5.2.2   MOVE TOOL

This function allows the user to translate any tool in any global direction.

1.   Select a defined tool from the tool list.

2.   Select the desired translational direction. The direction with the checkmark is current.

3.   Enter the number in the DISTANCE field.

4.   Select APPLY to move the tool in the selected direction the input distance. If APPLY is selected more than once, it will repeat the move.

    5.   Select REVERSE to move the tool in the reverse direction the input distance.

    6.   Select OK to close the window.

5.2.3   MIN (IMUM) DISTANCE

This function measures the minimum distance between two selected tools, see Figure 5.2.2.

Figure 5.2.2 Min. Distance

    1.   Select the master tool in the first tool list and the slave tool in the second tool list.

    2.   Select the direction of the distance between selected tools.

    3.   After both tools and the direction are selected, the minimum distance is displayed automatically.

    4.   Select OK to close the window.

5.3   DRAW BEAD (Ctrl + D)

The functions in the DRAW BEAD menu are used to create, modify, and lock draw beads on the user-defined part. The dialog window is shown in Figure 5.3.
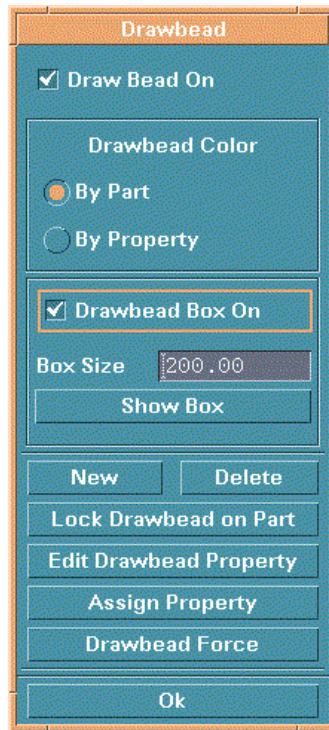
Figure 5.3 Draw Bead

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

5.3.1   EDIT DRAW BEAD PROPERTY

Select EDIT DRAW BEAD PROPERTY and the DRAW BEAD PROPERTY dialog window is displayed, see Figure 5.3.1. The functions of the buttons in this window are similar to those in the MATERIAL and PROPERTY sections (Sections 5.6, 5.7). The table DRAW BEAD PROPERTY is shown in Figure 5.3.2.
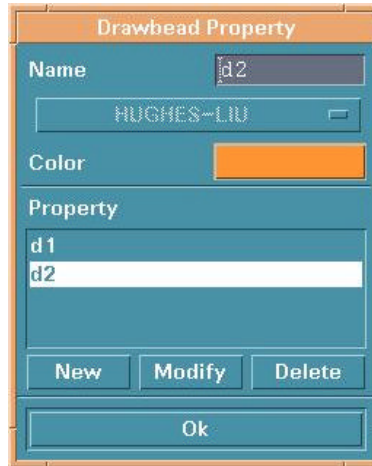
Figure 5.3.1 Draw Bead Property

Figure 5.3.2 Draw Bead Property Table

5.3.2   CREATE DRAW BEAD

This function allows the user to create draw beads. A draw bead is a line consisting of nodes and must have a property. The user can create a draw bead by completing the following three steps:

1. Select NEW in the DRAW BEAD window, see Figure 5.3.

    ● Select a property from the DRAW BEAD PROPERTY window, see Figure 5.3.1.

2. Select an existing property or create a new property.

    ● Select OK to close the window. The dialog window CREATE DRAW BEAD will display, see Figure 5.3.3.

3. The user may create draw beads and the locations for the nodes of the draw bead. This window allows the user to choose a way to create beads.

● Select LINE and then pick the location of a displayed geometric line. The nodes will be created on the line, one node at each point. The user can create another draw bead by selecting another line.

● Select POINT/NODE and choose DONE. The INPUT COORDINATE dialog window is displayed. See Sections 6.1 and 6.1.1 in Chapter 6 PREPROCESS for more information.

● If the user wishes to create other draw beads using another property, select EXIT or DONE and repeat steps 1 and 2.

● To end this function and return to the DRAW BEAD window (Figure 5.3.3) select ABORT.

Figure 5.3.3 Create Draw Bead

5.3.3   DELETE DRAW BEAD

Select DELETE in the DRAW BEAD dialog window. The SELECT DRAW BEAD dialog window (see Figure 5.3.4) provides the following operations for deleting the draw bead:
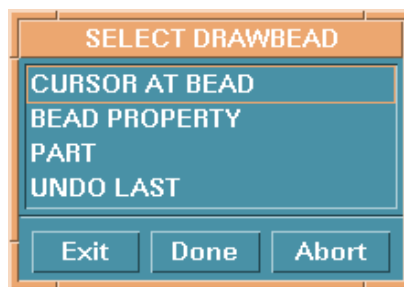
Figure 5.3.4 Select Draw Bead

1. SELECT BY LOCATION

● Select CURSOR AT BEAD, then select the draw bead(s).

2. SELECT BY BEAD PROPERTY

   ● Please see figure 5.3.5 Select Draw Bead Property.

   ● CURSOR AT DRAW BEAD; all draw beads with the same property of the selected draw
     bead will be deleted.

   ● PROPERTY NAME; all draw beads with the selected property will be deleted.

3. SELECT BY PART

   ● Select PART and choose DONE to open the SELECT PART window (see Figure 4.5.1 of
     Section 4.5 PART/CURRENT PART).

   ● Select a part and close the part window. All beads in the selected part will be deleted.

4. DESELECT

   ● Select UNDO LAST and the last selection will be canceled.

After selecting the draw bead(s):

5. Select EXIT or DONE to confirm the deletion and exit the window.

6. Select ABORT to cancel the deletion and exit the window.

5.3.4   ASSIGN PROPERTY

This function allows the user to change the property of a defined draw bead. To complete the change, the
user needs to go through steps similar to creating a draw bead.

Figure 5.3.5 Select Draw Bead Property

1. Select ASSIGN PROPERTY to display the SELECT BEAD PROPERTY dialog window, see
   Figure 5.3.5.

2.   Select the property to assign.

- Select PROPERTY NAME and then DONE. The process for this option is the same as Section 5.3.2.

- Select CURSOR AT DRAW BEAD and choose an existing draw bead. The property of the chosen draw bead is selected.

3.   Select the draw beads to modify.

- After selecting the property, the SELECT DRAW BEAD dialog window (Figure 5.3.4) is displayed. To select draw beads, see Section 5.3.3.

5.3.5   LOCK DRAW BEAD TO PART

This function assigns a draw bead to a rigid body part. The SELECT PART dialog window is displayed. See Figure 4.5.1 of Section 4.5 PART/CURRENT PART for more information on part selection.

5.3.6   DRAW BEAD FORCE

This function calculates the draw bead bending and normal forces based on the input of the draw bead geometry and the blank material properties. The dialog window is shown in Figure 5.3.6.
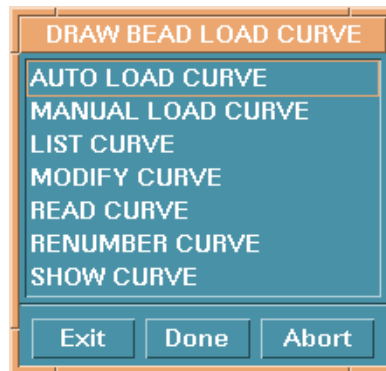
Figure 5.3.6 Draw Bead Force

-DRAW BEAD FORCE
AUTO LOAD CURVE
Select AUTO LOAD CURVE and then DONE.   The DRAW BEAD FORCE PREDICTION window is displayed. The user can edit the draw bead type and the values of the parameters.

-DRAW BEAD FORCE
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 common with those in Section 5.1.5.

5.3.7   DRAW BEAD ON

Toggles the draw bead ON to display the bead lines.

5.3.8   DRAW BEAD COLOR

● Select BY PART to display the draw bead in the color of its assigned part.

● Select BY PROPERTY to display each draw bead in the color of its assigned property.

5.3.9   DEFINE BOX AND SHOW BOX

The DEFINE BOX function is used to define a box around the draw bead to limit the size of the master surface considered for the draw bead. The SHOW BOX function is used to display graphically the draw bead boxes that have been defined.

● Toggle on SHOW BOX ON to activate the field BOX SIZE and the SHOW BOX button.

● Type the desired size in the BOX SIZE field.

● Select SHOW BOX and the location of a draw bead to show the box.

5.4   BLANK GENERATOR

This function creates a blank mesh from an existing surface or line; see Figure 5.4.1. Select BLANK GENERATOR, the SELECT OPTION dialog window is displayed.

Figure 5.4.1 Select Option for Blank Generator   Figure 5.4.2 Concern Tool Radii

● BOUNDARY LINE- creates the blank mesh from a boundary line.

● SURFACE- creates the blank mesh from a selected surface.

5.5   DEFINE BLANK (Ctrl + B)

Select the menu item DEFINE BLANK. The dialog window is shown in Figure 5.5.1.

Figure 5.5.1 Define Blank

5.5.1   ADD PART

This function adds parts to the blank. It is similar to adding parts to a tool (see Section 5.1.1).

5.5.2   DEFINE BLANK MATERIAL

Select the button to the right of MATERIAL and the MATERIAL dialog window appears. Select an existing material or create a new material, see Section 5.6.

5.5.3   DEFINE BLANK PROPERTY

Select the button to the right of PROPERTY. The PROPERTY dialog window appears. Select an existing property or create a new property, see Section 5.7.

5.6   BLANK OPERATION

The submenus in this menu allow user to change shape of a blank. There are eight submenus. Please see the figure 5.6.1.

Figure 5.6.1 Blank Operation

1. Auto Blank Position

2. Blank Mapping

3. Result Mapping

4. Dynain Contour

5. Trim

6. Tip

7. Tailor Welded
8. Lacing

5.6.1   AUTO BLANK POSITION

This function is used to automatically position the blank with reference to the selected tool, direction and contact gap. The dialog box is shown in Figure 5.6.2.

Figure 5.6.2 Auto Blank Position

5.6.2 BLANK MAPPING

This function allows the users to map a flat remeshed blank to the deformed blank shape.

>   Once MAPPING is selected, eta/DYNAFORM prompts:
>   **SELECT PART WITH INITIAL SHAPE**

>   After selecting the blank with the initial mesh. eta/DYNAFORM will prompt:
>   **SELECT PART WITH DEFORMED SHAPE**

>   Once the user selects the deformed blank, eta/DYNAFORM will prompt:
>   **SELECT REMESH PART FOR BLANK MAPPING.**

Once the user has selected the blank with the remesh, eta/DYNAFORM will automatically map the remeshed blank to the deformed blank shape.

5.6.3 RESULT MAPPING

This function allows the users to map a deformed blank with strain and stress information to the remeshed blank with same shape.
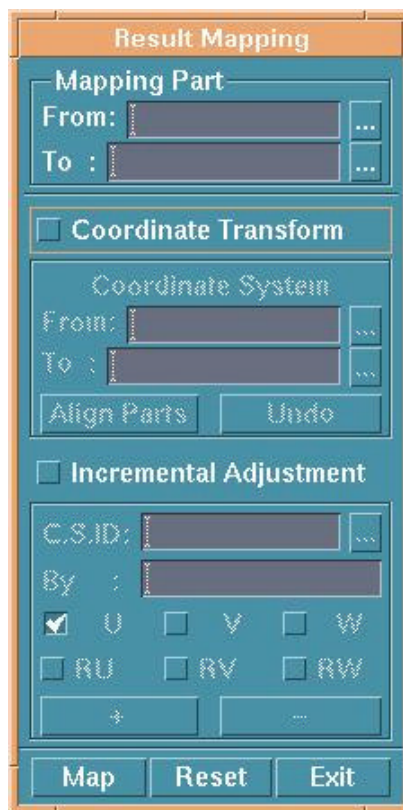
Figure 5.6.3 Result Mapping

Once File/Import a nastran file with remeshed blank and DYNAIN files with the deformed blank, the result mapping can be used. Select RESULT MAPPING, a RESULT MAPPING window appears, see the Figure

5.6.3.

- Select the FROM button to select the deformed blank, and select the TO button to select the new remeshed blank.
- Select Coordinate Transform toggle, then define (select button) or input (already existing) two coordinate systems to align the deformed part and new remeshed part.
- If the deformed blank doesn't align the new blank, please select INCREMENTAL ADJUSTMENT to adjust the deformed blank position. The user can input or define C.S, input the value and select direction to adjust the deformed blank position.   After align the two parts, select MAP to finish mapping.
- Select Reset to recover the deformed part to the original position, and select EXIT to close the result mapping without mapping.

5.6.4 DYNAIN CONTOUR

Import DYNAIN file, the TOOLS/BLANK OPERATION/DYNAIN CONTOUR can be used. Please refer to the section POST-PROCESSING.

5.6.5   TRIM

This function is used to trim a blank along the projection of the trim line(s). The elements that have been trimmed from the blank are included in a new part called TRIMOUT.

1. The SELECT LINE window appears first. The user selects a trim curve.

2. After a line is selected, the LCS dialog window appears (see Section 2.5 LOCAL COORDINATE SYSTEM). The user defines the projection direction, which is the W-direction or global Z-axis.

3. The TOLERANCE dialog window appears, see Figure 5.6.4. The user can define the trimming tolerance with a value between 0 and 1. The default value is 0.3.

Figure 5.6.4 Tolerance

4. Finally, a DYNAFORM Question dialog window appears, see Figure 5.6.5. YES excludes the elements outside the boundary of the trim line; NO excludes elements within the boundary of the trim line.
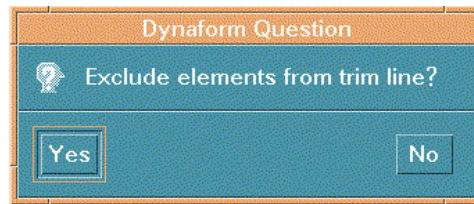
Figure 5.6.5 Dynaform Question

5.6.6   TIP

This function tips the blank according to a user-defined coordinate system.

- Drag the slider in BLANK TIP window (see Figure 5.6.6) to set a tip angle increment.

- Select DEFINE W-AXIS. The LCS dialog window appears, see Section 2.5 LOCAL COORDINATE SYSTEM.

Figure 5.6.6 Blank Tip

- After the coordinate system is created, select APPLY. The blank is changed and rotated about the W-axis by the input angle increment. Select REVERSE to rotate the blank in the opposite direction.

- Select REPEAT to repeat the process.

5.6.7   TAILOR WELDED

This function gives the users the ability to weld parts of different material thickness to define a blank.

- Once the user selects TAILOR WELDED, eta/DYNAFORM displays the SELECT PART dialog window (Figure 4.5.1) and prompts:
  SELECT TARGET PART FOR TAILOR

● After the target part is selected, eta/DYNAFORM displays the SELECT NODE dialog
  window (Figure 6.4.2) and prompts:
                              SELECT NODES TO TAILOR WELD

● After the nodes are chosen, select OK. The prompt window will display the number of
  spotweld nodes created and the welded part will be shown in the display area.

5.6.8   LANCING

This function allows the user to split the blank along a selected line.

● Select elements.

● Define LCS.

● After select elements and define LCS, the control option window appears.

LINE
 Once a line is selected, all elements crossing the line are split along the line.

LINE SEGEMENT
This option allows user to select more than one line; all elements crossing the line segments
are split.

NODES
This option allows user to select two nodes, all elements crossing the line consisting of the
two nodes are split.

5.7   MATERIAL

This dialog window allows the user to create, modify, and delete defined materials, as well as save and read
materials to the library. The dialog window is shown in Figure 5.7.
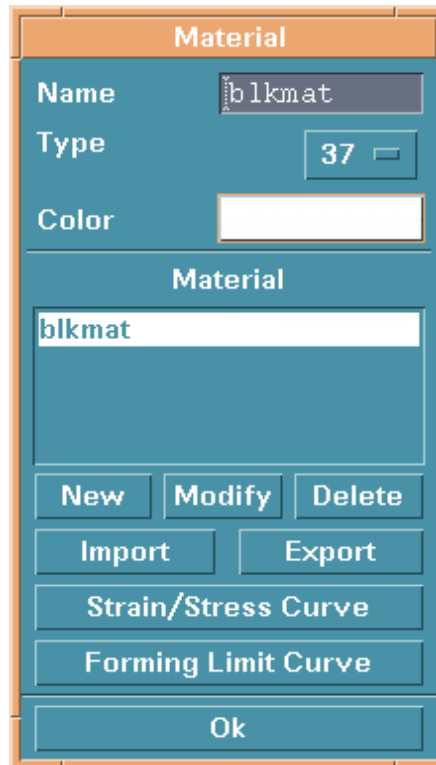
Figure 5.7 Material

5.7.1 CREATE NEW MATERIAL

1. Enter the new name in the NAME field.

2. Select the desired type and color by selecting TYPE and COLOR respectively.

3. Select NEW. A table in which the user may insert or modify the values for the material property appears.

4. Select OK to close the table, the new name will be listed in the window.

Note: **There are seven types of blank materials in eta/DYNAFORM, for detailed descriptions of these materials see the LS-DYNA User Manual.**

5.7.2 MODIFY MATERIAL

1. Select a name in the MATERIAL list window.

2. Select MODIFY and change the values in the Pop-Up window.

5.7.3 DELETE MATERIAL

1. Select a name in the MATERIAL list window.

2. Select DELETE.

5.7.4   EXPORT

Select the EXPORT button on the MATERIAL window, the EXPORT MATERIAL window will appear, see the figure 5.7.1.

Figure 5.7.1 Export Material

Input a name for the material, then select OK, the information about the material will be input the file with suffix mat.

5.7.5   IMPORT

This function allows the user to read in a material from the file with the suffix mat. Once choose a mat file and select OK to close the IMPORT MATERIAL window, the imported material will be displayed in the MATERIAL window.

5.7.6   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, see Figure 5.7.2.
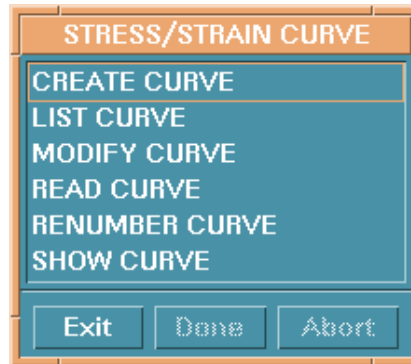
Figure 5.7.2 Stress/Strain Curve

These functions are similar to those in Section 10.9 UTILITIES/LOAD CURVE.

5.7.7   FORMING LIMIT CURVE

The functions in this menu are used to create, read, and display the FLD curve of the materials. The functions are shown in Figure 5.7.3.
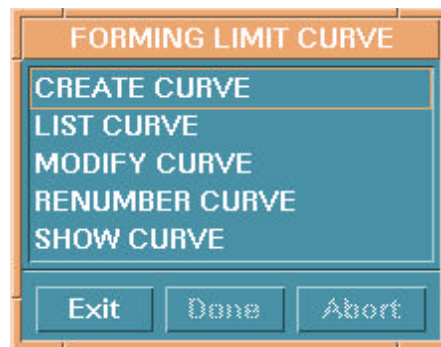
Figure 5.7.3 FLD Curve

These options are similar to those in Section 10.9 UTILITIES/LOAD CURVE.

5.8   PROPERTY

The functions in this menu are designed to define and modify the physical properties of the blank. The dialog window is shown in Figure 5.8.
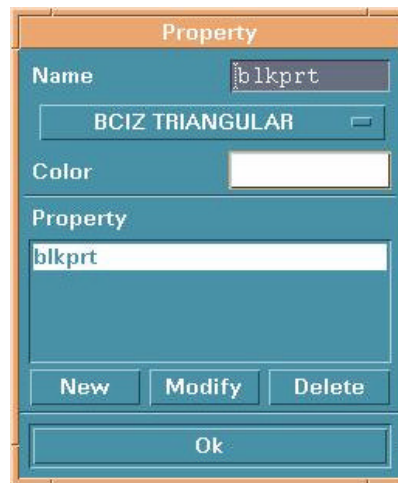
Figure 5.8 Property

5.8.1   NEW

This function creates a new property.

1.   Enter the new property name in the field at the right side of NAME.

2.   Select the button below NAME to choose a property type.

   NOTE: Refer to the *LS-DYNA User's Manual* for the eleven types of shell elements.

3.   Select NEW and a table in which the values of properties can be modified is displayed.

4.   Select OK after modifying the values.

The functions MODIFY and DELETE, are similar to those in Section 5.6 MATERIAL

5.9   ANIMATE

This function is used to show movement of tools having defined motion, see Figure 5.9.1.
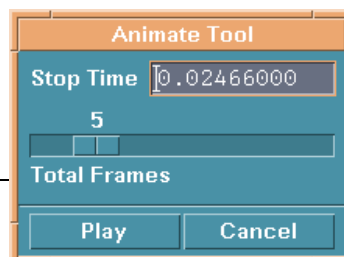
Figure 5.9.1 Animate

5.9.1   PLAY ANIMATION

- STOP TIME is the termination time that is automatically displayed.

- Select the slider to set the number of frames for the animation.

- Select PLAY. The tools with defined velocity will move and a dialog window (Figure 5.9.2) is displayed to allow the user to change the play mode.
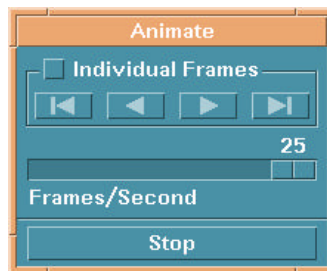
Figure 5.9.2 Animate

5.10   TOOLS ON/OFF

This function allows the user to toggle ON/OFF each individual tool or all tools, see Figure 5.9.

Figure 5.10 Select Tool

5.11   SUMMARY

This function displays statistical information about each tool. First, the SELECT TOOLS dialog window is

displayed.

After a tool is selected, a message window displays a list of tool statistics. See Figure 5.10.
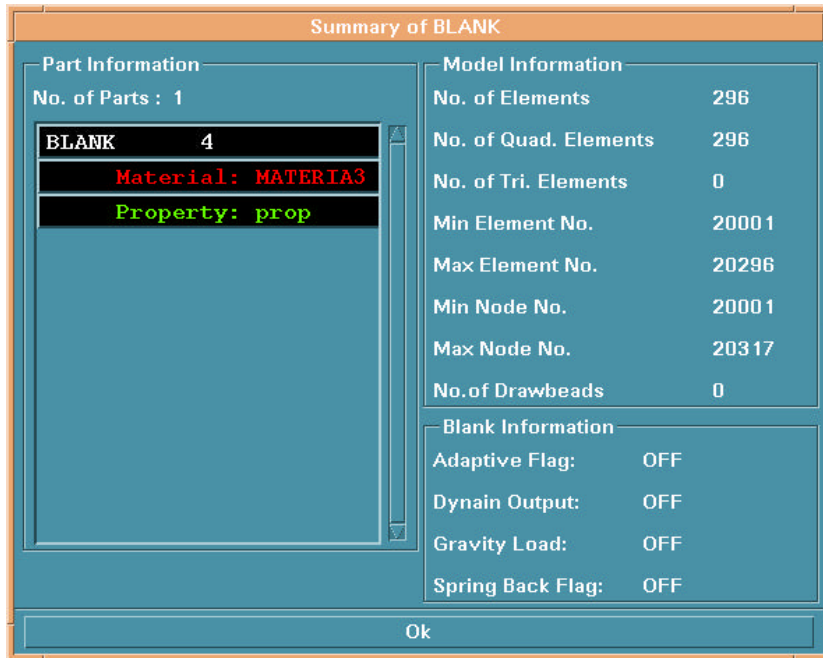
Figure 5.11 Summary Message

CHAPTER 6
PREPROCESS

The functions in this menu allow the user to build and modify a model, generate a model with elements, and also check and add boundary conditions to a model. The submenus are shown in Figure 6.1.

Line/Points	Ctrl+L/F4
Surface	Ctrl+S/F7
Element	Ctrl+E/F2
Node	Ctrl+N/F6
Mesh Repair	Ctrl+R
Model Check	Ctrl+M/F5
Boundary Condition	Ctrl+U
Nodes/Elements Set	Ctrl+V

Figure 6.1 Preprocess Menu

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

Many dialog and submenu windows work the same as or similar to windows that have appeared in previous chapters. References to the window locations will be discussed in the appropriate sections.

6.1   LINE/POINT   (Ctrl + L / F4)

The options in this dialog window are used to build line data. The user can move the mouse point to an icon to see the function name. The options are shown in Figure 6.1.1.

Figure 6.1.1 Line/Point Option

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

6.1.1   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 6.1.2 provides various ways to define the location of the points.

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

● Select NODE or POINT option.

● Select the location of the node or a point. If the POINT option is selected, the point will be created based on which of the following buttons is ON.

, at a point of an existing line near to the cursor.

, at an endpoint of a line.

, at a midpoint of a line.

, at the intersection of other lines.

, at the point on the work plane, see Section 10.12, UTILITIES/WORK PLANE.

Figure 6.1.2 Input Coordinate

2.   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 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.   Cancel the latest created point.

   ●   Select REJECT once.

4.   Finish the definition of the line.

● Select OK or the middle mouse button.

5. Close the dialog window.

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

6.1.2   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 6.1.3.

Figure 6.1.3 Arc Parameter

RAD: Radius of arc.

TH1 :   Beginning angle from U-axis.

TH2 :   Ending angle of 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 6.1.2 and refer to Section 6.1.1. When the third point is selected, the arc and its center will be generated.

6.1.3   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. The spline line will be generated after selecting DONE.

6.1.4   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.

6.1.5   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 needs to complete the options in the COPY LINES dialog window, see Figure 6.1.5.
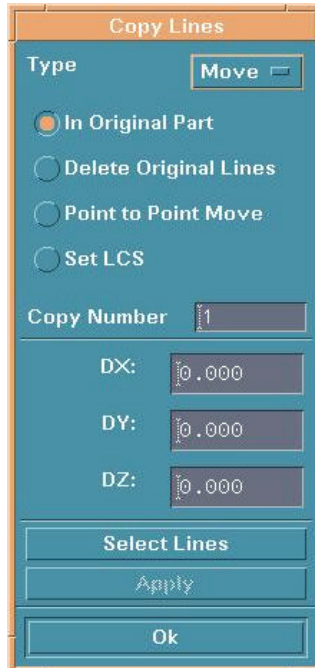
Figure 6.1.5 Copy/Transform Line

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 it 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 will be discussed later in the chapter. This option is only active for the translation.

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

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

9. Select APPLY.

    ● If POINT TO POINT MOVE is selected, the INPUT COORDINATE window, see Figure 6.1.2, 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.

6.1.6   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.

6.1.7   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.

6.1.8   COMBINE LINE

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

● In the SELECT LINE dialog window, select adjacent lines sequentially to form a new line.

● Select OK to combine lines.

6.1.9   SPLIT LINE

The user may 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 6.1.1 for more information.

6.1.10 EXTEND LINE

This function allows the user to extend a line by a specified distance.

● Select a line by mouse pick.

● Select the desired end of the line to extend.

● Enter a real number for the extended length in the next dialog window.

6.1.11   MIRROR LINE

This function allows the user to mirror a set of lines with respect to a given plane. The MIRROR LINES dialog window is displayed, see Figure 6.1.6.
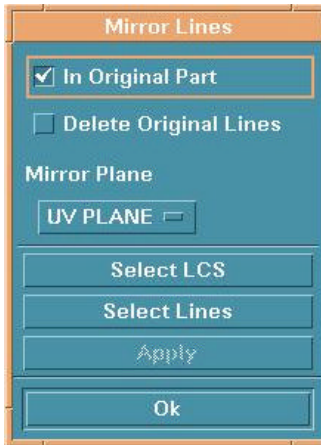
Figure 6.1.6 Mirror Lines

- Select line(s). See Section 4.3.1, PART/ADD LINE for more information.

- Select or deselect the option boxes.

- Select LCS. See Section 2.5, LOCAL COORDINATE SYSTEM for more information.

- Select a coordinate plane as the mirror plane.

- Select APPLY.

6.1.12 OFFSET LINE

This function creates an offset line from an existing line. Usually, it is used to offset a planar line and 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 6.1.7. There is an arrow in the display window for showing the offset direction.

Figure 6.1.7 Dynaform Question

Select YES to accept the direction.

Select NO to turn to the opposite direction.

6.1.13 SCALE LINE

This function allows the user to scale the selected lines.

- Select 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 6.1.8.
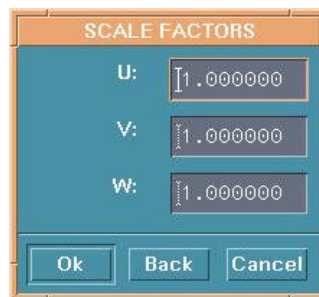

Figure 6.1.8 Scale Factor

6.1.14   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 next dialog window, then select OK.

6.1.15 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.

6.1.16 RESPACE LINE

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

● Select the desired line by cursor.

● Enter any integer greater than two in the next dialog window to determine the number of points in the line.

6.1.17 PROJECT LINE

This function projects lines onto a selected surface.

● Select 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 6.1.10 Project Line/Point

- PROJECT LINE/POINT is displayed, see Figure 6.1.10.

6.1.18 SECTION THROUGH LINE

This function will generate a line consisting of points that are the intersecting 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.

6.1.19 F E. BDY LINE

This function allows the user to generate a line around the boundary of a finite element mesh, see Figure 6.1.11.
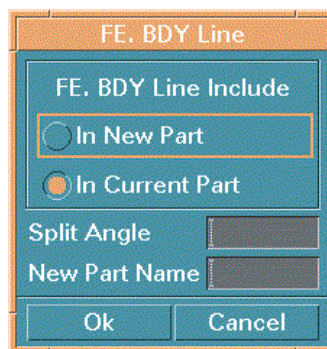

Figure 6.1.11 F E. Body Line

IN NEW PART

This includes the bodyline in a new part. The new name is entered in the field beside NEW PART NAME.

IN CURRENT PART

This includes the bodyline in the current part (default).

SPLIT ANGLE

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

6.2   SURFACE   (Ctrl + S / F7)

The options in this menu allow the user to create and modify surfaces in eta/DYNAFORM. The options are shown in Figure 6.2.

Figure 6.2 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 ON**

    Controls the on/off display of the U-V line on the surface.

- **SURFACE NORMAL ON**

    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 following sections.

6.2.1   CREATE 2L

This function creates 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 6.2.1.

Figure 6.2.1 Control Keys

- LINE allows the user select lines (default).

- LINE SEGMENT -this option allows the user to select several line segments to form a line. DONE should be selected after the line segments are complete.

- POINTS/NODES –this option allows the user to select two points/nodes to form a line. Select the lines, line segments or two points/nodes to form a line that the surface will be created from. After the second line is selected the surface will be generated.

6.2.2   CREATE 3L

This function creates a surface from three selected lines. The procedure is similar to CREATE 2L in

SURFACE.

6.2.3   CREATE 4L ⊞

This function creates a surface from four lines. The procedure is similar to CREATE 2L.

Note: Select line in a clockwise direction or counterclockwise direction. The direction of surface normal dictates the direction of plate element normal.

6.2.4   REVOLUTION SURFACE ⊡

This function creates a surface by rotating a selected line (generatrix) about a vector (the axis of revolution).

Figure 6.2.2 Revolution Angle

- 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 for the generatrix.

- Enter the start and end angles in the dialog window. See Figure 6.2.2. The surface will be generated after the window is closed.

6.2.5   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 6.2.1.

6.2.6   SHOW SURFACE

This function highlights the selected surface and UV lines. There are several ways to show surface, see Figure 6.2.3.
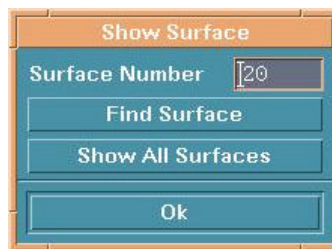

Figure 6.2.3 Show Surface

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

2. Select SHOW ALL SURFACES. All surfaces in active parts will be shown.

3. Select the surface in the graphics directly.

6.2.7   DELETE SURFACE

This function deletes the selected surface from the database. See Section 4.3.3, PART/ADD SURFACE for selecting surfaces.

6.2.8   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.

6.2.9 COPY SURFACE

This function copies 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, see Figure 6.2.4.
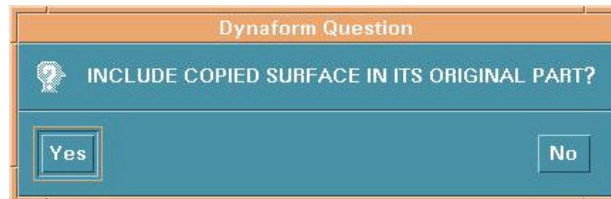

Figure 6.2.4 Copy Question

6.2.10 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.

- After APPLY, a DYNAFORM Question Window appears，select YES/NO to mirror surfaces to ORIGINAL/CURRENT part.

6.2.11 SCALE SURFACE

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

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

- Define a local coordinate system.

- Enter a multiplication 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.

6.2.12 CREATE BOUNDARY LINE

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

6.2.13 CREATE SECTION LINE

This function creates section lines on the selected surface.

- Select surface.

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

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

6.2.14 RESPACE U-V LINE

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

6.2.15 REVERSE NORMAL

This function reverses 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 automatically after a surface is selected.

6.2.16 SURFACE INTERSECT

This function creates a line at the intersection between two selected surfaces.

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

6.2.17 SURFACE SPLIT

This function splits a selected surface into two surfaces at a defined line.

After the user selects a surface, there are eight items in the submenu window. They provide ways to define splitting the line.

1. TWO BOUNDARY POINTS

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

2. SECTION U-V LINE

    Select a U-V line on the desired surface by cursor. The surface can be split at this U-V line.

3. SURFACE INTERSECTION

    Once two surfaces are selected, the first surface splits at the intersection line.

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 creating a new UV grid.

    7.   CURVE IN DATABASE

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

    8.   OVERLAP SURFACE
The function allows the user to pick an overlapping surface on the selected surface and smaller than the selected surface. The first selected surface will be split along the second surface boundary.

6.2.18 TRIM SURFACE

This function allows the user to trim a portion of a selected surface within specified inner and outer boundaries. When the function is selected, the operation is as below.

    1.   Click SELECT SURFACE to select the surface to be trimmed.

    2.   Select a line as the boundary. If there is no line in the current database, click EXIT, the DEFINE TRIM LINES dialog window is displayed, see Figure 6.2.5.

    3.   The next step is to define the trim boundaries.

- Select one of the curve types: line (polygon), circle, spline curve, and intersection line.

- Select CLOSED or OPEN to close the defined line automatically or leave it open.

- Select points on the surface to form a line.
  - If the type is CIRCLE, a circle will be formed and shown after two points are selected. The first point is the center and the distance between the two points is the radius.

- 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 and to show the line.

- If the type is INTERSECTION LINE, no point needs to be selected, but instead the user must choose 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 or selected, DYNAFORM prompts "KEEP THE ORIGINAL OUTER BOUNDARY", see the Figure 6.2.6. Select YES to keep the surface between the original outer boundaries and the defined or selected curve; select NO to keep the surface within the defined or selected curve.
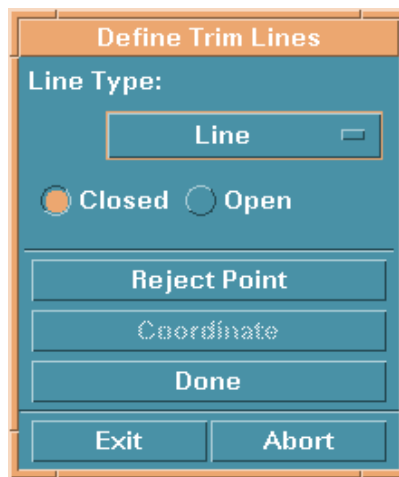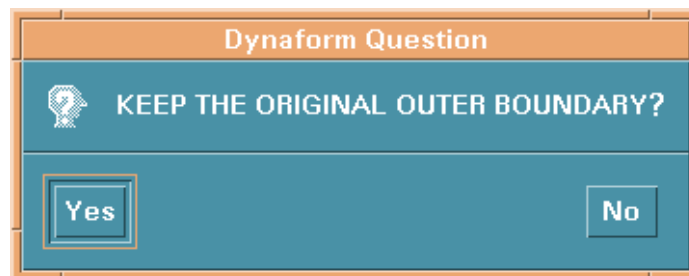

Figure 6.2.5 Define Trim Line

Figure 6.2.6 Dynaform Question Window

6.2.19   REMOVE HOLE

This function allows the user to remove trim holes from a selected surface. The user selects a surface by

using options in the SELECT SURFACE dialog window. The holes on the surface can be removed automatically.

1. Select the REMOVE HOLES icon from the PRE-PROCESS/SURFACE menu.

2. Select the surface that needs to be altered

3. Select the hole of type, either outer boundary hole, or inner hole

4. Select a method for removing the holes, either individually or all holes at once. NOTE: For outer boundary holes, you must define two points, one on either side. Then, Dynaform will fill in the hole.

6.2.20 SKIN SURFACE

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

6.2.21 UNTRIM SURFACE

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

6.2.22 DUPLICATE SURFACE CHECK

This function allows the user to check for duplicate surfaces. Duplicate surfaces are highlighted and can be added to a new part.

6.3   ELEMENT (Ctrl + E/F2)

The ELEMENT OPTIONS menu contains the commands to build elements. The options are shown in Figure 6.3.1.
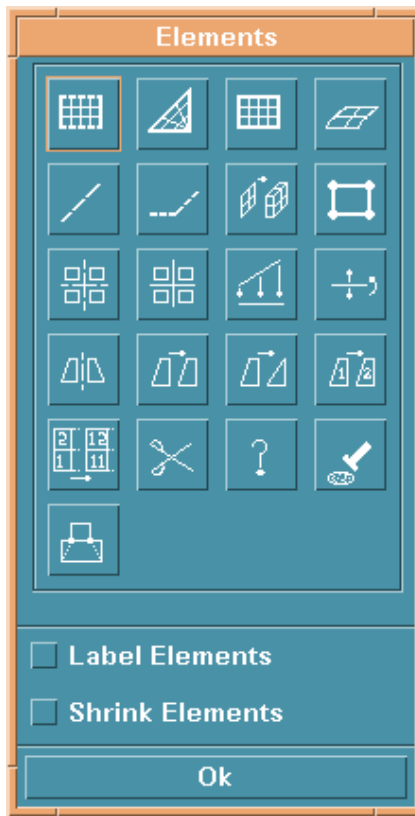
Figure 6.3.1 Element Option

● **LABEL ELEMENT ON**

   Toggles the element label ON/OFF.

● **SHRINK ELEMENT ON**

   Creates a plot with elements reduced in size by 20 percent.

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

6.3.1   2 LINE MESH

The created elements will be included in the current part.

● Select two lines in the submenu window, see Section 6.2.1.

● Enter the position integers in N1, N2, N3, N4 fields in the next dialog window.

The integers N1 and N2 are the number of elements created along line 1 and line 2 respectively. The integers N3 and N4 are the number of elements created along the side between the two selected lines. To create the mesh, those four integers must meet the criteria:

N1 < 2N3, N3 < 2N1, N2 < 2N4, and N4 < 2N2

In the case of that N3 and N4 are omitted, N1 will be the number of elements along N2 between the two lines.

- Select a button in the DYNAFORM Question window, see Figure 6.3.2.

Figure 6.3.2 Dynaform Question

Select YES to accept the mesh and prompt for more lines.

Select NO to not accept the mesh and prompt for more lines.

Select REMESH to reject the mesh and prompts for new N1, N2, N3, and N4.

6.3.2   3 LINE MESH

This function generates elements in an area defined by three lines. After the three lines are selected, the SELECT OPTION window is displayed, see Figure 6.3.3.
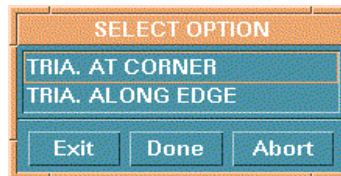

Figure 6.3.3 Select Option

- **TRIANGLE AT CORNER**
  The user needs to input three numbers N1, N2 and N3. There are N1 elements along line1, N2 along line2 and N3 along line3. The triangular elements will be concentrated at the vertex of line 1 and line 2.
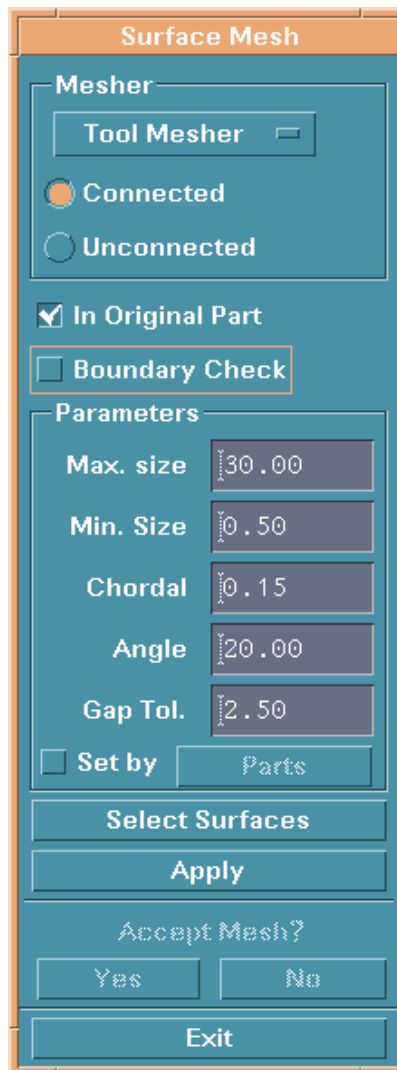
- **TRIANGLE ALONG EDGE**

The user needs to input three numbers N1, N2 and N3. There are N1 elements along line1, N2 along line2 and N3 along line3. All triangular elements will be along the third selected line.

6.3.3   4 LINE MESH

The options in this menu are similar with options in 2 LINE MESH.

6.3.4   SURFACE MESH

This function allows the user to create auto-mesh on the selected surface(s). To create mesh, the user needs to complete the following in Figure 6.3.4.

Figure 6.3.4 Surface Mesh

● Select SELECT SURFACES. See Section 4.3.3, PART/ADD SURFACE for more

information.

- Select a type of tool (connected or disconnected), blank (fixed or variable) or triangle (fixed).

  Connected tool mesh will connect the adjacent surfaces if the gaps are within the GAP TOL.

  Disconnected tool mesh will generate gaps between elements that are located on different and adjacent surfaces

  Fixed blank mesh will generate mesh with fixed element size.

  Variable blank mesh will generate mesh with variable element size on the blank surface geometry.

  Fixed triangle mesh will only generate triangular elements.

- IN ORIGINAL PART toggle: This function includes the created elements in the original part or in the current part.

- BOUNDARY CHECK: This function will check and display the model boundary after building the topological structures.

- Enter numbers of maximum size, minimum size, and chordal ratio in the field.

- Select APPLY TO MESH

If TOOL MESH is selected:

eta/DYNAFORM create mesh on the selected surfaces. If the number of selected surfaces is equal to or greater than 2 and the BOUNDARY CHECK toggle is on, the function first builds topological structures.

- After building the topological structures, the BOUNDARY CHECK window is displayed. Please the Figure 6.3.6.
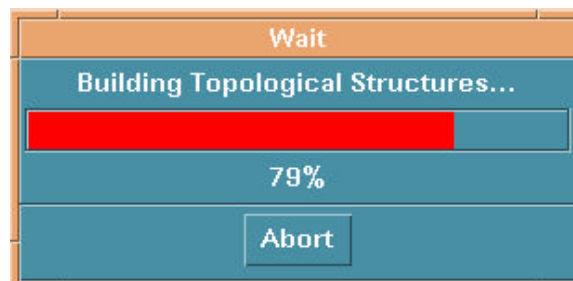

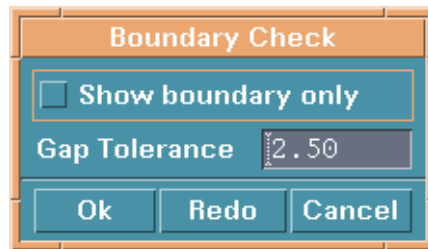Figure 6.3.5 Building Topological Structures

Figure 6.3.6 Boundary Check

Select/deselect SHOW BOUNDARY ONLY to show boundary only/surfaces with model boundary. Select OK to accept the Gap Tolerance, and REDO to input the new GAP TOL. and rebuild the topological structures.

- After selecting the gap tolerance, eta/DYNAFORM begins to mesh the surface boundary, please see Figures 6.6.7, 6.6.8.
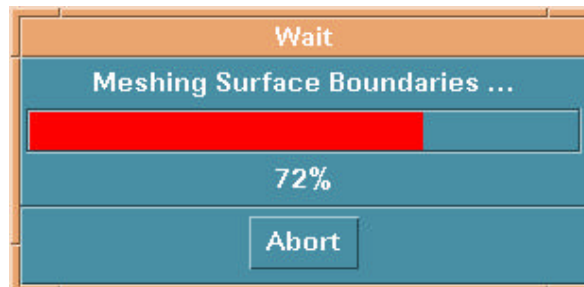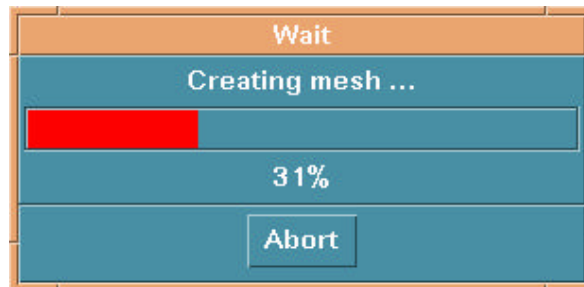
Figure 6.3.7 Meshing Surface Boundaries

Figure 6.6.8 Creating Mesh

- YES and NO 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, user can abort the current status by selecting ABORT.

If BLANK MESH FIXED is selected:

> This function is used for automatically meshing the multiple surfaces defining blank models. The elements generated by this option have conformity along common surface boundaries and are a fixed size.

- The user only enters the size value in the field. Other functions are common with those listed above.
- A mesh quality criteria table will be displayed to allow the user to define the criteria for mesh quality.

If BLANK MESH VARIABLE is selected:

> The user has to enter number of maximum size, minimum size, and ratio. Other functions are common with those in above.

If FIXED TRIANGULAR MESH is selected:

The user can only enter the element size and the gap tolerance.

A mesh quality table will be displayed (Figure 6.3.9) and YES or NO will be activated in the SURFACE MESH window.

Figure 6.3.9 Meshing Message

6.3.5   LINE MESH

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

- Enter the number of elements to be created and then press OK. See Figure 6.3.10.

Figure 6.3.10 Element Type

- Select a line.
- Choose either NODE/POINT or ORIENTATION VECTOR from the dialog window as in Figure 6.3.11.

Figure 6.3.11 Control Keys

- If you select NODE/POINT, a new dialog window will be displayed (Figure 6.3.12). This window prompts you to select the appropriate NODES/POINTS, and also gives you different options for doing so.
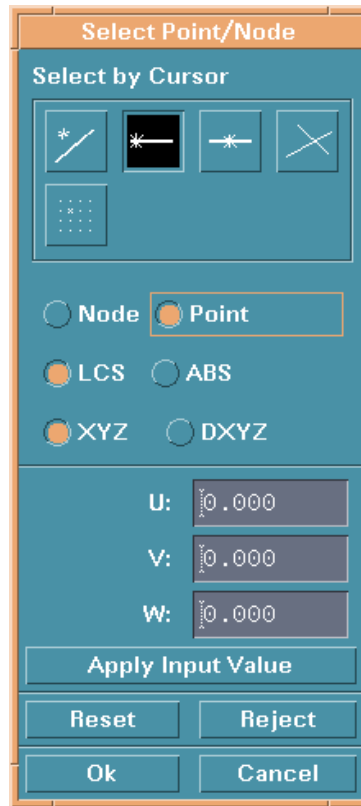
Figure 6.3.12 NODES/POINTS

● Select YES/NO/REMESH to accept/reject/remesh the result.

6.3.6 2 LINE POINT MESH

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

● Select two lines in screen.

● Enter the number of layers between lines.

● Answer the DYNAFORM question.

6.3.7   DRAG MESH

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

Note: **In eta/DYNAFORM, only certain types of elements can be put together in the same parts. For example, if the current part is "PART A" that contains plate (shell) elements, the user can not create any solid elements. eta/DYNAFORM will write a warning in the prompt window if a violation occurs.**

1.   ONE LINE DRAG

   ● Select an element 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, then $n-1$ elements will be generated for each base element.

   ● The user can accept, reject, or remesh.

2.   NORMAL DRAG

   ● Select an element by using options in the SELECT ELEMENT dialog window. Enter thickness of normal drag and number of layers in the Pop-Up dialog windows.

   ● The user can accept, reject, or remesh.

   Note: Plate element normals should be consistent prior to the execution of NORMAL DRAG.

3.   3 OR 4 LINE DRAG

   ● Select an element by using the options in the SELECT ELEMENT dialog window.

   ● Select three or four control lines. The user must select the volume lines in clockwise or counterclockwise order.

   ● Enter number of layers.

   ● The user can accept, reject, or remesh.

6.3.8   CREATE ELEMENT

This menu lists the options used to create elements via the connection of selected nodes or points. The element types supported are those recognized by LS-DYNA. The element types are shown in Figure 6.3.13.

Figure 6.3.13 Element Type

1.   BEAM

    ● Select two points or nodes by cursor in screen.

    ● Define material axis by selecting a third node/point or defining the vector:

                BY NODE

                BY POINT

                BY VECTOR

       Up to three values may be entered in the next dialog window to define a vector in the bar local XY plane. The local Y-axis of the bar 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
      The 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 appears.

    ● **CREATE QUADRILATERAL ELEMENT**

The element is created automatically once four nodes or points are selected in the INPUT COORDINATE dialog window.

- **CREATE TRIANGULAR ELEMENT**

  The user must click DONE after three nodes or points are selected in the INPUT COORDINATE dialog window.

5. SOLID
   This function allows the user to create cube, wedge, and tetrahedron element by using options in the INPUT COORDINATE dialog window.

   - **TETRAHEDRON**

     Select four nodes/points and select DONE. The first three nodes define the base of the tetrahedron. They should follow the right hand rule to point to the fourth node.

   - **PENTAHEDRON**

     Select six nodes/points (pick the second set of three nodes/points in the same order as the first set) and select DONE. 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 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 value of the mass in the field of 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 same order of the first set).

6.3.9 COARSE ELEMENT

This function allows the user to increase the coarseness of 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 desired element is selected, a larger element will be created.

- Select a pair of elements and select DONE in the SELECT ELEMENT dialog window.

- Repeat the previous step if the user wants to coarsen more elements.

- Select DONE in the SELECT ELEMENT dialog window. Each of the selected pairs above will be combined into one element.

6.3.10 SPLIT ELEMENT

This function divides either SHELL or SOLID elements into multiple elements based on the options described below. The options are shown in Figure 6.3.14.
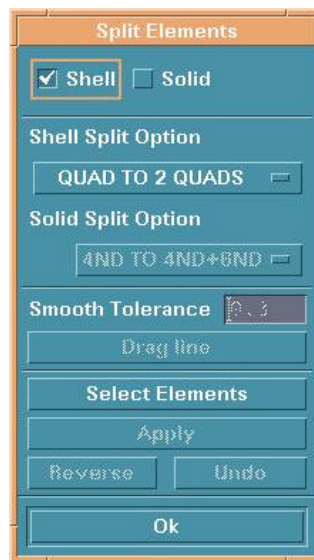

Figure 6.3.14 Split Element

1. SPLIT SHELL

- Select the button beside SHELL to activate the shell options, see Figure 6.3.15.
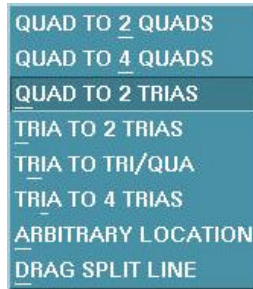
Figure 6.3.15 Shell Split

QUAD TO 2 QUADS
Splits a quadrilateral into two quadrilaterals.

QUAD TO 4 QUADS

Splits a quadrilateral to four quadrilaterals.

QUAD TO 2 TRIAS

Splits a quadrilateral to two triangular elements.

TRIA TO 2 TRIAS

Splits a triangular element to two triangular elements.

TRIA TO TRI/QUA

Splits a triangular element to a triangular and a quadrilateral element.

TRIA TO 4 TRIAS

Splits a triangular element into four triangular elements.

ARBITRARY LOCATION

Select an element first and select two nodes/points arbitrarily on the boundary of the element.

DRAG SPLIT LINE

Select DRAGLINE to drag a line to split several elements. The user can 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 select DONE to go back to the SPLIT ELEMENT dialog window.

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

2. SPLIT SOLID

- Select the button beside SOLID to open the solid options.
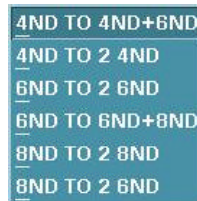
   The options are shown in Figure 6.3.16.

Figure 6.3.16 Solid Split

   These options are used to split four, six or eight nodes of a solid element to two solid elements.

- Other functions are similar with those in SHELL SPLIT.

6.3.11   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 and the element on the source part is projected onto the target part.

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

6.3.12   REVERSE ELEMENT

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

The user selects an element by using the options in the SELECT ELEMENT dialog window. The orientations of the selected elements are reversed contemporaneously.

6.3.13   MIRROR ELEMENT

This function allows the user to generate a mirror image of a selected element. The options are shown in Figure 6.3.13.

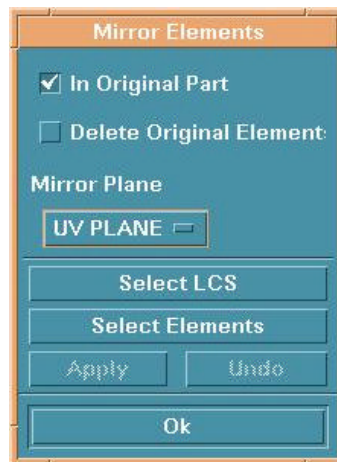For more information on mirror elements, see Section 6.1.17 MIRROR LINES.

Figure 6.3.17 Mirror Element

6.3.14 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 6.3.18. See Section 6.1.5 for COPY and TRANSFORM LINE and/or Section 5.1.7 for MATING TOOLS FOR OFFSETTING.
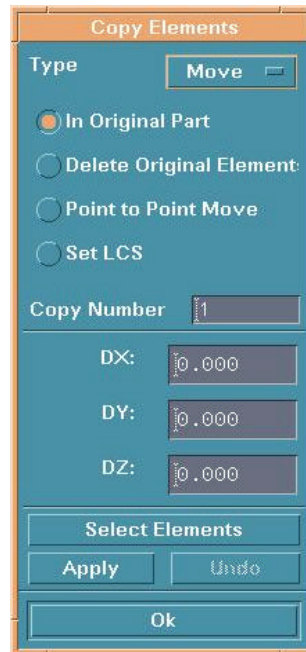
Figure 6.3.18 Copy Element

For MAPPING, the user only defines 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.

6.3.15 MODIFY ELEMENT

This function allows the user to recreate a selected element.

6.3.16 CHANGE NUMBER OF ELEMENT

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

6.3.17 RENUMBER ELEMENT

This function allows the user to renumber all the elements in the database.

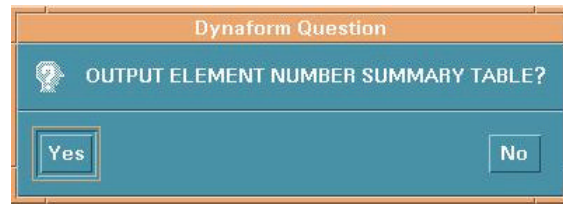   ● Answer the DYNAFORM Question window, see Figure 6.3.19.

Figure 6.3.19 Element Number Table Question

Select YES and the user types a file name. The file is saved in the current directory.

Select NO to not save the table.

BY SEQUENCE

The user inputs the starting number in the next dialog window (default is 1). The numbers of all the elements are changed automatically.

BY PART

A DYNAFORM Question window appears, see Figure 6.3.20.

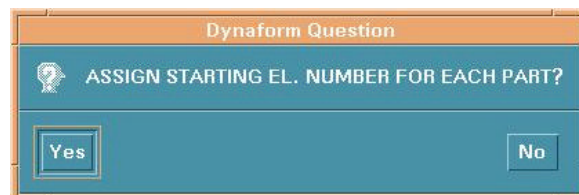
Figure 6.3.20 Starting Element Number Question

1. YES
   The user can enter a number in the input dialog window. eta/DYNAFORM prompts:

   <ENTER STARTING ELEMENT NUMBER FOR PART: xxx>

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

   <ELEM x TO xx ASSIGNED, NEXT ELEM NO: yyyy>

   The next input dialog window appears. This procedure will continue 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).

6.3.18 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 automatically after the user selects OK.

6.3.19 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 and number it's assigned to, and the node numbers that it contains.

6.3.20 FIND ELEMENT

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

- Once the element number has been entered, it will light up on the screen labeled with its node numbers. The node numbers will also be displayed in the prompt window of the screen.

- 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>

6.3.20 MESH REPAIR

This function allows the user to automatically repair bad elements. After selecting elements that need to be repaired, a list of parameters is displayed. See Figure 6.3.21.
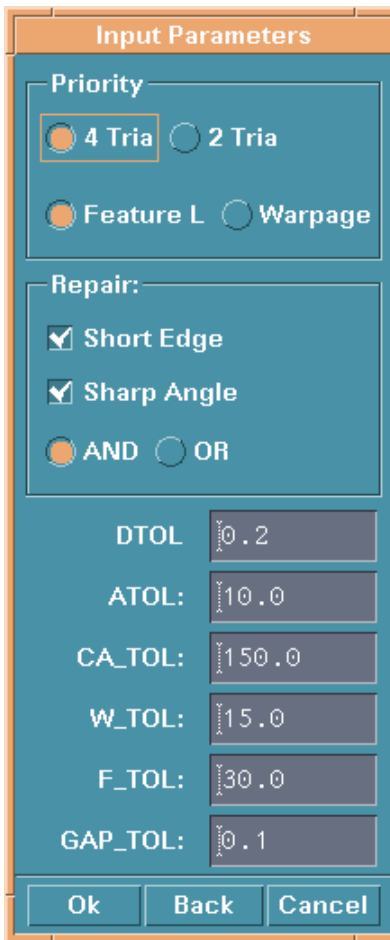
Figure 6.3.21 Mesh Repair

PRIORITY
This option sets a priority to split elements into 4 Triangles or 2 Triangles, and use a Feature line or Warpage as criteria for repair.

REPAIR
The user can select to repair short edges and/or sharp angles according to the following parameters.

PARAMETERS FOR DEGENERATE ELEMENTS
These parameters are the criteria by which degenerate elements are defined. They include Angle Tolerance, Warpage Tolerance, Feature Line Tolerance, Gap Tolerance, etc..

6.4   NODE

In eta/DYNAFORM nodes and points as "points" in a three dimensional space are different objects. Points are used to form lines and surfaces while nodes are used to form elements. There are two types of nodes ; referenced nodes and unreferenced nodes (free nodes). A referenced node is a node that is used by elements and is represented by a dot. A 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 6.4.1.
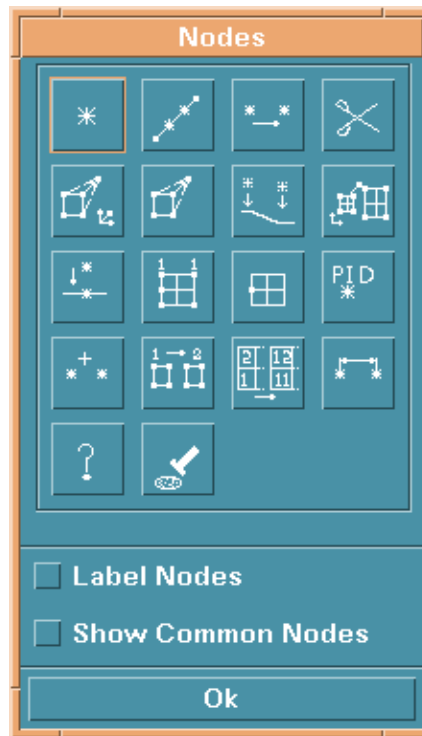

Figure 6.4.1 Node Option

- **LABEL NODES ON**

    Toggle the node labels ON/OFF.

- **SHOW COMMON NODES**

    Finds and toggles the common nodes ON/OFF.

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

6.4.1   CREATE NODE

This function creates unreferenced nodes at points or by entering the coordinates, see 6.1.1. The user can create nodes by using options in the INPUT COORDINATE dialog window.

6.4.2   ADD NODES BETWEEN TWO NODES/POINTS

This function generates nodes that are spaced equally between two existing nodes or points. The new nodes are displayed as unreferenced (free) nodes.

The user selects two nodes/points by using options in the INPUT COORDINATE dialog window. Enter the number of added points in the next dialog window and the points are generated automatically.

6.4.3   COPY NODE

This function generates a new set of unreferenced nodes at a user-specified location.

- Select nodes by using options in the SELECT NODE dialog window see Figure 6.4.2.
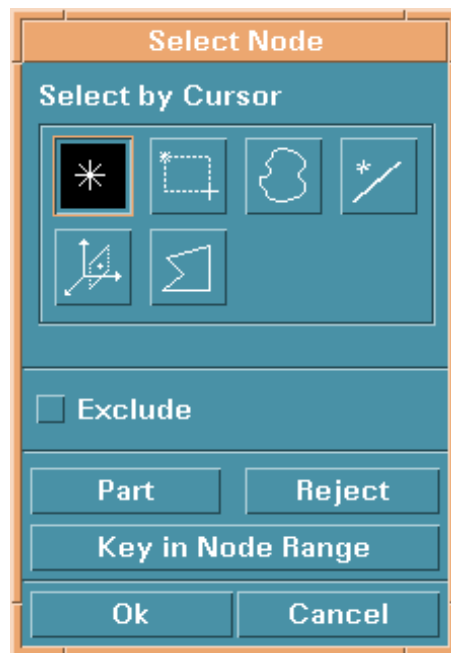

Figure 6.4.2 Select Node

There are several ways to select nodes:

Select node by cursor.
By drag window – select two points in screen to form a window.
By multi-point region – select several points in screen to form a multi-point region.
By free region –hit and hold the left mouse button to draw a circle around the nodes.
Near a line –select a line and input the tolerance.
Near a plane –select a plane and input the location and tolerance.

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

BY NODE RANGE
Select KEY IN NODE RANGE. A dialog window appears for entering starting number, ending number, and increment.

- Select OK after node selection.

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

- Enter copy number in the next dialog window.

There are two copy types:

1. MOVE
   Enter displacements in the next dialog window.

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

6.4.4 DELETE UNREFERENCED NODES

This function allows the user to delete either all or selected unreferenced nodes in the database. The unreferenced nodes are labeled with asterisks (*).

ALL FREE NODES
All unreferenced nodes are deleted automatically.

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

6.4.5 TRANSFORM NODE

This function translates or rotates selected nodes to a new location.

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

- Select MOVE or ROTATE 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 last translation.

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

   3. RESELECT NODE

6.4.6 MOVE NODE

This function moves nodes to any location in the display area.

- The user selects a node by cursor.
  If a coincident node is selected, there is message 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 moved automatically.

- **UNDO LAST**
  It is a valid selection only if a node has been moved.

6.4.7   ALIGN NODE

This function allows users to align the selected nodes along selected lines with a given tolerance.

6.4.8   SCALE NODE

This function moves nodes by scaling their coordinates.

- Select a node using options in the SELECT NODE dialog window.

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

- Enter X, Y, Z scale factor in the next dialog window.

6.4.9   PROJECT NODE

This function projects a node or group of nodes onto a plane, surface(s) or a set of elements.

1. ON F. E. MESH

   - Select 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 node.

   - UNDO LAST is used to reject the last procedure.

2. ON LOCAL UV-PLANE

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

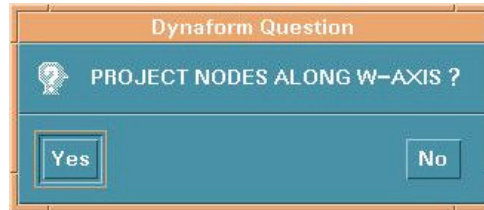- A DYNAFORM Question window is displayed, see Figure 6.4.3.

Figure 6.4.3 Project Node Question

Select YES and the user selects nodes using options in the SELECT NODE dialog.

Select NO and the user can create a new coordinate system and then select nodes.

3. ON SURFACE
    The user selects a surface by using the options in the SELECT SURFACE dialog window.

4. ON SHELL ELEMENT (NORMAL)
    The user selects shell elements by using the options in the SELECT ELEMENTS dialog window. After selecting elements, selected nodes will be projected on the selected shell elements along the shell element normal.

6.4.10 CHECK DUPLICATE NODES

This function renumbers any duplicate node ID numbers found in the eta/DYNAFORM database. Users are unable to create duplicate node numbers from within the program. However, users may import files containing duplicate nodes from other programs. eta/DYNAFORM will renumber the duplicates according to the value of the maximum node number + 1.

If any duplicate nodes are found, the following messages echo on screen and in the DYNAFORM.msg user files located in the work directory:

CHECKING xxxx NODES FOR DUPLICATE NODES

DUPLICATE NODE xxxx CHANGED TO yyyy

xxxx DUPLICATE NODES FOUND

If no duplicate nodes are found, the following messages echo on screen and in the DYNAFORM.msg user files located in the work directory:

CHECKING xxxx NODES FOR DUPLICATE NODES

NO DUPLICATE NODE FOUND

6.4.11 CHECK COINCIDENT NODES ⊞

This option checks and merges coincident nodes that may be in the display. Coincident nodes are defined as having distance between each other less than or equal to a specified tolerance.
eta/DYNAFORM merges all coincident nodes into the one having 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.

- Answer YES to merge the incident nodes.

6.4.12 PART CONNECT

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

6.4.13 COMPACT NODE

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

- Delete nodes, see Section 6.4.4.

- Enter a number as the starting number for the remaining nodes.

6.4.14 CHANGE NODE NUMBER

This function changes the ID numbers of the existing nodes.

1. NODE
   Select node by cursor.

2. KEY IN NODE NO.
   Enter 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 is already used, there is message in the prompt window:
   NODE NUMBER xxx ALREADY EXISTS, REQUEST DENIED

   ENTER NEW NODE NUMBER FOR xxx

4. UNDO LAST
   Reject last selection.

6.4.15 RENUMBER NODES

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

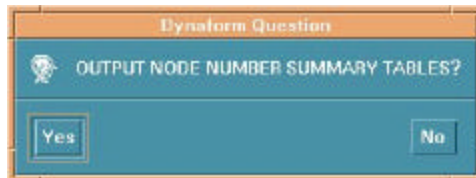- A DYNAFORM Question window appears, see Figure 6.4.4.

Figure 6.4.4 Node Number Table Question

Select YES and the user types a file name. The table is saved in current directory.

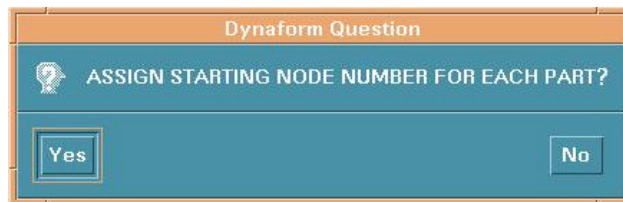Select NO to not save the table.

- Another DYNAFORM Question window appears, see Figure 6.4.5.

Figure 6.4.5 Starting Node Number Question

1. YES
    eta/DYNAFORM prompts:
    ENTER STARTING NODE NO. FOR PART? xxxx

    Once a number has been 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 gives a message that the location 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

6.4.16 DISTANCE BETWEEN NODES/POINTS

This function calculates 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.

6.4.17 IDENTIFY NODE/POINT

This function identifies 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.

6.4.18 FIND NODE

This function finds the location and coordinates of a node specified by the input ID number.

6.5   MESH REPAIR (Ctrl + R)

This submenu provides a convenient set of functions for repairing mesh. It integrates the most commonly used functions from the NODE, ELEMENT and MODEL CHECK menus. Please see the Figure 6.5.1.
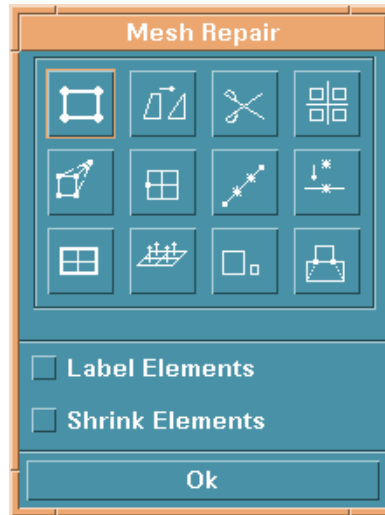

Figure 6.5.1 Mesh Repair Menu

6.6   MODEL CHECK (Ctrl + M/F5)

The functions in the MODEL CHECK menu validate the eta/DYNAFORM models to meet specified criteria. The options are shown in Figure 6.6.1.
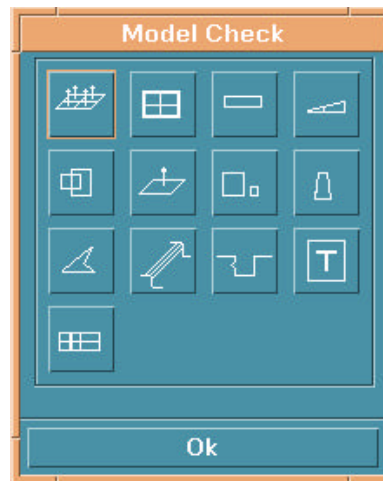

Figure 6.6.1 Model Check

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

6.6.1   AUTO PLATE NORMAL

This function converts the normal direction of all elements in the selected part(s) to the specified direction.

- There are two options to select elements: ALL ACTIVE PARTS and CURSOR PICK PART. The selected elements will display their current normal direction.

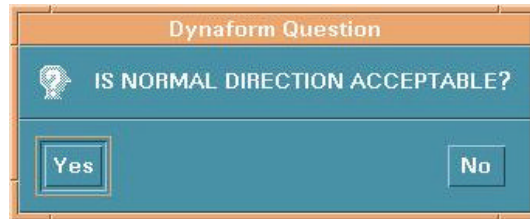- A DYNAFORM Question window appears, see Figure 6.6.2.

Figure 6.6.2 Plate Normal Question

Select YES and all selected element normals will be made consistent with referenced element normal.

Select NO and the reference element normal will be reversed. All selected element normals will be made consistent with the reference direction.

Note: **We suggest selecting one part at a time. Parts that branch off or are separated do not always get adjusted properly.**

6.6.2   BOUNDARY DISPLAY

This function checks the boundary of displayed part(s). It is used to check the proper corrections. Please see Figure 6.6.5.

1. Select SINGLE SURFACE or MULTIPLE SURFACE.

- **SINGLE SURFACE**
  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.
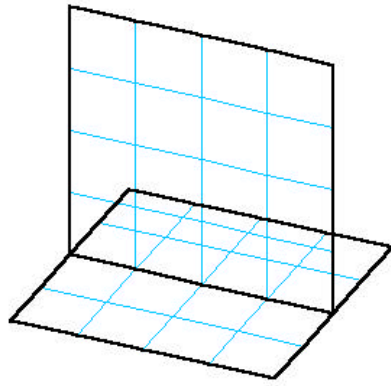
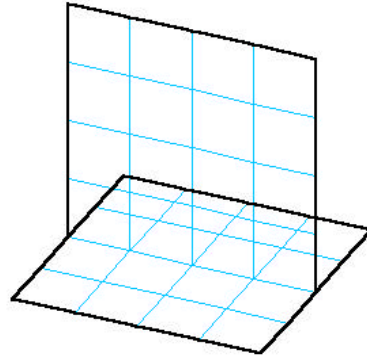Figure 6.6.3 Single Surface   Figure 6.6.4 Multiple Surface

- **MULTIPLE SURFACE**
  Surfaces that have been connected improperly will be highlighted where the elements intersect, as do the boundaries of each surface. For example, if three plate elements share a common edge the, the common edge will not be displayed as a boundary.

2. DYNAFORM prompts the user to create plotel elements for the free edges. If YES is selected DYNAFORM prompts the user to put the elements in a new part.
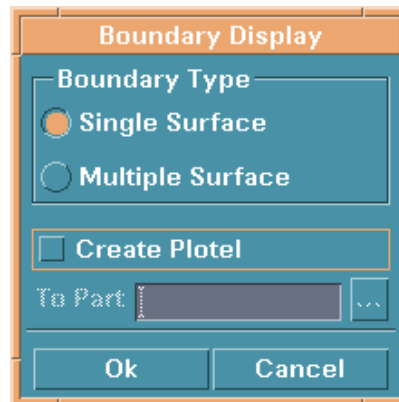

Figure 6.6.5 Boundary Display

DEFINE PART MATERIAL
See Section 5.4, TOOL/MATERIAL.

DEFINE PART PROPERTY
See Section 5.5, TOOL/PROPERTY.

Note: **The boundary will remain highlighted until the user selects CLEAR.**

6.6.3   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, but the user may adjust it as necessary.

1. Users may accept the default value of 8.0 or they may enter any 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 desired 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 6.6.6.
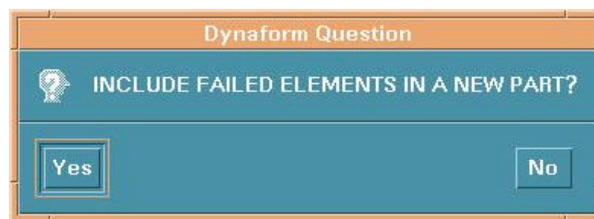

Figure 6.6.6 Include Failed Element Question

Select YES and the CREATE PART dialog window appears. Options are common with BOUNDARY DISPLAY.

Select NO to skip to the next function.

3. eta/DYNAFORM prompts:
                    SELECT ELEMENT FOR ASPECT RATIO

    The user can select any displayed element and its value will be shown in the prompt window.

6.6.4   INTERIOR ANGLE CHECK

This function checks the minimum values of the interior angles for shell and solid elements. eta/DYNAFORM checks and highlights any interior angles that are less than the specified values. Users have the option to adjust the default values as necessary.

1.   Enter interior angle in the next dialog window (DEFAULT = 30 DEGREES). If the elements
     pass the interior angle check, eta/DYNAFORM echoes the following message and returns the
     user to the MODEL CHECK menu:
                              ALL ELEMENTS PASS CHECK!

     If all of the elements fail the model check, eta/DYNAFORM displays the following message
     with the option to create a new part.
                              xxxx ELEMENTS FAILED CHECK

2.   The user can create a new part to include these failed elements.

3.   The user can select any element and its value will be given in the prompt window.

6.6.5   OVERLAP ELEMENT

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

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

6.6.6   PLATE NORMAL

This function draws a boundary line between elements with opposite normal directions.

If there are not any opposing plate normals, eta/DYNAFORM prompts:
                    NORMAL CHECK COMPLETED, NORMAL IS CONSISTENT

6.6.7   ELEMENT SIZE

This function checks 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 and the
user can change the value.

●   The user can create a new part to include the failed elements.

●   The user can select any element and its value will be given in the prompt window.

●   Degenerate the elements option available.

6.6.8   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 create a new part to include the failed elements.

● The user can select any element and its value will be given in the prompt window.

6.6.9   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 warpage criteria (default is 5.0 degrees).

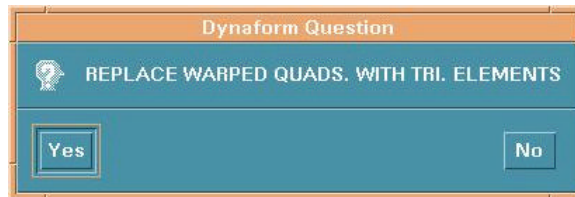2.   If there are any failed elements, a DYNAFORM Question appears. See Figure 6.6.7.

Figure 6.6.7   Replace Warpage Question

Select YES to replace warped quadrilateral with triangular element.

Select NO to reject.

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

4.   The user can select any element and its value will be given in the prompt window.

6.6.10   FEATURE LINE ▦

This function allows the user to check the angle between plate elements based on user specified angle criteria. The user can also generate beam elements at these locations.

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

2.   Any feature line where the angle is larger or equal to the angle criteria will be highlighted. eta/DYNAFORM prompts to generate the plotel element, see Figure 6.6.3.

3.   If the user selects YES, eta/DYNAFORM will prompt to include the element in new part, see Figure 6.6.6.

6.6.11 DIE LOCK

This function verifies that the construction and size of the die at any given point will not limit the punch's motion. It also allows the user to check for undercut in the tooling. The user has the option to add the failed elements to a new part.

6.6.12 TIME STEP

This function will display the 10 smallest blank elements that set the minimum time step.

6.6.13 SECTION CUT

There are several functions in this menu, see Figure 6.6.8.

Figure 6.6.8 Section Cut Option

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.

    ● Create the 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 desired line by cursor. The entire length of this section line will be calculated and
      given in the prompt window.

    ● **REJECT LAST**
      Reject last selection.

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

- Select desired line by cursor.

- Select two points on the line to define 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 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 just defined. 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 come back ON.

6.7   BOUNDARY CONDITIONS   (Ctrl + U)

The functions of the BOUNDARY CONDITIONS MENU create and verify constraints and loads on finite element models. The functions in the BOUNDARY CONDITIONS MENU are shown in Figure 6.7.1.
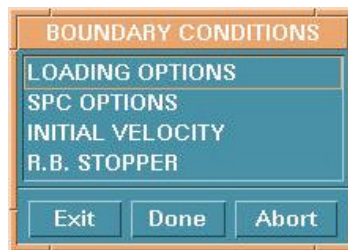
Figure 6.7.1 Boundary Condition

6.7.1   LOADING OPTIONS

The LOAD SET dialog window displays, see Figure 6.7.2.
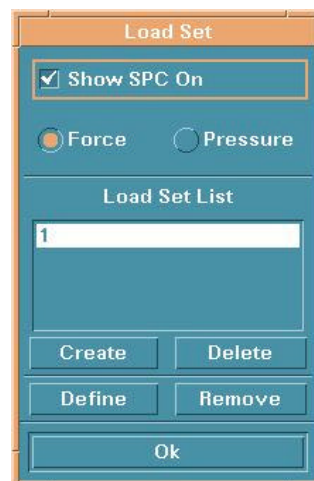
Figure 6.7.2 Load Set

SHOW SPC ON
Toggles SPC (single point constraint) ON/OFF, see SPC OPTIONS.

1.   CREATE FORCE on selected nodes.

●   Toggle FORCE on.

●   Select CREATE.

●   Enter load set number in the next dialog window.

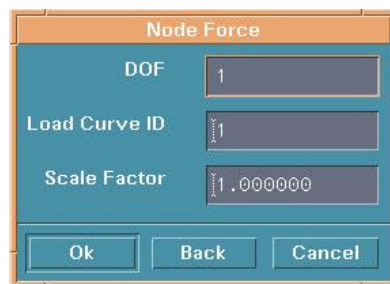●   Define node force see Figure 6.7.3.

Figure 6.7.3 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 motion value versus time, see Section 10.9.1 UTILITIES/LOAD
CURVE/CREATE CURVE.

SCALE FACTOR
Load curve scale factor.

●   Select OK in Figure 6.7.3 to select the nodes to load. See Figure 6.4.2 for selecting nodes.

●   Select CANCEL in Figure 6.7.3 to exit.

2.   CREATE PRESSURE on selected elements.

●   Toggle PRESSURE on.

●   Select CREATE.

- Enter load set number in the next dialog window.

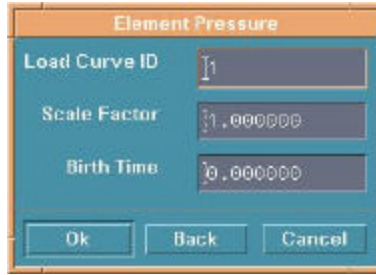- Define element pressure see Figure 6.7.4.

Figure 6.7.4 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 6.7.4 to select element to load.

- Select CANCEL in Figure 6.7.4 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 6.7.5.

Figure 6.7.5 Delete Load Set Question

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.

6.7.2   SPC OPTION

This function allows the user to create and/or modify the SPC options. The SPC SET dialog window appears, see Figure 6.7.6.
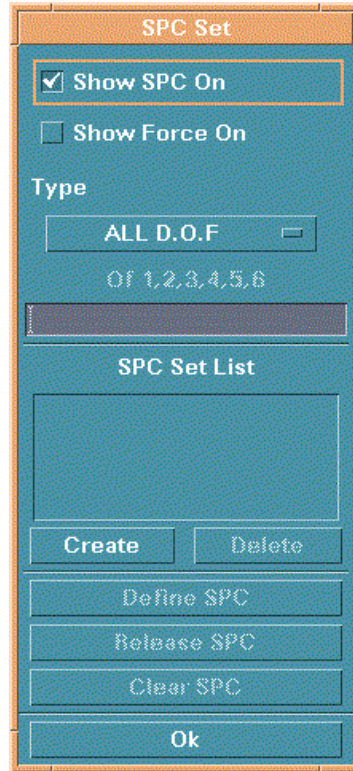

Figure 6.7.6 SPC Set

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

SHOW FORCE ON
Toggles 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:
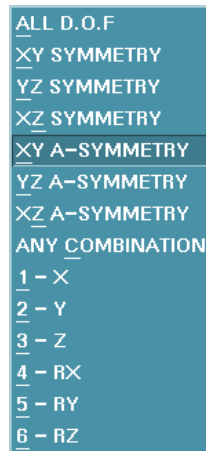

Figure 6.7.7 SPC Type

- 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 delete in the DYNAFORM Question window (Figure 6.7.8).
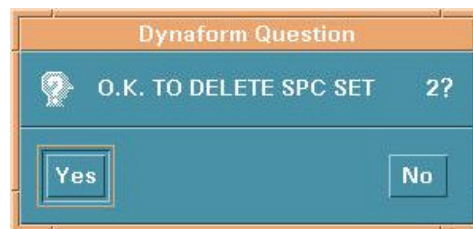

Figure 6.7.8 Delete SPC Set Question

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
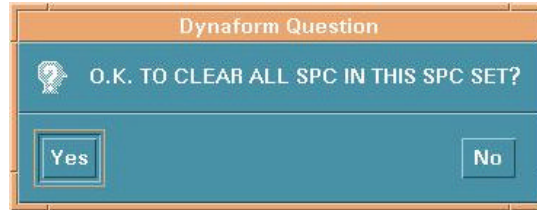Remove all nodes from a selected SPC set. A DYNAFORM Question window is displayed, see Figure 6.7.9.

Figure 6.7.9 Clear SPC Set Question

Select YES to delete all SPC.

Select NO to reject the choice.

6.7.3   INITIAL VELOCITY

This function allows the user to define and assign linear or angular velocities to selected nodes. See Figure 6.7.10.

Figure 6.7.10 Velocity Set

SHOW SPC ON
See SPC OPTION.

SHOW VELOCITY ON
Toggles defined velocity set ON/OFF.

DEFINE
Create velocity set.

- Select DEFINE.

- Enter the initial velocity in the INTIAL VELOCITY dialog window, Figure 6.7.11.
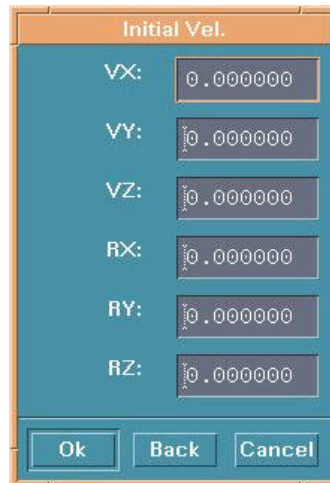
- Select nodes.

Figure 6.7.11 Velocity Define Table

REMOVE
Remove some defined nodes from selected velocity set.

CLEAR
Remove all nodes from the selected velocity set. A DYNAFORM Question window appears, see Figure 6.7.12.
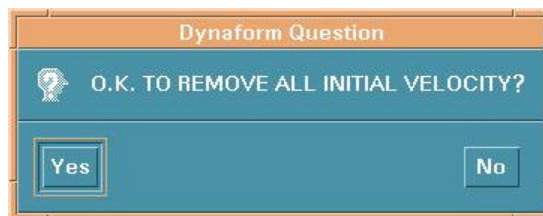

Figure 6.7.12 Remove Velocity Set Question

LIST
There are two choices:

1. ALL NODE
   All nodes with an initial velocity are highlighted with an arrow.

2. NODE
   The selected initial velocities of nodes are listed in the prompt window.

6.7.4   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 metal forming. For a detailed description, refer to the LS-DYNA User's Manual. The options are shown in Figure 6.7.13.

Figure 6.7.13 Rigid Body Stopper

- Enter the value in the field to define the distance from the upper stopper (or lower stopper) to the master part.

- Enter the value in the field to define the maximum velocity of the stopper.

- Enter the value in the field to define the begin time and the end time of the stopper working.

- 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, see Figure 6.7.14.

Figure 6.7.14 Delete Stopper Question

- After changing the UP or DOWN value, select MODIFY. The user can change the position of the rigid body stopper.

- Select SHOW STOPPER ON window to show a stopper in the list.

6.8   NODE/ELEMENT SET (Ctrl + V)

This function creates node and element sets to facilitate the organization of output control data. The options are shown in Figure 6.8.1.

Figure 6.8.1 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 highlighted in the list.

CHAPTER 7
ANALYSIS

The functions in this menu allow the user to run a complete analysis or generate an input file. The options are shown in Figure 7.1.

Figure 7.1 Analysis

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

7.1   ANALYSIS

This function is used to define the parameters for output.

7.1.1   ANALYSIS WITH ORDINARY DRAW TYPE

If the draw type is one of Single Action, Double Action and User Define, the options are shown in Figure 7.1.1.

Figure 7.1.1 LS-DYNA Analysis

● **ANALYSIS TYPE**

There are two types of analysis output the user may select:

1. LS-DYNA INPUT FILE
This function outputs analysis results as an LS-DYNA input file.

2. FULL RUN DYNA
This function outputs an LS-DYNA input file and submits it to LS-DYNA, which will run in the background.

● **CONTROL PARAMETERS**

This function is used to define control parameters. A parameter table appears.

Figure 7.1.2 Dyna3D Control Parameter

OK - Submits the control parameters as they are in the edit fields.

ADVANCED - Activates advanced parameters.

DEFAULT – Returns all values to their default settings.

RESET - Resets the last redefined setting to its previously defined value.

CANCEL - Aborts the table. All values are reset to their previous setting.

Note: Refer to *LS-DYNA User's Manual* for more information.

- **GRAVITY LOAD (toggle)**
  This function toggles gravity loading ON/OFF.

- **DYNAIN OUTPUT (toggle)**
  This function will automatically write out the INTERFACE_SPRINGBACK_DYNA3D card to the input deck. Therefore, LS-DYNA will create a DYNAIN file when it completes the running job.

- **SEAMLESS (toggle)**
  This function toggles the LS-DYNA seamless explicit/implicit switch on/off. This function can directly handle trimming and spring back when running the job.
  If the SEAMLESS switch is on, the user can modify the default implicit parameters.

- **ADAPTIVE MESH (toggle)**
  If this function is toggled on, ADAPTIVE PARAMETERS is activated. The user can define the parameters by selecting the button and the parameter table is displayed, see Figure 7.1.4. Refer to CONTROL PARAMETERS, Figure 7.16, for a description of the function buttons at the bottom of the window.

Note: Refer to the *LS-DYNA User Manual*, keyword *control-adaptive*, for detailed information about each parameter.
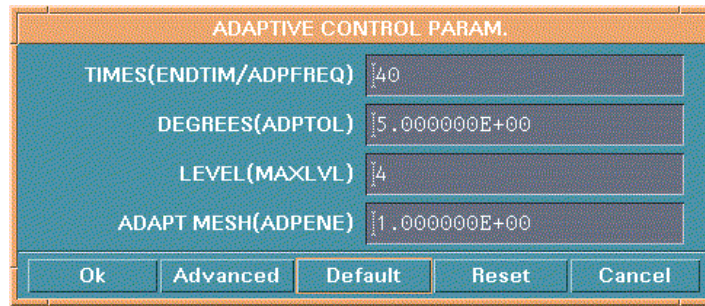
Figure 7.1.3 Adaptive Control Parameter

● **DEFINED TOOLS ONLY (toggle)**

   If ON, only parts defined as tools will be output.

   If OFF, all parts in the database will be output.

   • 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 that the user can select the file name from.

   • 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.

7.1.2   GRAIVTY ONLY ANALYSIS

Gravity Only Load analysis is controlled by the ANALYSIS PARAMETERS function in the SETUP menu. The GRAVITY ONLY option in DRAW TYPE should be activated before analysis. Click the ANALYSIS function in the menu. The dialog box is displayed as Figure 7.1.4.

Note: Reference *LS-DYNA User's Manual* **for more information.**

Figure 7.1.4 Gravity Only Analysis

7.1.3   SPRING BACK ANALYSIS

The functions in the SPRING BACK menu allow the user to use the LS-DYNA implicit simulation for Spring Back Analysis, which is controlled by the ANALYSIS PARAMETERS function in the SETUP menu. The SPRING BACK option in DRAW TYPE should be activated before analysis. Click the ANALYSIS function in the menu. A dialog box is displayed as Figure 7.1.5. The user is required to set up Control Parameters and Implicit Parameters. Several implicit simulation types are provided as shown in Figure 7.1.6.

Note: **Reference *LS-DYNA User's Manual* for more information.**

Figure 7.1.5 Spring Back Analysis

Figure 7.1.6 Implicit Parameter Types

7.2   OUTPUT NEW DYNAIN FILE

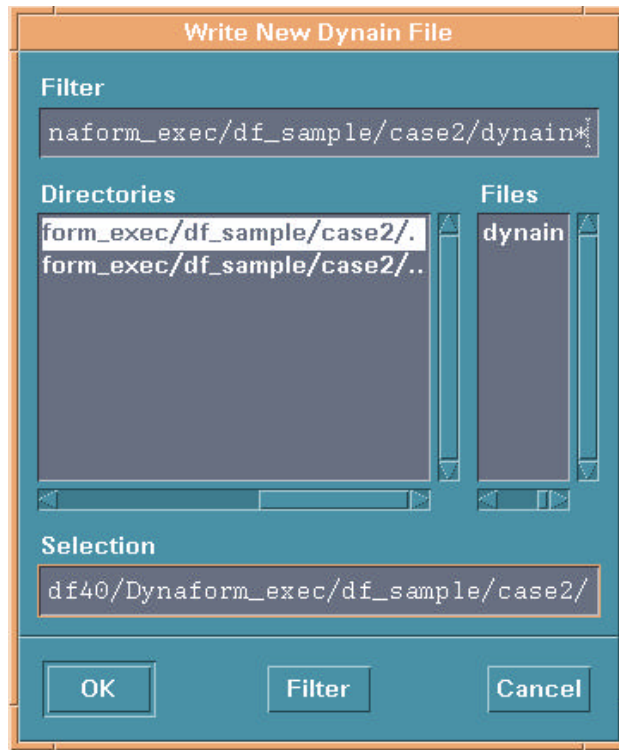This function allows the user to output a new DYNAIN file. See Figure 7.2.1.

Figure 7.2.1 Output New Dynain File

CHAPTER 8
POSTPROCESS

The functions in the POSTPROCESS menu allow the user to graphically display the LS-DYNA result.

Animate Contour	Ctrl+C
Plot Contour	Ctrl+Shift+C
Animate Deformation	Ctrl+D
Plot Deformation	Ctrl+Shift+D
Animate FLD	Ctrl+F
Plot FLD	Ctrl+Shift+F
Vector	Ctrl+Shift+V
Blank Outline	
Mirror Result By ...	▷
Reverse Title	
Summary	
Exit Postprocess	Ctrl+Shift+X

Figure 8.1 Postprocess

Note: To load LS-DYNA result files and to activate the post process menu, see FILE/OPEN in Chapter 3 FILE MANAGER.

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

8.1   ANIMATE CONTOUR (Ctrl + C)

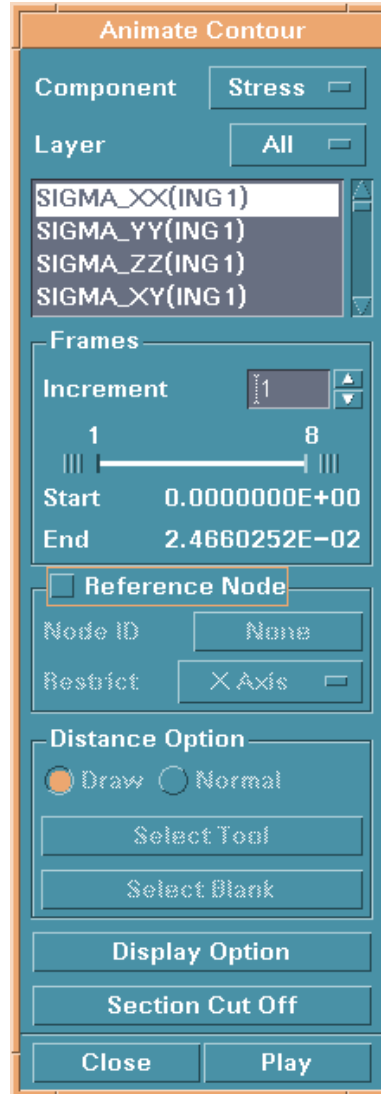The functions in this menu animate the transient analysis results in real time. See Figure 8.1.1.

Figure 8.1.1 Animate Contour

8.1.1   SELECT COMPONENT

The user can select the button to the right of COMPONENT and select a component type to be displayed (stress, strain, displacement, etc). The corresponding components will be listed in the field below.

8.1.2   SELECT A LAYER

Refer to NIP, number of integration point, in LS-DYNA menu.

8.1.3   FRAMES

This function allows the user to select the animation frames to be displayed.

- The user can select the arrow button [icon] to set the increment or enter a number by key.

- Drag the two sliders to set the start and end frames of animation.

8.1.4   REFERENCE NODE

This function allows the user to fix a selected node in a specified direction during animation. All other nodes will move relative to the node's restricted direction.

- Select the box REFERENCE NODE.

- Select the button beside NODE ID to select a node by cursor.

- Select the button beside RESTRICT to select a restricted direction, see Figure 8.1.2.

Figure 8.1.2 Restrict

8.1.5   DISTANCE OPTION

This function is used to display, animate and list the distance between the blank and a tool. The component type is set to OTHER and BLANK TOOL DISTANCE should be selected in the field below. After selection, the following functions in DISTANCE OPTION can be used.

- **DRAW**
  Measure the distance by the direction of the stamping.

- **NORMAL**
  Measure the distance by the direction of the elements normal.

- **SELECT TOOL**
  Select a part by SELECT PART OPTION as the tool.

● **SELECT BLANK**
Select a part by SELECT PART OPTION as the blank.

8.1.6   DISPLAY OPTION

The DISPLAY OPTION dialog window appears once selected, see Figure 8.1.3. The options in this menu allow the user to select what will be displayed or animated.
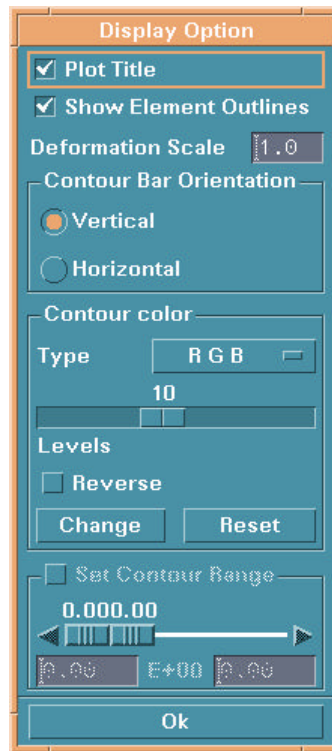

Figure 8.1.3 Display Option

1.   PLOT TITLE (toggle)
     The title of the result file is displayed in the top left corner of the display window. This option allows the user to toggle the title on/off.

2.   SHOW ELEMENT OUTLINES (toggle)
     When post processing the stress results, eta/DYNAFORM displays the outline of the elements in white. The stress results can be viewed without the outline by toggling off the SHOW ELEMENT OUTLINE.

3.   CONTOUR BAR ORIENTATION
     This function allows the user to display the contour bar in either a vertical position on the right hand of the screen or a horizontal position at the top of the screen.

4.   DEFORMATION SCALE
     This function adjusts the default scale factor of the deformed shape plot. The default is the maximum deflection scaled to 7.5% of the largest dimension of the screen size. eta/DYNAFORM will allow the user to adjust the scale factors of either the deformed shape

plot or the deformed shape animation (linear static or non-linear transient).

5.   CONTOUR COLOR
     This function defines contour level and the color of each contour level.

   ● Select the button beside TYPE to select a color type. There are five types of color:

         RG (red, green)
         RGB (red, green, blue)
         RGBM (red, green, blue, magenta)
         GRAY SCALE
         CONTINUE (CHANGE and RESET are inactive in this type)

   ● Drag the slider to set the number of the contour levels used in the contour plots. This
     function displays a user-specified contour value range in contour plot. The program
     defaults to 10 contour levels. The maximum display value for the plot is the continuous
     238 colors and the minimum display value is 2.

   ● Select CHANGE and the color bar will appear on screen, eta/DYNAFORM prompts:
         PICK THE TARGET COLOR FROM COLOR BAR

     After a color is picked, another color bar appears. eta/DYNAFORM prompts:
         PICK NEW COLOR FROM SECOND COLOR BAR

     After the second color is selected, the first color is replaced by the second color.

   ● Select RESET and the program returns the contour level to the default setting.

   ● Toggle REVERSE on/off to reverse the maximum and minimum color on the contour
     plot.

6.   SET CONTOUR RANGE

     This function allows the user to set the range of the contour with the slider provided.

8.1.7   SECTION CUT

This function cuts the model with a plane to expose the inside of the model during the post processing
operation. After a plane is defined, the title of the button is changed to SECTION CUT ON. The option is
shown in Figure 8.1.4. After defining the section, the option is shown in Figure 8.1.5.
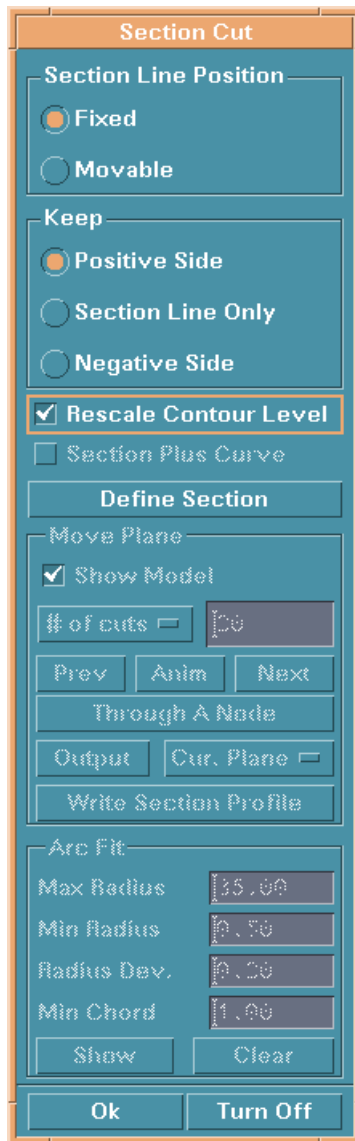
Figure 8.1.4 Section Cut
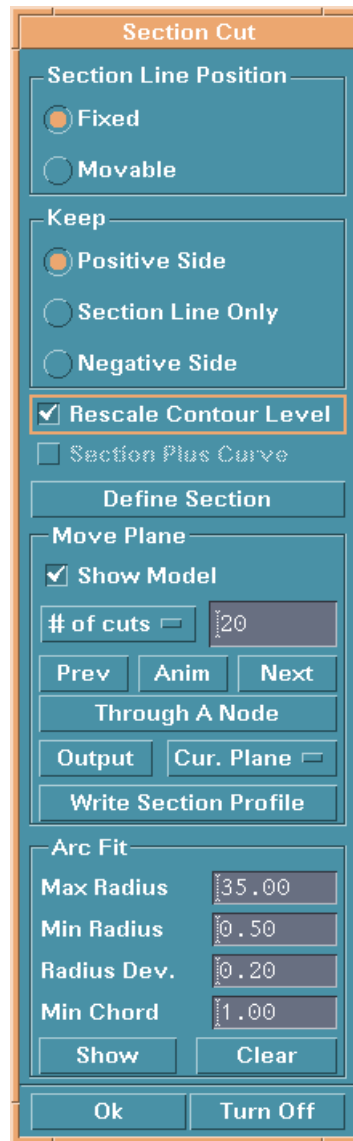

Figure 8.1.5 Section Cut On

1. SECTION LINE POSITION
    - **FIXED**
      The section plane is fixed in space.

    - **MOVABLE**
      The section plane moves with the model.

2. KEEP
    - **POSITIVE SIDE**
      Displays the model on the positive side of the local W-axis.

    - **NEGATIVE SIDE**

Displays the model on the negative side of the local W-axis.

- ● **SECTION CUT ONLY**
  Only the section cut line will be displayed.

3. RESCALE CONTOUR LEVEL
    Rescale the contour levels based on the elements displayed using the SECTION CUT feature.

4. SECTION PLUS CURVE
    Displays a curve along with the section plane.

5. DEFINE SECTION
    The LCS window is displayed for defining a coordinate system. The section plane will be perpendicular to the W-axis. After the offset is entered (offset is the distance between origin and section plane), the section line will be created in white color.

    Note: When SECTION CUT ONLY and SECTION PLUS CURVE are toggled on, PLOT CONTOUR will plot a curve to display the results of the section.

After define the section cut, the SECTION CUT window will become as the Figure 8.1.6.

6. MOVE PLANE
    The functions in the menu allow the user to control the section cut plane.
    - ● **SHOW MODEL**
      The function toggles the model ON/OFF when move the section cut plane.

    - ● **# OF CUT**
      It's the number of the plane to cut the model.

    - ● **DISTANCE**
      The model is cut by a plane every a distance along the W-axis.

    - ● **PREV**
      Select the previous plane of the current plane.

    - ● **ANIM**
      Once select the ANIM, the section cut plane will move and the ANIM will become STOP. The user can select stop when the plane moves the proper position.

    - ● **NEXT**
      Select the next plane of the current plane.

    - ● **THROUGH A NODE**
      User can select a node and the section cut plane will be located through the node.

    - ● **OUTPUT CUR.PLANE and OUT PUT ALL PLANE**
      User can put out the current plane and all plane into *.nas file.

    - ● **WRITE SECTION PROFILE**
      This function will write the section line into the *.txt file.

7. ARC FIT
   This function will label the arcs that accord with the input parameters.

   - **MAX RADIUS**
     Set the maximum radius that can be displayed.

   - **MIN RADIUS**
     Set the minimum radius that can be displayed.

   - **RADIUS DEV**

   - **MIN CHORD**
     Set the minimum chord that can be displayed.

   - **SHOW**
     Locate and label the arcs in the display window.

   - **CLEAR**
     Erase the label.

8. TURN OFF
   Select this button and the defined section cut is closed.

8.1.8   PLAY ANIMATION

Select PLAY if all the options above are set. The animation starts and the user can control the animation by using options in the next dialog window, see Figure 8.1.6.
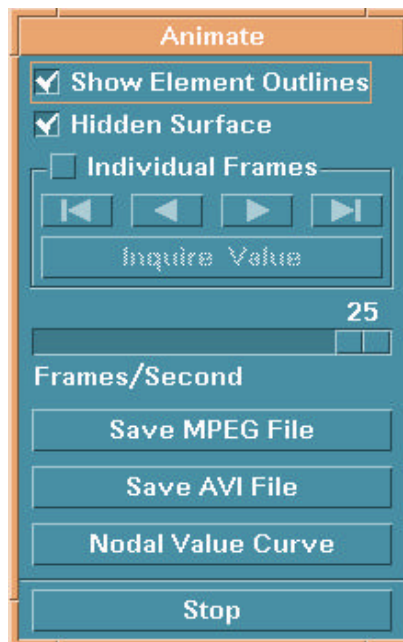
Figure 8.1.6 Animate Control

1. SHOW ELEMENT OUTLINE (toggle)

    When post processing the results, eta/DYNAFORM displays the geometry of the model in
    FILL COLOR mode. The outline of the elements appears in white. The results can be viewed
    without the outline by toggling off the button. The SGI platform does not support this
    function.

2. HIDDEN SURFACE (toggle)

    This option removes hidden surfaces and eliminates any appearance of penetration in the
    model.

3. ANIMATION SPEED

    Drag the slider to adjust the speed of animation.

4. SAVE MPEG FILE

    This function creates a *.avi file from current animation.

5. SAVE AVI FILE

    This function creates an *.avi file from the current animation.

6. NODAL VALUE CURVE

    This function allows the user to trace results using graphs.

    ● Select a component in the next dialog window, see Figure 8.1.7.
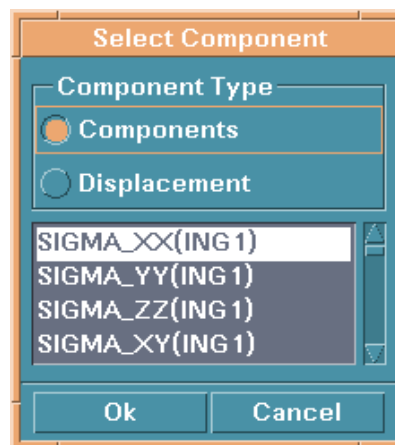
    ● Select a node.

Figure 8.1.7 Select Component

● The curve can be shown in full screen or in a user-defined window. A detailed description is given in Section 8.9.8, UTILITY/LOAD CURVE/SHOW LOAD CURVE.

7. INDIVIDUAL FRAME

Toggle this button on. Select the four-arrow button to select the frame to be displayed.

        `|◄` -- Displays the first frame.

        `◄` -- Displays the last frame.

        `►` -- Displays the next frame.

        `►|` -- Displays the end frame.

8. INQUIRE VALUE

This function allows the user to inquire about stress/strain results and displacement results for individual nodes. The LIST CONTOUR VALUES dialog window appears, see Figure 8.1.8.

Figure 8.1.8 List Contour Value

● The user can select a node by cursor, dragging a window or multi-region, or key in number. The resultant numbers will be displayed in a table.

● **PROFILE**

Displays ten resultant values of nodes a table at a time. These values can be scrolled through using the four control keys: HIGHEST, DOWN, UP, and LOWEST. The default values are

the ten highest values.

- The number of rows in a display table can be changed by keying in a number or adjusting the up and down rolling button.

- Select the button beside LOCATION to select the location of the table.

8.2   PLOT CONTOUR   (Ctrl + Shift + C)

This function is used to display the results of individual steps. The color legend, located on screen, displays the corresponding contour values. The location of the highest contour value on the model is labeled with an asterisk (*) and the lowest is labeled with a zero (0). The options are shown in Figure 8.2.1.
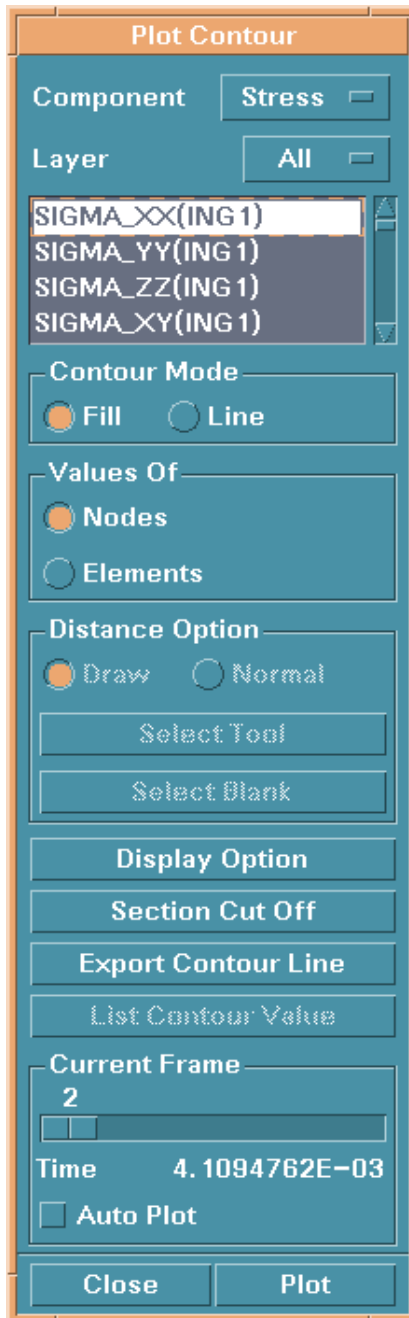
Figure 8.2.1 Plot Contour

8.2.1   CONTOUR MODE

There are two modes: FILL and LINE.

FILL

This function superimposes a fill-color contour image onto the current model.

LINE
This function superimposes a color, wire-frame, and contour-line plot of the component result onto the current model.

8.2.2   TYPE OF VALUE

The user can display results of NODES or ELEMENTS by toggling the button on/off. They are not the displacement results of the element in the result file.

8.2.3   CONTOUR FRAME

This function allows the user to select an individual frame to display.

- The user can adjust the slider to a desired position.

- Select PLOT and the frame will be displayed.

- If AUTO PLOT is toggled on, the frame is displayed automatically while dragging the slider.

8.2.4   EXPORT CONTOUR LINE

This function allows the user to export the contour lines into .lin files.

8.2.5   LIST CONTOUR VALUE

This function is used to list the result of an individual node/element. The LIST CONTOUR VALUE dialog window appears. A detailed description is given in ANIMATE CONTOUR.

8.2.6   CURRENT FRAME

This function allows the user to select a frame to display.

- Drag slider to a desired position.

- Select PLOT and the selected frame will be displayed.

- If AUTO PLOT is toggled on, the individual frame will be displayed automatically while dragging the slider.

Note: The other functions, COMPONENT, LAYER, and DISPLAY OPTION, in the PLOT CONTOUR window are common with those in the section 8.1 ANIMATE CONTOUR.

8.3   ANIMATE DEFORMATION (Ctrl + D)

The functions in this menu animate the displacement of the model in real time. They are similar to those in Section 8.1. The options are shown in Figure 8.3.1.
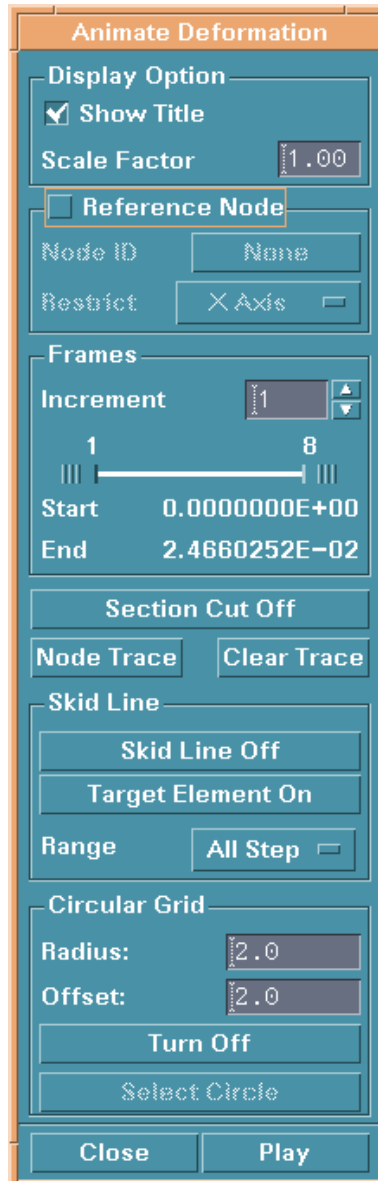

Figure 8.3.1 Animate Deformation

1. **TRACE**

    This function allows the user to trace the path of selected nodes.

    - **NODE TRACE**
      Allows the use to select node(s) and the selected node(s) progress is marked with arrows on the display window.

    - **CLEAR TRACE**
      Erases the trace plot in the display window.

2. **SKID LINE**

    This function will mark the blank elements that come into contact with the binder or other selected tooling. The user can see whether the skid-line is drawn into the part.

    - **SKID LINE OFF**
      The user selects the tooling elements with the SELECT ELEMENT dialog window. After selecting the elements, the SKID LINE OFF will become SKID LINE ON.

    - **TARGET ELEMENT ON**
      This function will highlight the selected elements.

    - **RANGE ALL STEP**
      eta/DYNAFORM marks the skid line area of the current frame and all the frames before the current frame.

    - **RANGE ONE STEP**
      eta/DYNAFORM marks the skid line area of the current frame.

3. **CIRCULAR GRID**

    This function allows user to mark the blank with a circular grid.

    - **RADIUS**
      The radius of the circle in the circular grid.

    - **OFFSET**
      The distance between the circles within the circle grid.

    - **TURN OFF**
      User can select elements by the SELECT ELEMENT dialog window, then user can define the pave direction by the INPUT COORDINATE dialog window.
      After the circular grid is paved on the selected elements, the button will become TURN ON, if turn off the circular grid, the button will become into the TURN OFF.

    - **SELECT CIRCLE**

This function allows the user to select circle(s) by cursor to measure the amount of deformation. eta/DYNAFORM will mark the major and minor axis of the selected elements, the value of the axis will be displayed in the prompt window.

Note: The other functions in the ANIMATE DEFORMATION window are common to those in the preceding sections'.

8.4   PLOT DEFORMATION (Ctrl + Shift + D)

The functions in this menu display displacement of an individual step and list displacement for the selected node(s), see Figure 8.4.1.
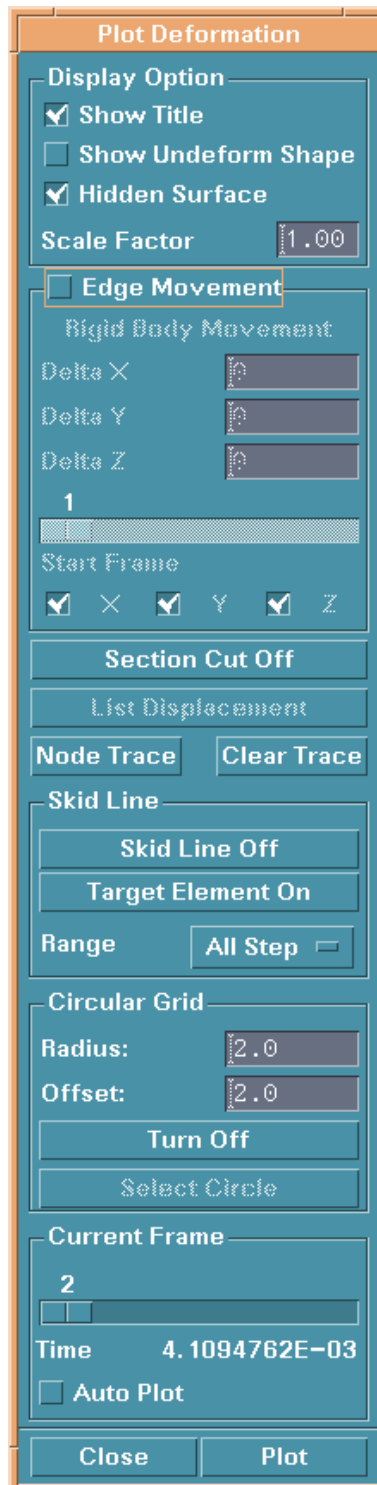
Figure 8.4.1 Plot Deformation

8.4.1   EDGE MOVEMENT

This function shows displacement of the nodes at the edge by arrows in different colors. The values of displacement are shown in the color bar located on the screen.

When select the EGDE MOVEMENT, the EDGE MOVEMENT window will be displayed

- **DELTA X Y Z**
  Those are the offset based the displacement of the edge movement. User can input none, any one or any combination of them.

- **START FRAME**
  The displacement shown on the window is the relative displacement between the selected frame and the start frame.

- **X Y Z**
  User can select any direction X Y Z and any combination of them to display the displacement.

8.4.2   LIST DISPLACEMENT

This function is used to display the displacement of nodes. The options in this menu are common with those in INQUIRE VALUE of ANIMATE CONTOUR.

Note: The other functions in the PLOT DEFORMATION window are common with those relative sections above.

8.5   ANIMATE FLD (Ctrl +F)

This function is used in the simulation of metal stamping to study the formability of the blank (safety and failure zones). The X and Y-axes on the diagram represent the minor and major strains of each element. The options in this menu are shown in Figure 8.5.1.
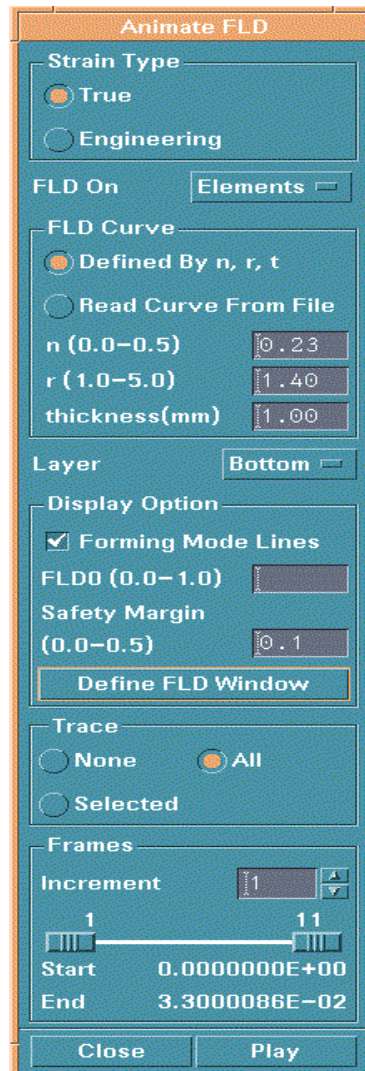

Figure 8.5.1 Animate FLD

8.5.1   STRAIN TYPE

This function allows the user to toggle between the FLD display of the geometry or the true strain.

8.5.2   FLD ON

This function allows the user to toggle on/off the FLD display of the node or element strain.

8.5.3   FLD CURVE

This function allows the user to define a new FLD curve by entering values of n, r, and thickness or by loading a curve file.

8.5.4   LAYER

Select the button beside LAYER to select surface through the thickness.

8.5.5   DISPLAY OPTION

These functions are not active until the animation has been shown once.

- **FORMING MODE LINES**
  Toggles ON/OFF the extra forming lines on the FLD display.

- **FLD0**
  FLD0 is the lowest point on the failure curve. The user can move the failure curve in Y-direction by resetting FLD0 (0.0-1.0).

- **SAFETY MARGIN**
  The user can reset the margin area of the FLD by changing the distance from the margin curve to the failure curve.

- **REDEFINE FLD WINDOW**
  eta/DYNAFORM prompts the user to define a window for displaying FLD.

8.5.6   TRACE

This option traces selected or all elements along the FLD curve. Element progress is marked with arrows on the curve in the FLD window.

- **NONE**
  No tracing marks will be displayed in the FLD WINDOW (default).

- **ALL**
  All elements/nodes FLD values will be shown when the user animates.

- **SELECTED**

    The user will be prompted to select up to 25 elements or nodes. The FLD value will be shown in the FLD window and there will be arrows to represent the change of value in each step.

8.5.7   FRAME

- **INCREMENT**

    Define the increment between frames.

- The user can drag the slider to set the starting frame and ending frame.

8.5.8   PLAY ANIMATION

The animation will be displayed automatically when the user selects PLAY and the ANIMATE CONTROL dialog window appears, see Figure 8.5.2.
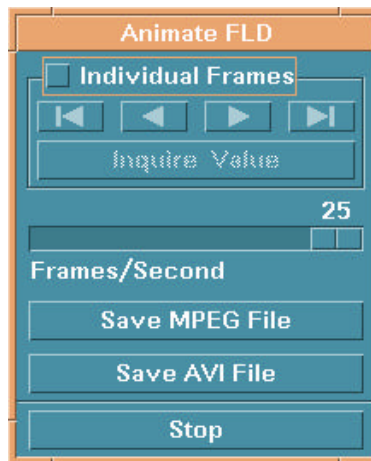
Figure 8.5.2 Animate Control

The options in this menu are common with those in Figure 8.1.5, refer to ANIMATE CONTOUR.

Note: The other functions in the ANIMATE FLD window are common with those relative sections before.

8.6   PLOT FLD (Ctrl + Shift + F)

This function is used to display the FLD plot of an individual step. Most options are common with those in ANIMATE FLD.

- **LIST FLD VALUES**
  Lists the FLD value and status of nodes/elements. Once selected, the LIST FLD VALUES dialog window appears. The options in this dialog window are common with those above.

- Select step by dragging the slider. Select PLOT to display the plot or turn AUTO PLOT on.

Note: The other functions in the PLOT FLD window are common with those relative sections above.

8.7   Vector (Ctrl + Shift + V)

The function is used to show the direction and value of Displacement, Velocity, Principal Stress and Principal in Plane Strain, the window appears as the Figure 8.7.1.
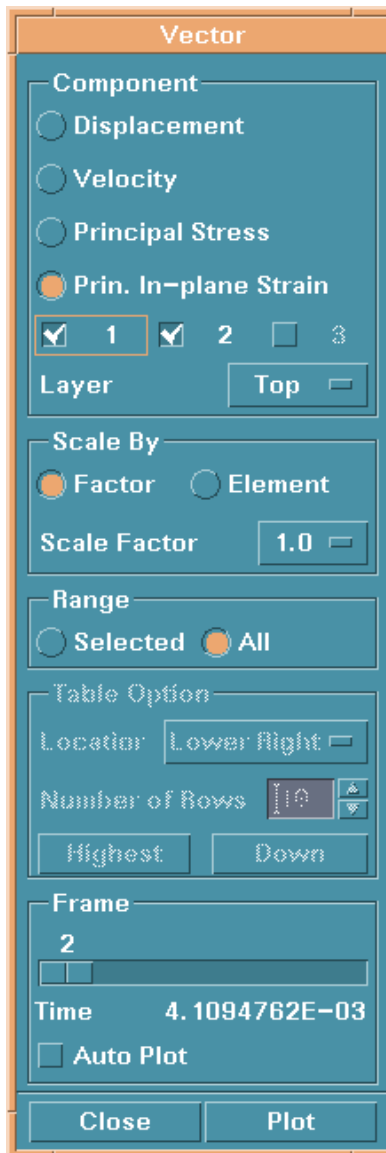
Figure 8.7.1 Vector

1. COMPONENT
    The function allows the user to select any

2. SCALE BY
    The functions allow the user to select what vector will be displayed.

    ● **FACTOR (toggle)**
        User can select the button beside the SCALE FATOR to select the proper scale that
        avoids the vectors are too long or too small. User can know the value by the color and
        the length of the vector.

  ● **ELEMENT (toggle)**
    The displayed vector is adjusted according the size of the element automatically. The
    user can know the value only by the vector color.

3. RANGE

  ● **SELECTED (toggle)**
    User can select elements by the SELECT ELEMENTS dialog window.
    eta/DYNAFORM will only plot the resultant vectors of the selected elements.
    When the SELECT is toggled on, the TABLE OPTION can be used. Please refer the
    previous related section.

  ● **LIST VALUE**
    This function allows the user to select element(s) by SELECT ELEMENTS dialog
    window to list the strain value in a table.
    The 10 largest values will be listed in the table if the number of selected elements is
    more than 10.

Note: The other functions in the VECTOR window are common with those relative sections above.

8.8   BLANK OUTLINE

This function calculates the outline of the blank in the binder surface (undeformed shape). It is useful to
estimate the necessary blank size to form the part. The calculation is based on the trim line on the part after
the draw-die or spring back simulation. The following steps should be followed.

  ● Select the POST-PROCESS option in the FILE/OPEN dialog box and read in d3plot or .pp file
    from the DYNA analysis.

  ● Turn off all other parts except the blank.

  ● Read in a line data file for the trim line using the IMPORT function in the FILE menu. The line
    data file should only consist of the trim line(s).

  ● Select the BLANK OUTLINE function from the POSTPROCESS menu.

  ● The final step of the deformed shape of the blank will be displayed to show the relative position
    of the trim line. The user will then be prompted to select the trim line, please see the Figure
    8.8.1.

Figure 8.8.1 Control Keys

  ● The calculated blank outline will then be displayed in white color on the undeformed shape of

the blank.

8.9   MIRROR RESULT BY…

This function allows the user to select a mirror plane to mirror the analysis result if the resultant value is displayed by contour.
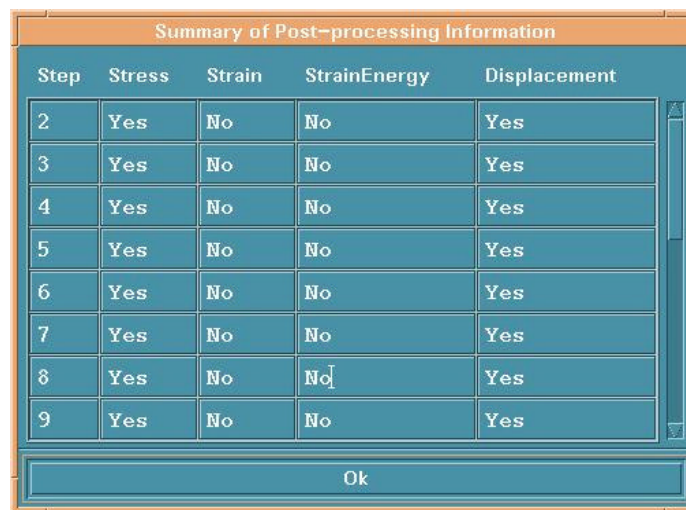
There are three mirror planes, XY PLANE, YZ PLANE and ZX PLANE in the function.

8.10   REVERSE TITLE (toggle)

The function will display the title in reverse.

8.11   SUMMARY

This function displays the status of the result file via Pop-Up window. It includes the stress, strain energy, and the availability of displacement at given time steps. See Figure 8.11.1.

Step	Stress	Strain	StrainEnergy	Displacement
2	Yes	No	No	Yes
3	Yes	No	No	Yes
4	Yes	No	No	Yes
5	Yes	No	No	Yes
6	Yes	No	No	Yes
7	Yes	No	No	Yes
8	Yes	No	No	Yes
9	Yes	No	No	Yes

Figure 8.11.1 Summary

8.12   EXIT POSTPROCESS (Ctrl + Shift + X)

When post processing, preprocess menu items are not active. The user must exit from the post processing in order to open and work on a new database in the preprocessor. A DYNAFORM Question window appears, see Figure 8.12.1.

Figure 8.12.1 Exit Question

Select YES and preprocess menus such as TOOLS and PREPROCESS are activated while POSTPROCESS is inactive.

Select NO to return to POSTPROCESS.

CHAPTER 9
SETUP MENU

This menu contains various preprocessing utilities that are toggle switch activated, see Figure 9.1.
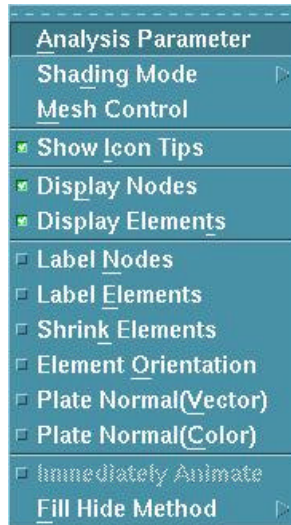
Figure 9.1 Setup Menu

Unlike the pop-up windows in FILE, PART, TOOLS, PREPROCESS, ANALYSIS, and POSTPROCESS, this menu is a TEAR-OFF window. It can be changed into a window with a frame by selecting the line at the top. This allows the user to have the menu conveniently displayed.

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

9.1   ANALYSIS PARAMETERS

This function allows the user to define forming parameters for the LS-DYNA analysis, see Figure 9.1.1.
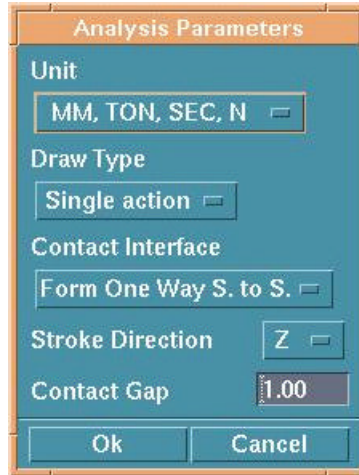
Figure 9.1.1 Analysis Parameters

● The user can select a unit type by selecting the button below UNIT, see Figure 9.1.2.
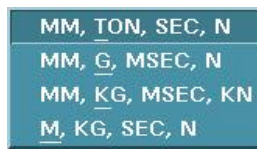
Figure 9.1.2 Unit Type

● The user can select a draw type by selecting the button below DRAW TYPE, see Figure 9.1.3.
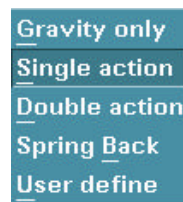
Figure 9.1.3 Draw Type

GRAVITY ONLY

The blank is deformed by gravity.

TOGGLE DRAW

The punch is above the blank.

INVERTED DRAW

The punch is under the blank.

SPRING BACK

Read in the nikin file. Rewrite the file as *.nik file. For running SPRING BACK, use NIKE3D.

USER DEFINE

The draw type is user defined.

> **Note:** **The defined DRAW TYPE works in conjunction with the TOOL/BLANK INITIAL GAP when the tools are in AUTO POSITION.**

● There are several contact interface types, see Figure 9.1.4.

Auto Nodes to Surf.
Auto One Way S. to S.
Auto Surf. to Surf.
Form Nodes to Surf.
Form One Way S. to S.
Form Surf. to Surf
Nodes to Surf.
One Way S. to S.
Surf. to Surf.
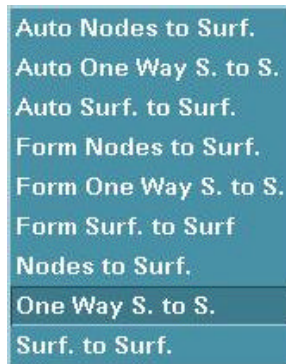
Figure 9.1.4 Contact Interface Type

The user can select the interface type for all tool/blank interfaces.

> **Note:** **Refer to the *LS-DYNA User's Manual*, keyword *CONTACT, for detailed information about each interface type.**

● The user can define stroke direction in X, Y, or Z.

● In Section 5.3 TOOL/POSITION TOOLS, the user can set a clearance between the tool pieces and the blank for the AUTO POSITION command. The default is 1.0 unless a blank thickness is defined under the BLANK PROPERTY menu.

9.2   SHADING MODE

This function offers three options for shading a part or a model, see Figure 9.2.1. GOURAND SHADING, which is smooth shading, is the default setting for eta/DYNAFORM.

Figure 9.2.1 Shading Options

9.3   MESH CONTROL

The options in this dialog window are used to control the mesh generation. See Figure 9.3.1.

AUTO SURFACE MESH (toggle)
This function will mesh the displayed surfaces directly when select SURFACE MESH.

AUTO PLATE NORMAL (toggle)
 This function will mesh the part with consistent plate element normal.

9.3.1   CONTROL POINT (toggle)

This function allows the user to pre-select the points on a line for the desired node locations associated with the 2L, 3L, 4L, 6S, 8S, 9S, and 12S mesh programs. It is used in conjunction with the options in Section 6.3.4 PREPROCESS/ELEMENT/SURFACE MESH. See Figure 9.3.1.

9.3.2   CORNER BIAS MESH

The corner bias mesh command 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. See Figure 9.3.2.
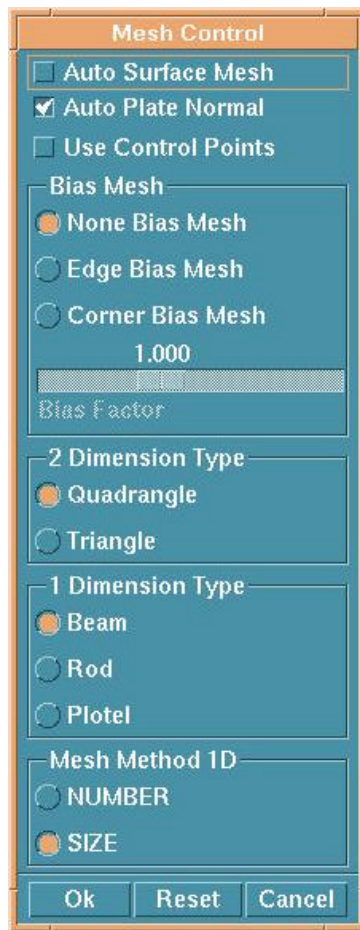
Figure 9.3.2 Comer Bias

If the button at the top of the window is turned on, the user can define the bias factor by dragging the slider.

Note: **A bias factor greater than 1.0 generates proportionately larger elements from the selected corner. A bias factor less than 1.0 generates proportionately smaller elements from the selected corner. This function may be used with only the 2L, 3L, 4L, 6S, 8S, 9S, and 12S functions.**

9.3.3   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. See Figure 9.3.2.

If the button on top of the window is turned on, the user can define the bias factor by dragging the slider.

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 may be used with only the 2L, 3L, 4L, 6S, 8S, 9S, and 12S functions.**

9.4   SHOW ICON TIPS (toggle)

This function toggles the explanations of all icons on the icon bar and dialogues ON/OFF.

9.5   DISPLAY NODES (toggle)

This function toggles all nodes ON/OFF.

9.6   DISPLAY ELEMENTS (toggle)

This function toggles all elements ON/OFF.

9.7   LABEL NODES (toggle)

This function toggles the node labels on/off. All nodes in the displayed parts will display their node numbers. This function is also found under NODE OPTIONS.

9.8   LABEL ELEMENTS (toggle)

This function toggles the element labels on/off. All elements in the displayed parts will display their element numbers. This function is also found under ELEMENT OPTIONS.

9.9   SHRINK ELEMENTS (toggle)

This function reduces the size of the selected elements by 20 percent. SHRINK ELEMENTS is also useful for locating any missing elements in a shell or solid structure.

9.10   ELEMENT ORIENTATION (toggle)

This function displays element orientation. Beam element orientation is shown by drawing a vector at the first node of the beam toward the element local Y-axis. Shell orientation is shown by drawing an arrow from the first node of an element, toward the second node of an element. Drawing an arrow from the bottom surface towards the top surface shows solid element orientation. This display is useful for picturing the beam orientation and the normal direction of the shell elements (the normal of the shell follows the right hand rule from the first node to the second node).

Note: **The clarity of the element orientation display improves when the SHRINK ELEMENTS**

command is activated.

9.11   PLATE NORMAL (Vector)

This function displays 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.

9.12   PLATE NORMAL (Color)

This function displays the normal direction of an element with color.

9.13   IMMEDIATELY ANIMATE

When this function is checked, the post-processor only animates with deformation. It does not load information about strain, stress, etc. therefore cutting down the time it takes to animate.

9.14   FILL HIDE METHOD

This function is to define the algorithm used for Hidden Surface plot. Two algorithm are available in the program.

9.14.1 STENCIL BUFFER

This algorithm will produce a more accurate Hidden Surface plot however it requires more time than the Polygon Offset function.

9.14.2 POLYGON OFFSET

This algorithm plots "fill hide" quicker than the Stencil Buffer but at slightly lower accuracy. This is the default setting for the HILL HIDE plot.

CHAPTER 10

UTILITY

The functions in the UTILITY menu make up eta/DYNAFORM's "tool kit." Many of these functions are also found in other menus. The options are shown in Figure 10.1.

Figure 10.1 Utility Menu

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

10.1   ANGLE BETWEEN LINES

This function measures the angle between two selected vectors.

In the INPUT COORDINATE window, select two points/nodes to define the first vector. Select two more points/nodes to define the second. eta/DYNAFORM displays the measures of four angles in the prompt window.

10.2   DISTANCE BETWEEN POINTS/NODES

This function calculates the distance between two points, two nodes or a node and a point. This function is also listed in the NODE OPTIONS menus.

The user selects 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 window.

10.3   RADIUS THROUGHT 3PTS/3NDS

This function calculates the radius through 3 points or nodes.
 The user can select three points/nodes by using option in the INPUT COORDINATE dialog window, and the result will displayed in the prompt window as follow,
      R=XXX CENTER AT, X=XXX, Y=XXX, Z=XXX

10.4   AREAR OF SELECTED ELEMENTS

This function calculates the area of the selected elements, and the area and the mass center are given in the prompt window.

10.5   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 shown from the first location to the second location.

Note: This function is used in conjunction with DEFINE TITLE.

10.6   DEFINE TITLE

This function allows the user to enter a title or text at any location of the displayed screen. See Figure 10.6.1.
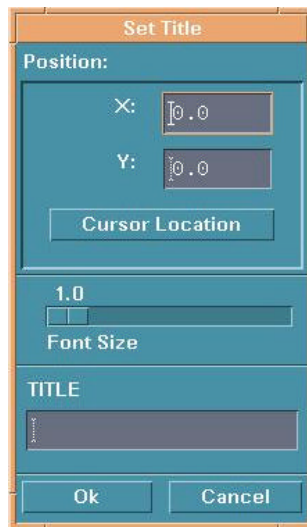
Figure 10.6.1 Define Title

- Enter the title in the field under TITLE.

- Drag the slider to set the font size of the title.

- The user can define the position of the title by entering the screen coordinates of the X, Y values or by selecting CURSOR LOCATION.

10.7   IDENTIFY NODE/POINT

This function identifies any node/point and its corresponding global location in X, Y, Z coordinates. This function is also found in Section 6.4, PREPROCESS/NODE.

10.8   IDENTIFY ELEMENT

This function allows the user to identify an element and its nodes. It is also found in Section 6.3, PREPROCESS/ELEMENT.

10.9   FIND ELEMENT

This function allows the user to find or identify an element via its element number. This function is also listed in the ELEMENT OPTIONS menu.

10.10   FIND NODE

This function allows the user to determine the X, Y, Z coordinates of the specified node. This function is also listed in Section 6.4, PREPROCESS/NODE.

10.11   LOAD CURVE

The options in this menu are used to define or modify the load curves. Select an option and then choose DONE to go to the next dialog window. The options are shown in Figure 10.11.1.

Figure 10.11.1 Load Curve

10.11.1 CREATE CURVE

This function allows the user to create load curves, see Figure 10.11.2.
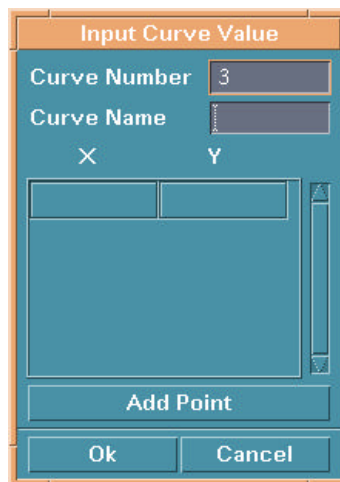
Figure 10.11.2 Create Curve

- Enter CURVE NUMBER and NAME in the fields at the top. If the curve number is already being used by another curve or only one point is on the curve, there is a warning message in

the prompt window.

- Enter a pair of numbers to define a point on the curve.

- Select ADD POINT to define another point.

10.11.2 DELETE LOAD CURVE

This function allows the user to delete a load curve.

- Select a curve in the SELECT CURVE window. The user can select all curves by selecting ALL. See Figure 10.11.3.
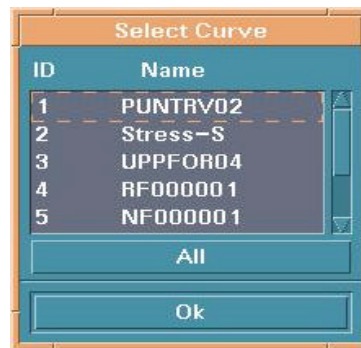
Figure 10.11.3 Select Curve

- A DYNAFORM Question window appears, see Figure 10.11.4.
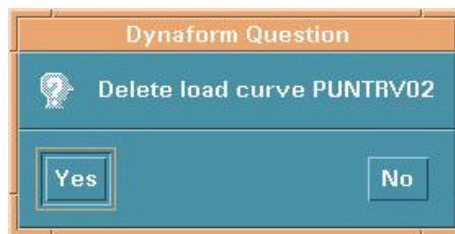
Figure 10.11.4 Delete Curve Question

- Select YES to delete the selected curve.

- Select NO to reject the choice.

10.11.3 LIST LOAD CURVE

This function lists all available load curves.

The LIST CURVE window appears, see Figure 10.11.5.

Figure 10.11.5 List Curve

10.11.4 MODIFY LOAD CURVE

This function allows the user to modify points on a selected load curve.
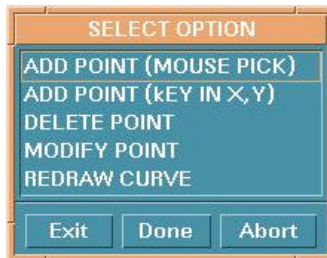
Figure 10.11.6 Modify Curve

After selecting a curve in the SELECT CURVE dialog window, the SELECT OPTION dialog window is displayed, see Figure 10.11.6.

1. ADD POINT (MOUSE PICK)

    The user can select any location in the curve screen. The selected point will be added to the curve.

2. ADD POINT (KEY IN X, Y)

    In the fields of the next dialog window, enter the X, Y values and select DONE.

3. DELETE POINT

    The user can select any point on a curve. The point will be deleted automatically.

4. MODIFY POINT

There are several options, see Figure 10.11.7.

Figure 10.11.7 Modify Point

- **INCREMENTAL XY**

  Enter the X, Y increments and select OK in the next dialog window. Select point(s) in the curve screen to move.

- **KEY IN XY**

  Select a point in the curve screen. The X, Y values will be shown in the next dialog window. Change 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 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. REDRAW CURVE
   This function redraws the modified curve to fill the display screen.

10.11.5 READ CURVE DATA

This function imports existing curve data into the eta/DYNAFORM database. 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 name.
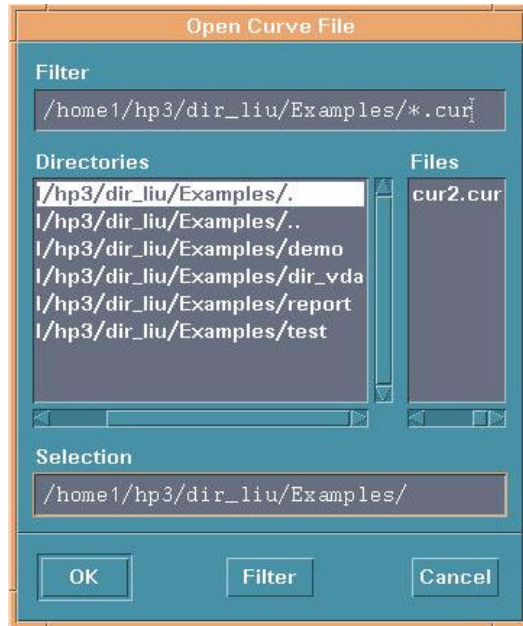
Figure 10.11.8 Open Curve File

10.11.6 RENUMBER LOAD CURVE

This function renumbers the ID of a selected curve.

- The user selects a curve from the SELECT CURVE window, see Figure 10.11.9.

- Enter the new number in the next dialog window.
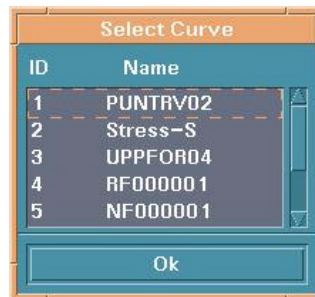
Figure 10.11.9 Select Curve

10.11.7 RENAME CURVE

This function renames 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.

10.11.8 SHOW LOAD CURVE

This function displays the selected curve(s). Also, the user can organize the display by using the options in the dialog window CURVES, see Figure 10.11.10.
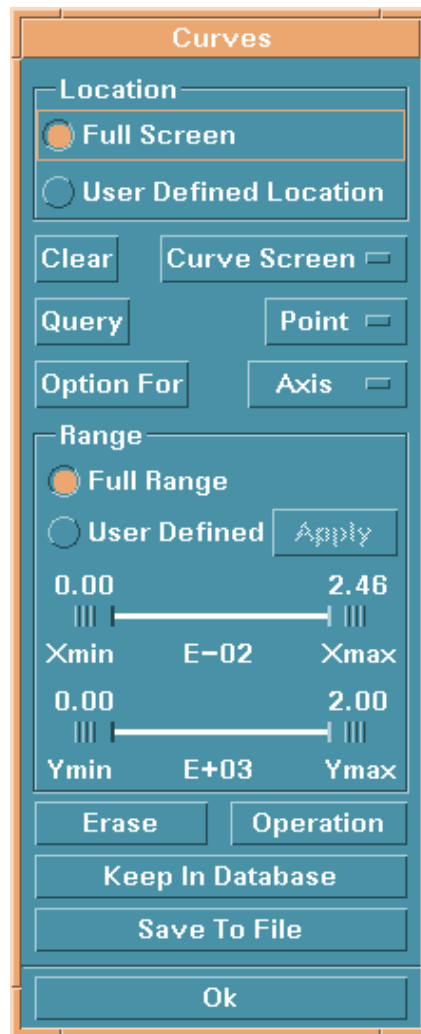
Figure 10.11.10 Curve Plot Option

● Select the load curve in the SELECT CURVE dialog window. See Figure 10.11.3.

-SHOW LOAD CURVE
1. LOCATION

   ● Toggle FULL SCREEN on to display the curve(s) in the entire graphics.

   ● Toggle USER DEFINED LOCATION on and drag a window to display the curve(s) in the drag window.

-SHOW LOAD CURVE
2. CLEAR
    This function removes an object from the screen.

   ● Select the button to the right of CLEAR to select an object:

      CURVE SCREEN

      Clears all objects in the curve screen.

      MODEL

      Clears the displayed model.

      ETA LABEL

      Clears the label at the left bottom of the display screen.

      ALL

      Clears all objects in display screen.

   ● Select CLEAR and the objects are cleared.

-SHOW LOAD CURVE
3. QUERY
    This function displays information about a graph in the curve screen.

   ● Select the button to the right of QUERY and select one of the three items.

      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. The user can select one from the list or pick 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.

- After selecting one of the above options, select QUERY.

-SHOW LOAD CURVE

4.   OPTION FOR GRAPH DISPLAY
     This allows the user to manipulate the display settings of axis, graph, curve, grid and legend. See Figures 10.11.11 ~ 10.11.15.

- 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.

Figure 10.11.11 Axis Option
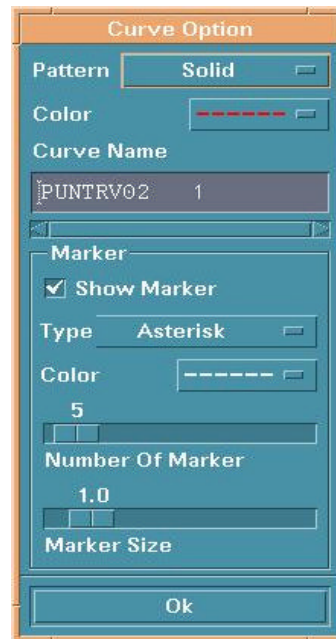
Figure 10.11.12 Curve Option

Figure 10.11.13 Graph Option
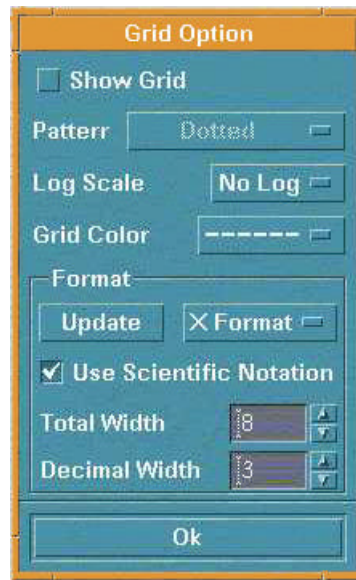
Figure 10.11.14 Grid Option

Figure 10.11.15 Legend Option

-SHOW LOAD CURVE
5. DEFINE RANGE OF GRAPH
    This function zooms in on a specific section of graph.

    ● Toggle FULL RANGE on to show a curve in full range (default).

    ● Toggle USER DEFINED on to adjust the values of four sliders and then select APPLY to reset the range.

-SHOW LOAD CURVE
6. OPERATION

This function allows the user to apply mathematical operations to the shown curve. The options are shown in Figure 10.11.16.
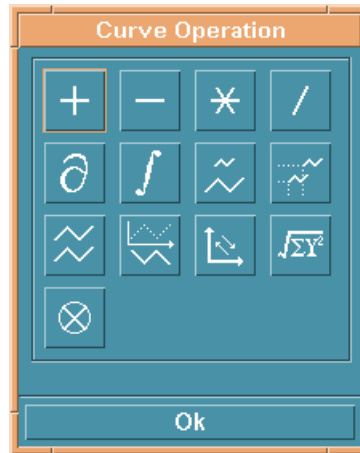
Figure 10.11.16   Curve Operation

After an operation is selected, the SELECT CURVE dialog window will appear for selecting curves for the operation, see Figure 10.11.17. 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
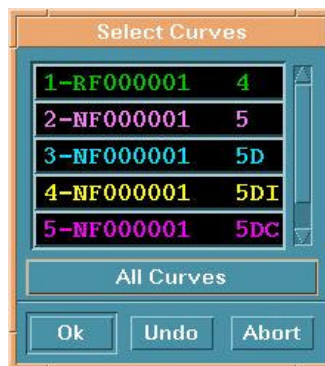
Figure 10.11.17   Select Curve

 -- This function creates a new curve in the curve screen whose values of Y are the sums of the selected curves. At least two curves must be selected.

 -- This function creates a new curve whose values are the differences of two selected curves.

 -- This function creates a new curve whose values are the product of two curves.

| | -- | This function creates a new curve whose values are quotients of two selected curves. |

| | -- | This function creates a new curve that is the derivative of the selected curve with respect to time. |

| | -- | This function creates a new curve that is the indefinite integral of the selected curve with respect to time. |

| | -- | This function scales the selected curve using magnification factors in the X, Y directions. |

| | -- | This function moves the selected curve in the X, Y directions. |

| | -- | This function creates a new curve that is a copy of the selected curve at the same position and in different colors. |

| | -- | This function negates the values of the selected curve in the Y direction. |

| | -- | This function switches X-axis and Y-axis. |

| | -- | This function creates a new curve that is the resultant of the selected curve. |

| | -- | This function creates a new curve that is cross plot of the selected curve. |

Note: **All results under the above operations are temporarily displayed in the curve screen. They will not be automatically saved as loaded curves. To keep the curve data, the user must perform either steps 8 or 9.**

-SHOW LOAD CURVE
   7.   ERASE
      Removes a selected curve from a displayed graph.

-SHOW LOAD CURVE
   8.   KEEP IN DATABASE
      This function saves the selected curve in the database with a name beginning with CUR.

-SHOW LOAD CURVE
   9.   SAVE TO FILE

This function saves the display as a text file in a directory, see Figure 10.11.19.
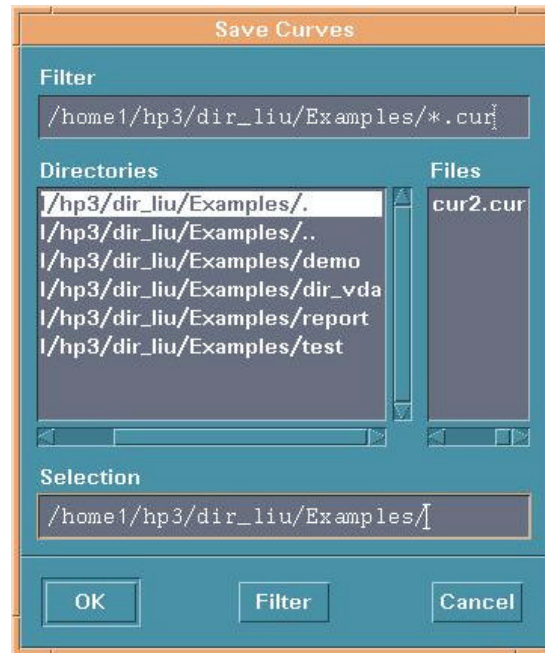

Figure 10.11.19 Save Curve

10.12   SHOW LINE

This function identifies any existing line and its direction. See Section 6.1, PREPROCESS/LINE/POINT.

10.13   COORDINATE SYSTEM

The functions in this menu are used to create and modify local coordinate systems. The functions are shown in Figure 10.13.1.

- Dragging the slider changes the size of the displayed coordinate system. Select APPLY when finished.

- **COORD. SYS. ON**
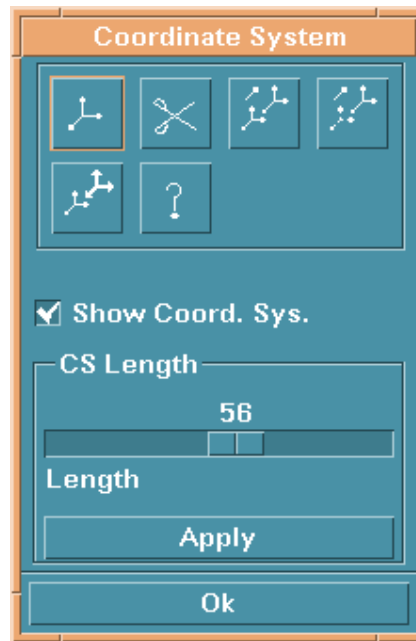
Toggles all created coordinate systems ON/OFF.

Figure 10.13.1 Coordinate System

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

10.13.1 CREATE COORDINATE SYSTEM

This function creates a coordinate system.

Enter coordinate system ID and select OK in the next dialog window. The LCS dialog window appears, see Figure 10.13.2

As mentioned in previous chapters, this window is displayed when a current local coordinate system is needed. In those cases, the user can create a new LCS or select a system as the current LCS by selecting GLOBAL, CURRENT LCS, or LAST DEFINED LCS.
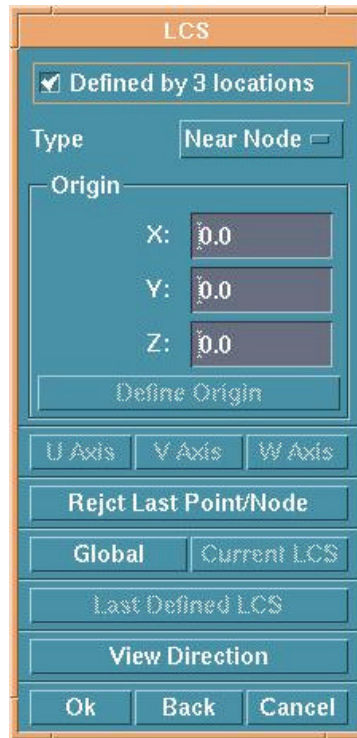
Figure 10.13.2 LCS

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 DEFINED LCS

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

VIEW DIRECTION

Define the coordinate system that the view direction is the W axis as 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 U-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. The 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 and the V-axis is determined by the cross product WXU.

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

   - Select DEFINE ORIGIN. The INPUT COORDINATE dialog appears. Select a point or node as the origin and 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 10.13.3) 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 DONE.

   - Select OK from the AXIS dialog.

   - Select OK from the LCS dialog.

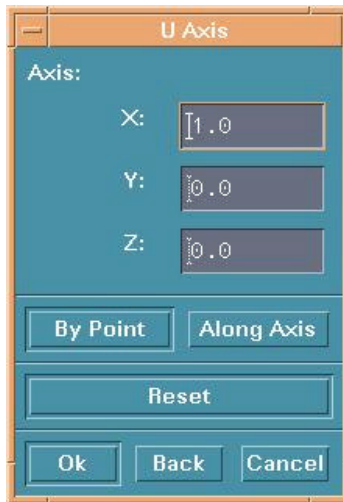   - The coordinate system is displayed.

Figure 10.13.3 U Axis Definition

10.13.2 DELETE COORDINATE SYSTEM

This function deletes a local coordinate system from the database.
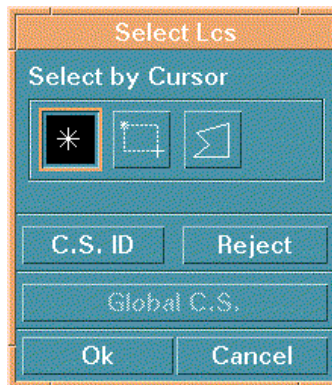

Figure 10.13.4 Select LCS

● The SELECT LCS dialog window appears, see Figure 10.13.4.

 -- Pick a coordinate system.

 -- Drag a window to include the coordinate system.

 --Define a multi-point region to include the coordinate system.

C.S. ID

Enter the starting and ending coordinate system ID and the increment in the next dialog window.

REJECT

Reject last selection.

Note: **To remove the local systems/definitions from individual nodes, the user must reassign the nodes to the global coordinate system.**

10.13.3 COPY COORDINATE SYSTEM

This function copies a selected coordinate system to a new location.

- The SELECT LCS dialog window appears with only two options activated.

    1. Pick the desired coordinate system displayed on screen using the default mouse pick or select GLOBAL C.S. to select the global coordinate system.

    2. Define the new location using the INPUT COORDINATE dialog window. The selected coordinate system will be copied automatically.

10.13.4 MODIFY COORDINATE SYSTEM

This function modifies the definition of an existing coordinate system.

- After a coordinate system is selected in the SELECT LCS dialog window, the SELECT OPTION window is displayed. See Figure 10.13.5.

Figure 10.13.5 Modify Option

If MOVE is selected, the INPUT COORDINATE dialog window appears. The selected coordinate system will be moved automatically after a new location is defined.

If ROTATE is selected, the user must enter a rotation angle in the next dialog window. The coordinate system will be rotated automatically.

10.13.5 CURRENT COORDINATE SYSTEM

This function allows the user to designate a CURRENT status to a selected coordinate system.

- The user selects a coordinate system by cursor.

- The selected coordinate system will be highlighted and the last one changes its color.

10.13.6 IDENTIFY COORDINATE SYSTEM

This function identifies the type and origin of a coordinate system.

- Select a coordinate system by cursor.

- After a coordinate system is selected, eta/DYNAFORM prompts:

  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.

10.14 WORK PLANE

This function allows the user to create and modify a work plane. A work plane is a reference plane displayed as white dots. It allows a user to select coordinates without having to select a point or node. When the Work Plane is active the user can select coordinate data by clicking anywhere on the screen. LENGTH adjusts the size of the work plane and the distance between dots can be adjusted by PRECISION. Selecting the button in the DISPLAY OPTIONS window toggles the current work plane on.

- Input plane name, precision (distance between two near points), and length, see Figure 10.14.1.

- Select DEFINE LCS OF PLANE to define a coordinate system at the desired position.

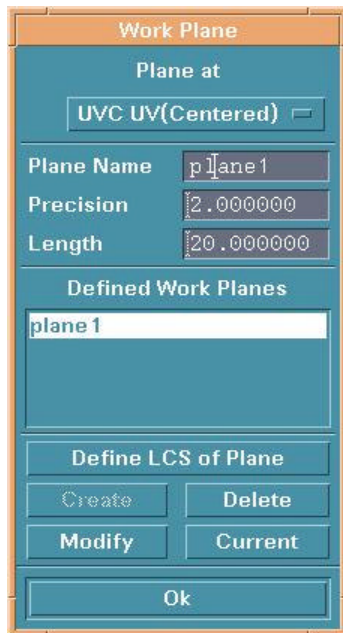● Select the button below PLANE AT to select a plane location in the coordinate system, see Figure 10.14.2.

Figure 10.14.1 Work Plane

Figure 10.14.2 Plane Position

● **CREATE**

The work plane will be created.

● **DELETE**

Deletes the selected work plane.

● **MODIFY**

Select this button after a plane name, precision or length are modified.

● **CURRENT**

Turns the selected work plane on.

10.15 DATABASE STATISTIC

This function displays statistics about the eta/DYNAFORM database regarding geometry, models, materials and interfaces.

CHAPTER 11
VIEW OPTIONS

The functions in this menu are used to adjust the display of the items on the screen. The options are shown in Figure 11.1.
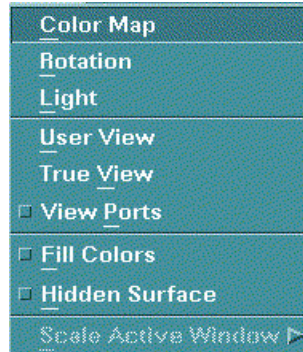

Figure 11.1 View Option

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

11.1   COLOR MAP

This function is used to change the default color in the color bar. See Figure 11.1.1.
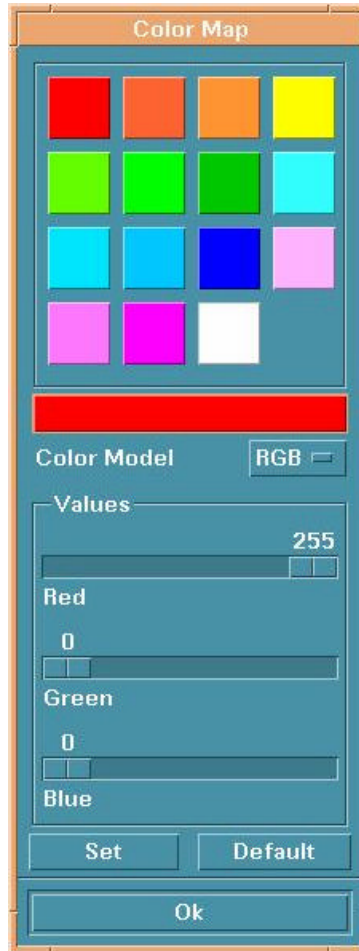
Figure 11.1.1 Color Map

● Select a color from the color palette at the top. The color will be shown in the long rectangle beneath it.

● Select COLOR MODEL to choose a color model, RGB (Red, Green, and Blue) or HSV (Hue, Saturation, and Luminosity).

● Drag the slider to change the values.

● Select SET and the new color will be exchanged with the selected color in the color palette.

● Select DEFAULT to change the colors in the color palette to default.

11.2   ROTATION

This function allows the user to rotate an object in the display area about the global (or virtual) X, Y, and Z-axis. The options are shown in Figure 11.2.1.
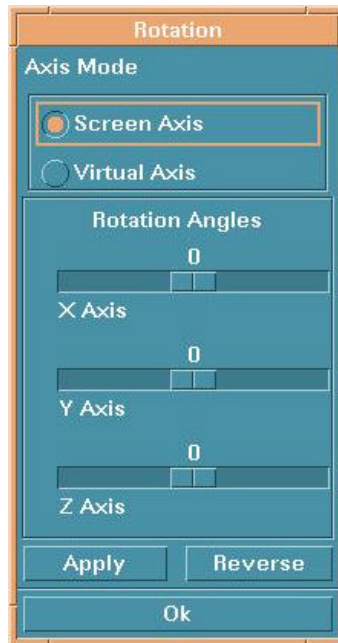

Figure 11.2.1 Rotation

● **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 is the direction from the left to right of the terminal screen.

   2. SCREEN Y
      Axis is the direction from the bottom to the top of the terminal screen.

   3. SCREEN Z
      Axis is the direction from the screen to the user.

● **VIRTUAL AXIS**
   Rotates an object about global (or virtual) X, Y, and Z-axes according to an angle.

● Drag the sliders to adjust angles.

● Select APPLY to rotate. This function can be repeated as many times as desired.
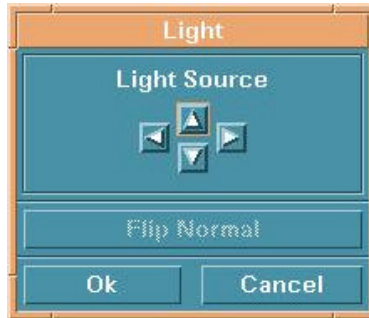

Figure 11.3.1 Light Source

● Select REVERSE to reverse. This function can be repeated as many times as desired.

11.3   LIGHT

This function moves a directional light source uniformly with a consistent intensity, see Figure 11.3.1. The right and left arrow buttons move the light source in the positive and negative X-direction, while the up and down arrow buttons move the light source in the positive and negative Y-direction.

11.4   USER VIEW

This function is used to save or store a desired view in the current eta/DYNAFORM database and recall any previously saved view. See Figure 11.4.1.
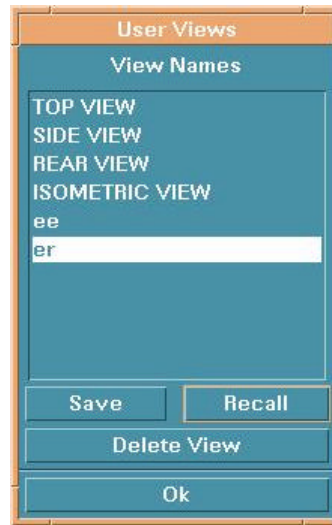
Figure 11.4.1 User View

- **SAVE A VIEW**
  Select SAVE and enter a title to save the current displayed view. The saved view will be listed in the window. eta/DYNAFORM allows the user to save up to six new views plus the four pre-defined views (a total of 10 views).

- **RECALL A VIEW**
  Select a view from the list and choose RECALL.

- **DELETE A VIEW**
  Select a view and choose DELETE VIEW.

11.5   TRUE VIEW

This function allows 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 for defining a local coordinate system.

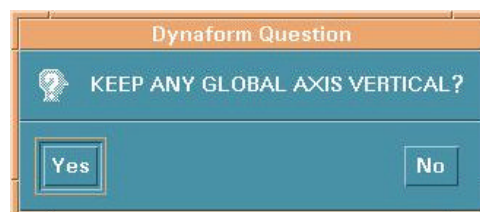- A DYNAFORM Question appears, see Figure 11.5.1.

Figure 11.5.1 Axis Question

1. YES
    Allow the user to selects 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).

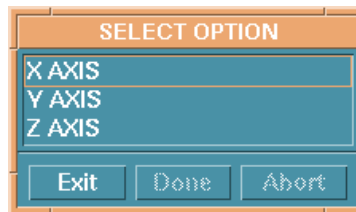●   If select YES in the figure 11.5.1 The SELECT OPTION window appears, see Figure 11.5.2.

Figure 11.5.2 select Option

User can select any axis (X, Y and Z) that will be displayed from bottom to the top of screen

11.6   VIEW PORTS (toggle)

This function allows the user to view an object in the split-screen display mode. The left upper corresponds
to the top view. The left lower corresponds to the side view. The right upper corresponds to the isometric
view. The right lower corresponds to the rear view.

11.7   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.

11.8   HIDDEN SURFACE (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.

11.9   SCALE ACTIVE WINDOW

This function scales an active window from measurements provided from the pull-down menu.

APPENDIX A

HARDWARE AND SOFTWARE REQUIREMENTS

PLATFORM	OS VERSION	GRAPHICS CARD	DISK SPACE
HP	HP-UX 9.01+	Minimum CRX 8 Plane	63
IBM	AIX 3.2+	Minimum 24 Plane Graphics	69
SGI	IRIX 5.2+	All Graphics Boards Supported	37
DEC	OSF1 v.3+	Minimum PXG 8 Plane	39
SUN	Solaris 2.5	Minimum Creator 3D	45

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

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:

Voice: 800-382-3362
Fax: 248-729-3020
E-mail: bmorse@eta.com

The DYNAFORM Team
Engineering Technology Associates, Inc.