DYNAFORM-PC
User's Manual

A pre-processor compatible with

LS-DYNA/PC

Version 2.0

Release Date: April 11, 2002

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INTRODUCTION

The *DYNAFORM-PC Version 2.0* 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 950 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 component of DYNAFORM-PC's analysis engine is the LS-DYNA program developed and currently supported by the Livermore Software Technology Corporation (LSTC) of Livermore, California. This program is a general purpose, non-linear, dynamic, finite element analysis codes utilizing explicit and implicit solver approaches for fluid and solid structural problems. The code has 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 DYNAFORM-PC 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, DYNAFORM-PC 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.

Users can also take advantage of well-defined tooling surface data to predict the performance of a panel stamp in areas such as *cracking, wrinkling,* and *thinning*, in addition to predicting *skid mark* and *spring back* effects.

Availability:
DYNAFORM-PC is available for PC environments with Microsoft Windows 95, 98, 2000, NT.

Special features for DYNAFORM-PC Version 2.0 include:

GRAPHIC USER INTERFACE
DYNAFORM's new Graphic User Interface (GUI) is convenient and friendly. The functions are activated via mouse pick, command names and icons.

AUTOMESHING
Automeshing allows the user to define the maximum/minimum element size and chordal deviation to automatically mesh surface data through the different types of surface mesh available including an auto-mesh designed for tooling. This unique auto-meshing function promises to change the way people view modeling by eliminating 99% of the time required for tool meshing.

ANALYSIS MENU
DYNAFORM-PC allows users to either run the analysis directly from the DYNAFORM-PC window or output the LS-DYNA input file. The user can toggle on the Gravity Loading, Dyna Initial (DYNAIN), Adaptive Mesh and Spring Back functions.

DEFINE TOOLS
This menu allows the user to define more than the four conventional tool pieces (DIE, PUNCH, UPPER RING, and LOWER RING) for a simulation. It provides automatic positioning and interface definition for standard tooling setups. To access this function, the user selects DEFINE TOOLS from the TOOLS selection card when creating a database. Once in the menu, the user selects USER DEFINED TOOLS to define tools.

DEFINE BLANK
This menu in TOOL DEFINITION allows the user to change the shape of the blank through its convenient functions.

INTERACTIVE TRIMMING
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.

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.

DRAW BEAD DEFINITION
Provides LS-DYNA contact draw bead interface for the quick creation, modification and assignment of draw beads. The function also includes Draw Bead Force Prediction as an alternative for non-linear spring definition.

COPY ELEMENTS/NORMAL OFFSET
Automatically generates the mating part of the male/female tool surface and warns the user for the failed elements.

WORK PLANE
A Work Plane 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.

QUICK SETUP INTERFACE
The Quick Setup interface allows the user to set up a model and run it quicker than ever before. After meshing the die binder and blank, the program automatically all travel curves and mating parts.

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.

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 **DYNAFORM-PC 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 DYNAFORM-PC's capabilities, features and functions:

> Case 1. Binder Wrap Simulation
> Case 2. NUMISHEET'96 Case Study
> Case 3. Draw Die with Adaptive Mesh Case Study
> Case 4. Multiple Stage Forming Example---ALCAN Beer Can Case
> Case 5. Tool Surface Automeshing Examples
> Case 6. Tube Bending Simulation
> Case 7. Hydroforming Case Study

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 Preprocessor User's Manuals; several sections specific to sheet metal forming simulation are also included.

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

GENERAL DISCUSSION

1.1 GENERAL

DYNAFORM-PC Version 2.1 is a complete graphic user interface operated in a PC environment.

The DYNAFORM-PC's GUI interface allows the user to quickly access most functions anytime during the session. The tool bar, the menu bar, right mouse pull-down menus, etc. are available for user convenience.
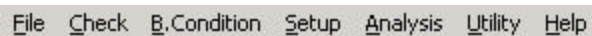
1.2 GRAPHICS DISPLAY WINDOW

DYNAFORM-PC separates the screen into several distinct regions. The regions are used to receive input from or display messages for the user. They are illustrated and described in the following sections.
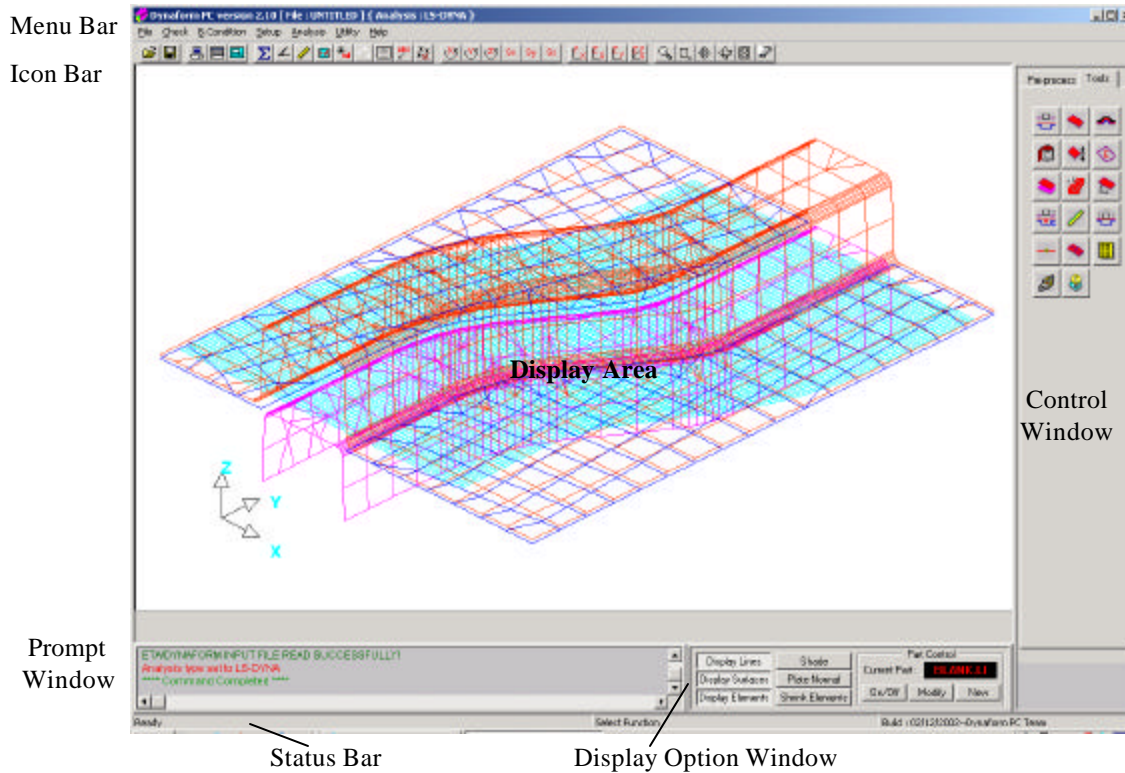
1. DISPLAY AREA
    Models and graphs are displayed in this area.

2. MENU BAR
    Functions and submenus are displayed in this area.

3. ICON BAR
    Gives the user easy access to the most used functions of DYNAFORM-PC.

4. CONTROL WINDOW
    This window contains two selection cards: Preprocess and Tool. The selection
    cards display icons representing various functions. The functions in each card are
    fully discussed in Chapter 4, 5, 6, 7 and 9. Also, if the user selects a function
    from one of the selection card or from the MENU BAR, a corresponding dialog
    window may display in place of the selection cards. When the function is
    completed, the selection cards redisplay.

5. DISPLAY OPTION WINDOW
    This group of commands controls the way the model is displayed. It also
    contains part control functions allowing the user to create, modify or change the
    current part. For a description of the display options and part control see
    DISPLAY OPTION WINDOW, Section 1.5.

6. PROMPT WINDOW
    DYNAFORM-PC displays color code comments and messages to the user here.
    A green prompt denotes feedback from a function or key the user has selected. A
    red prompt denotes an error or the beginning or ending of a routine. A blue
    prompt is displayed when the user must enter or select a value or key. All the
    messages displayed in the PROMPT WINDOW are saved and can be reviewed at
    anytime during the DYNAFORM session.

7. STATUS BAR
    The status bar, located below the PROMPT WINDOW, displays message
    prompts, the current function, the current part, any defined load and any defined
    SPC.

1.3   MENU BAR

File   Check   B.Condition   Setup   Analysis   Utility   Help

The user selects a menu by mouse pick. Descriptions for these menu options are located in
their respective chapters.

FILE MANAGER	Imports and exports data from DYNAFORM-PC.
CHECK	These functions validate models.
BOUNDARY CONDITION	Create and verify constraints and loads, defines initial velocities on finite element models.

SETUP	Contains various pre-processing utilities that are toggle switch activated.
ANALYSIS	Defines the parameters of the output.
UTILITY	DYNAFORM-PC's "tool kit."
HELP	Displays the information to contact DYNAFORM-PC Technical Support.

1.4   TOOL BAR

The tool bar is designed to give the user easy access to the most frequently used functions of DYNAFORM-PC. The user may simply click on these icons to activate the functions.

OPEN
This function allows the user to open a database.

SAVE
This function updates the current database.

DISPLAY PROPERTIES
This function accesses the DISPLAY PROPERTIES WINDOW allowing the user to alter the graphics display. It is also accessible via the SETUP menu.

REARRANGE WINDOWS
This function moves the CONTROL WINDOW to the opposite side of the GRAPHICS DISPLAY WINDOW. It is also accessible via the UTILITY menu.

MICROSOFT CALCULATOR
Clicking on this icon displays the Windows virtual calculator.

MODEL STATISTICS
This function displays the MODEL STATISTICS WINDOW. It is also accessible through the UTILITY menu.

ANGLE BETWEEN 3 POINTS/NODES
This icon accesses the ANGLE BETWEEN LINES function in the UTILITY menu.

DISTANCE BETWEEN 2 POINTS/NODES
This icon accesses the DISTANCE BETWEEN POINT/NODE function in the UTILITY menu.

IDENTIFY ELEMENT(S)
This icon accesses the IDENTIFY ELEMENT function of the ELEMENTS menu.

IDENTIFY NODE(S)/POINT(S)
This icon accesses the IDENTIFY NODES function of the NODES menu.

SHOW LINE AND DIRECTION
This icon accesses the SHOW LINES function of the LINE/POINT menu.

CREATE WORK PLANE/GRID
This icon accesses the CREATE WORK PLANE function of the LINE/POINT menu.
It is also accessible via the UTILITY menu.

DEFINE LABEL/POINTER
This icon accesses the DEFINE LABEL/ARROW function in the UTILITY menu.

DELETE ALL FREE NODES
This icon deletes all free nodes.

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

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

ROTATE ABOUT Z-AXIS
The displayed model will dynamically rotate about the global Z-axis when the cursor is
moved up or down.

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

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

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

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

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

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

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

CURSOR ZOOM
The user picks a point about which to zoom. The model is centred about this point and the user may move the cursor up or down to zoom in or out.

ZOOM BY WINDOW
The user defines the upper left corner of the zoom window by positioning the cursor on the display screen. The user presses the left most mouse button and releases it, and drags the cursor diagonally down until the desired window size is reached, then clicks the left most button once more. The section included in the window is displayed in full screen.

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 centre of the screen. Clicking the left most mouse button stops the command.

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 most mouse button stops the free rotation.

FILL
The model is re-scaled 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 highlighted entities from the screen such as those that are made with the commands SHOW LINE, BOUNDARY CHECK, ID ELEMENTS, DEFINE TITLE, etc.

1.5   DISPLAY OPTION WINDOW

The functions in the Display Option Window organize lines, surfaces and elements. They are described in the following pages.
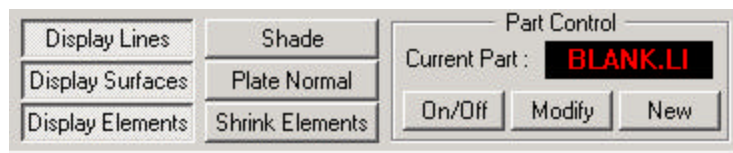
DISPLAY LINES (default)
Clicking this icon turns all line data in the database on. Clicking the icon again turns the line data off.

DISPLAY SURFACES (default)
This function turns on/off surfaces in a database.

DISPLAY ELEMENTS (default)
This function turns on/off elements in a database.

SHADE
Displays realistic rendering of objects using light sources. DYNAFORM-PC represents elements as if a physical light source was present. Elements that are not directly exposed to the light source are appropriately "shaded" to emulate real life shading.

PLATE NORMAL
This function turns on/off plate element normal with vector in a database.

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

1.6   PART CONTROL

A part in a DYNAFORM-PC 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.

1.6.1   CURRENT PART

Changes the current or active part. Lines, surfaces and in most cases elements will be automatically included in the CURRENT PART when created. The CURRENT PART name is displayed in its part color. Caution should be taken to make sure that the correct part is current before creating any new lines or elements.

1. Click in the current part field. DYNAFORM-PC prompts:

    PICK AN ELEMENT TO IDENTIFY THE PART

2. Once the target part has been selected it will become current and its name will be displayed in the current part field in its color.

ON/OFF
Toggles selected part(s) off or on.

1. DYNAFORM-PC prompts:

    SELECT AN ELEM TO IDENTIFY PART

- A part can be turned on/off by selecting a line, element, name or surface by selecting the option in the right click pull down menu or from the OPTION LIST ICONS. The user also has the option to turn on/off all parts.

- If SELECT BY NAME is selected, a Pop-Up window will display. The part can be toggled on/off by selecting its name from the list.

2. If the selected part(s) are currently on, then the part(s) are turned off or vice versa.

1.6.2   MODIFY

Selecting the MODIFY button will display the EDIT PART window in the bottom of the CONTROL WINDOW.
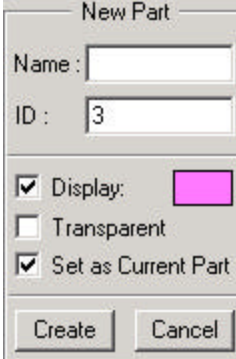
- Select a part in the parts list field or click the SELECT PARTS BY CURSOR button and pick a part in the GRAPHICS DISPLAY WINDOW.

- After the part is selected, the user can change the part name in the NAME field and/or change the part's ID in the ID field.

- After the part is selected, the user can also check DISPLAY to display it, check TRANSPARENT to make it transparent, check SET AS CURRENT PART to set it as the current part.

- Select the MODIFY button to accept the above change(s).

- After selecting a part, the user can click on the DELETE button to delete the part.

- After selecting a part, the user can click on the SUMMARY button to see the part's information in a pop-up PART SUMMARY window.

1.6.3   NEW

Displays the NEW PART window in the bottom of the CONTROL WINDOW.

- Input the new part name in the NAME field.

- Input the new part ID in the ID field.

- Check DISPLAY to display it.

- Check TRANSPARENT to make it transparent.

- Check SET AS CURRENT PART to set it as the current part.

- Click in the color block to set the part color.

- After completing the above steps, select CREATE to create the new part or CANCEL to abort the function.

- If CREATE is selected, DYNAFORM-PC prompts:

  OK, PART XXX IS ADDED WITH ID = XX IS ADDED TO DATABASE

1.7   OPTION LIST ICONS

During the DYNAFORM-PC session, the option list icons will present the user with the available options pertaining to the current command selection. These icons are displayed on the left side of the screen above the PROMPT WINDOW. To determine the function of the icon, place the mouse pointer over the icon. The tool-tip will provide a label for the icon indicating its function. Clicking the right-mouse button pull-down menu also accesses the same functions. The icons shown below are an example of this function.

1.8   SELECTION CARDS

The CONTROL WINDOW contains two selection cards: Pre-process and Tools. The selection cards display icons representing various functions. A function description is highlighted when the cursor is placed over the individual icon. Click on the tab label to move from one card to the other. The Pre-process selection card contains all the functions in order to perform geometry building. The pull-down menu provided within the card allows the user to select from one of the four options in geometry building: Lines/Points, Surfaces, Elements, and Nodes. Once the user selects one of the options, the corresponding functions display within the card. The functions in each card are fully discussed in Chapters 4, 5, 6, 7 and 9.
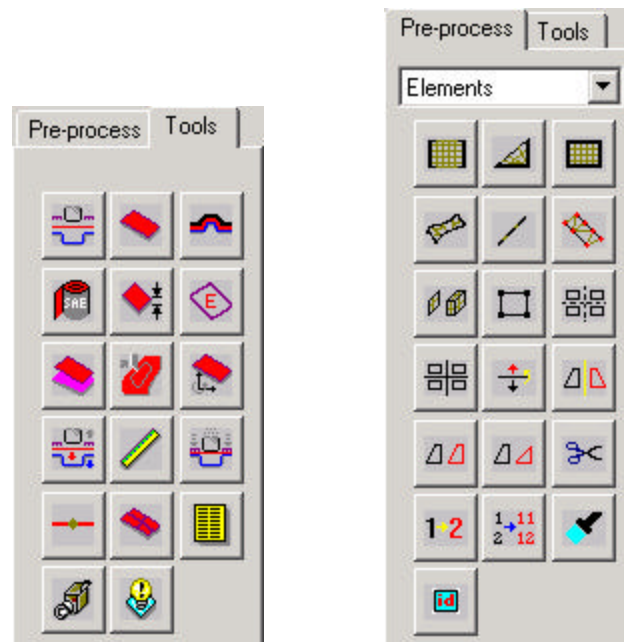
1.9   MOUSE FUNCTIONS

DYNAFORM-PC functions are accessible via the left mouse button. To access a function, the user selects the desired button or menu by moving the mouse pointer and depresses the left mouse button.

In the GRAPHICS DISPLAY WINDOW, the right mouse button activates a pull-down window with commonly and currently used functions.

In the PROMPT WINDOW, the right mouse button activates a pull-down window with font sizes. The user can select a value to define the font size displayed in the PROMPT WINDOW.

1.10   LOCAL COORDINATE SYSTEM

DYNAFORM-PC refers to a local or global coordinate system to translate, rotate, mirror, copy and generate points, lines, nodes and elements. When such a function is selected, the program automatically prompts the user to generate a local coordinate system designated as the U, V and W coordinate frame.

Either one, two or three reference point(s) is (are) required to establish a local coordinate system. The first reference point defines the local origin.

For a three point system the second reference point defines the local U axis which is extended from the first reference point. The third reference point, along with the first two reference points, defines the local U-V plane. The local V axis is defined in the U-V plane and perpendicular to the U axis. The local W axis is then defined according to the right hand rule perpendicular to the U-V plane.

For a two-point reference option, END SELECT in the SELECT OPTION BAR should be selected after the second reference point is defined. The local W axis lies along the vector from the origin to the second reference point and the V axis lies in the V-W plane. The U axis is defined by the right hand rule.

Note: All rotational commands (i.e., generating arcs, copy with rotating, etc.) execute around local W or global Z axis.

For a one-reference point system, the user selects a point or node as the local origin, and then selects one of the X, Y and Z options available to define the local W axis along one of the global axes.

Below are the series of prompts and steps involved in creating a local coordinate system whenever a local coordinate system is required (mirror lines/elements, transform nodes, etc.).

    1. DYNAFORM-PC prompts:

        DEFINE A LOCAL COORDINATE SYSTEM
        PICK POINT FOR THE ORIGIN

    2. After the origin is defined, the program prompts:

        PICK THE NEXT POINT

    3. After the coordinate system is defined, the program will prompt the user to accept/reject the displayed coordinate system.

1.11   RECOMMENDED NAMING CONVENTION (.df, .lin, .bin, etc.)

The protocol for naming files during a DYNAFORM-PC 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 screen.

Examples of suffixes include:

    1. DYNAFORM-PC database file name: *filename*.**df**
    2. DYNAFORM-PC geometry data file name: *filename*.**lin**
    3. IGES geometry data file name: *filename*.**igs**
    4. DYNAFORM-PC 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. DYNAIN input file name: *filename*.**din**
    9. DYNAFORM-PC binary result file: *filename*.**pp**
  10. VDA surfaces data file: *filename*.**vda**

Example: When reading in a line data file, DYNAFORM-PC 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 filename selection convenient and organizes the user's work directory.

1.12   SPECIFICATIONS

The standard version of DYNAFORM-PC has the following specifications per database:

150,000	LINES
450,000	POINTS
4,000	SURFACES
	-350,000 Edge Points (SURFACE)
	-160,000 Control Points (SURFACE)
250,000	GRIDS
250,000	ELEMENTS
1,000	PROPERTIES
1,000	PIDS

CHAPTER 2
GETTING STARTED

2.1   GETTING STARTED

1. Select the DYNAFORM-PC program in the PC window to activate it. Please see Figure 2.1.
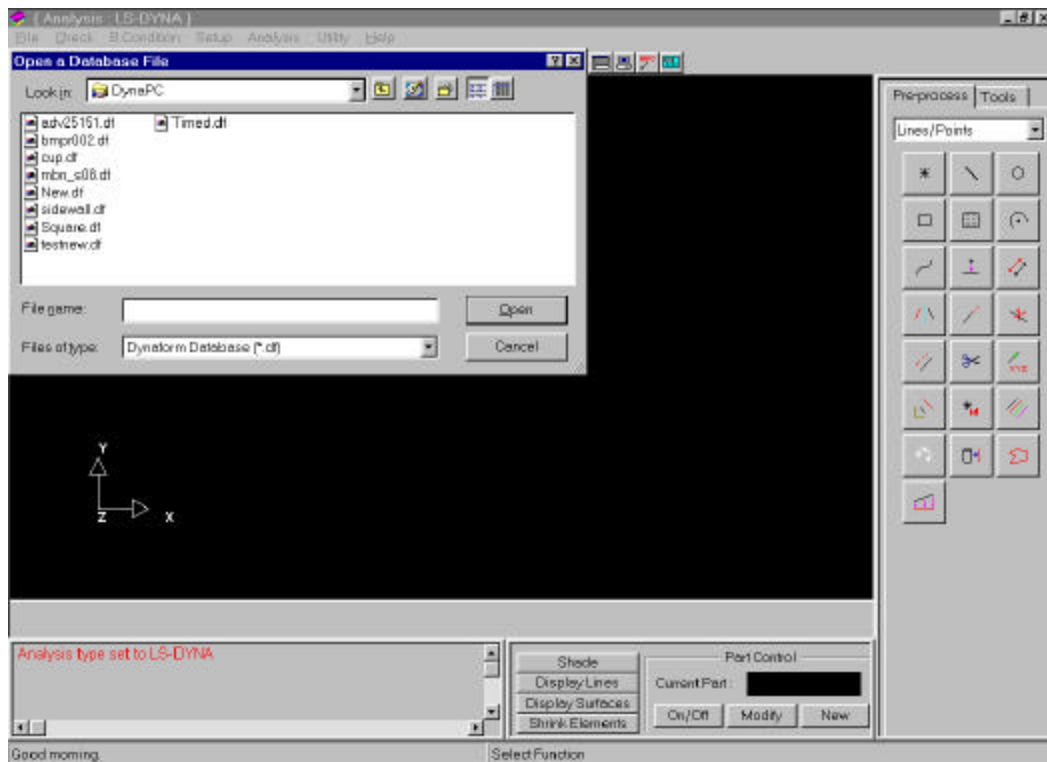
Figure 2.1 Getting Started

2. Once the program is activated, it automatically displays the OPEN A DATABASE FILE window.

3. The user supplies CAD-IGES data and VDA data that will be translated and read directly into DYNAFORM-PC.

4. CAD or model data is not supplied. Therefore, the user begins with an empty database and generates or "digitizes" line data into the drawing.

2.2   CREATE A DYNAFORM PC DATABASE FILE

1. Select NEW from the FILE MENU and DYNAFORM-PC displays the CREATE
    A DATABASE FILE window shown below. The default FILES OF TYPE is
    Dynaform Database (*.df). There is no alternative type. The user may type in any
    filename they choose and select OPEN to create the database file. Please see
    Figure 2.2.
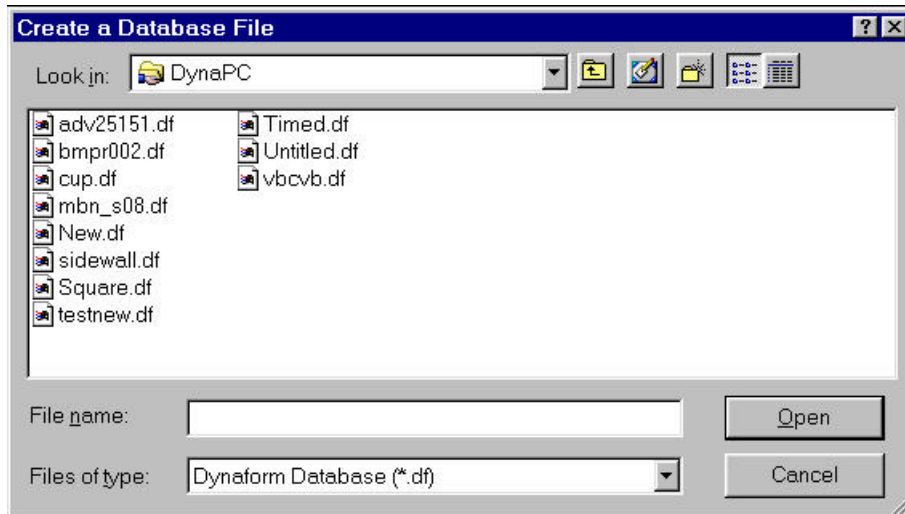
Figure 2.2 Create a Database File

2. Once the database file is created, DYNAFORM-PC displays the ANALYSIS
    SETUP menu as Figure 2.3.
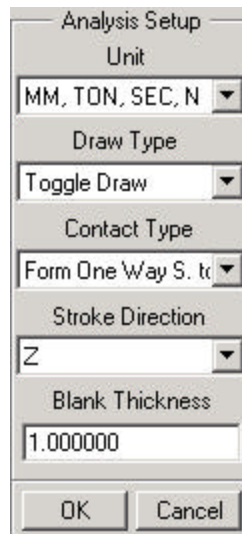
Figure 2.3 Analysis Setup

3. Select a unit type from the UNIT pull-down menu.

4. Select a draw type from the DRAW TYPE pull-down menu.

> **Note: Draw Type Option should relate to the type of machine used to produce the true work piece, which associates the directions of punch moving and binder loading. If you are not sure or face a new process, the user may select User Defined item. Refer to the *DYNAFORM-PC Application Manual* for description of draw type.**

5. After the other items in the menu have been considered, select OK.

2.3   READING GEOMETRY DATA

The user has several options when opening a file in DYNAFORM-PC. One of the ways the file can be opened is by dragging and dropping the desired file into the DYNAFORM-PC program window or short-cut icon. DYNAFORM-PC will automatically open the file if the extension is recognized and prompts the user for key parameters. The user can also double click on files with extensions that DYNAFORM-PC recognizes.   This will open DYNAFORM-PC (if unopened) and automatically load the file. The third choice is to open the desired file via the OPEN command in the FILE MENU. This will display the OPEN A DATABASE FILE window, which also opens automatically at the start of the program. This window controls the input and output of data in DYNAFORM-PC. Please see Figure 2.4
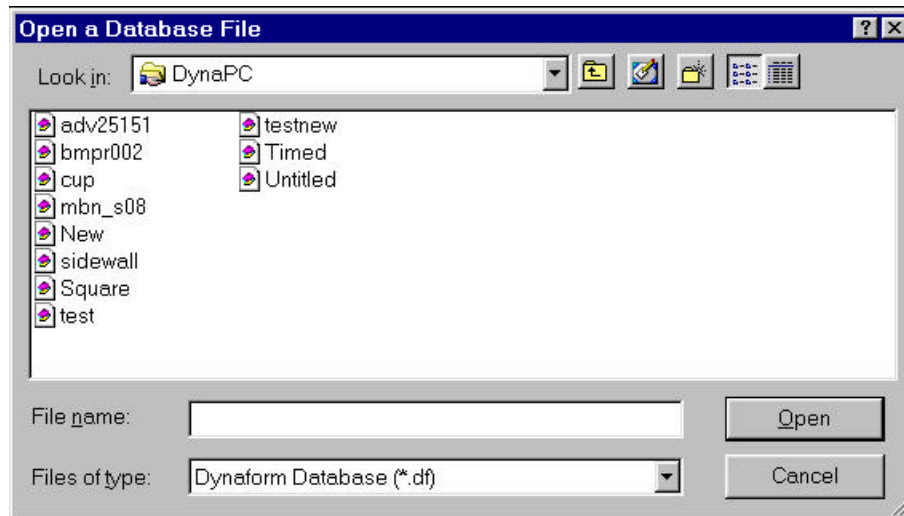
Figure 2.4 Open a Database File

LOOK IN
Allows the user to navigate directories when opening/saving files.

FILE NAME
Allows the user to specify the name of the file to open/save.

FILES OF TYPE

       Allows the user to specify the type of file to open/save. The available file types are displayed in the scroll-down window. The OPEN DATABASE WINDOW only displays the files that match the current FILES OF TYPE extension.

If the user accidentally enters the wrong extension, DYNAFORM-PC will open another window asking for the file type. Once the correct file type is selected, DYNAFORM-PC opens the file.

CHAPTER 3
FILE

The functions in the FILE menu are shown in Figure 3.1.
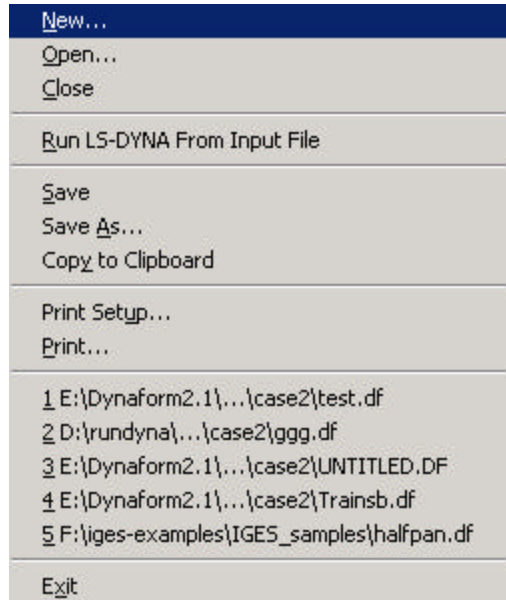
Figure 3.1 File Manager

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

3.1   NEW

This function allows the user to create a new database file. Please see Figure 3.2.
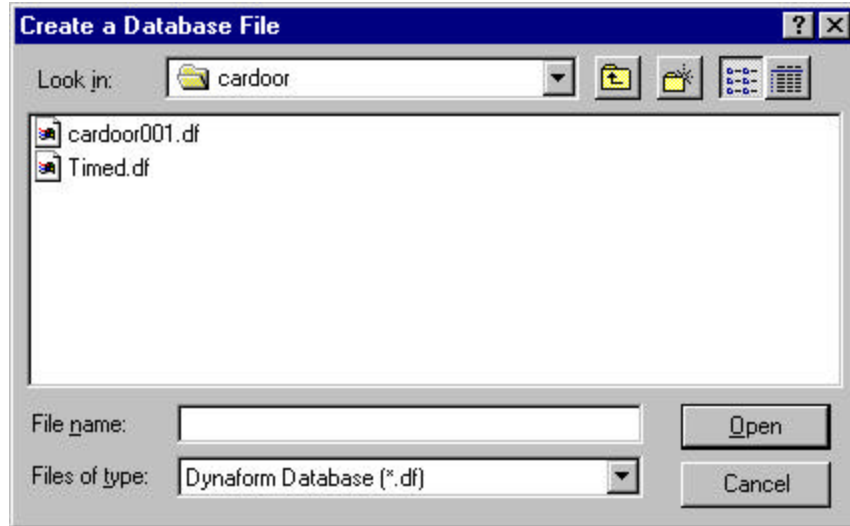
Figure 3.2 New a Database

- The default FILES OF TYPE is Dynaform Database (*.df). There is no alternative type. The user types a new file name in the FILE NAME field and presses the enter key or clicks the OPEN button.

- If a database file has been opened and modified in DYNAFORM-PC before, there will be a warning message notifying the user to save the opened file before creating a new one.

- If the filename typed by the user already exists or if the user selects a filename from the file list area of the dialog window, a prompt window appears asking whether the user wants to delete this file and create a new one or not.

3.2   OPEN

This function allows the user to open a database or other files. Please see Figure 3.3
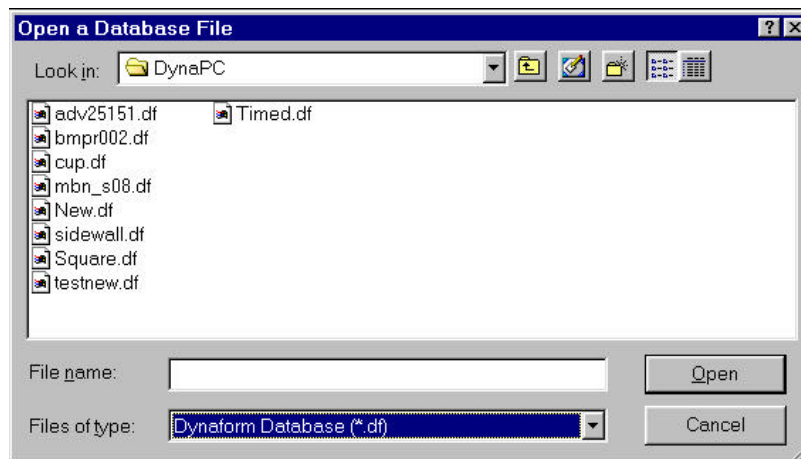
Figure 3.3 Open a Database

To specify a file type, the user selects an option in the FILES OF TYPE menu as Figure 3.4.
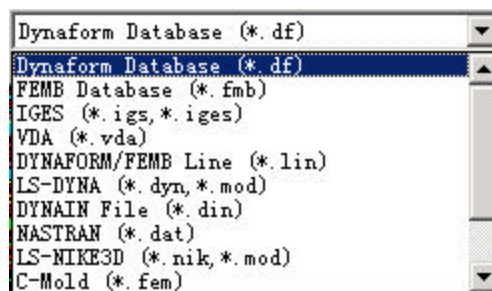
Figure 3.4 File Type

All the file types and their extensions are listed below.

Dynaform Database (*.df)
FEMB Database (*.fmb)
IGES (*.igs, *.iges)
VDA (*.vda)
DYNAFORM/FEMB Line (*.lin)
LS-DYNA (*.dyn, *.mod)
DYNAIN (*.din)
NASTRAN (*.dat)
LS-NIKE3D (*.nik, *.mod)
C-Mold (*.fem)
Moldflow (*.mfl)
IDEAS Universal (*.unv)
All files (*.*)

3.3   CLOSE

This function closes the database file. If needed, a message window will display a prompt for

the user to save his work. The user has the options to save or not to save the work before closing.

3.4   RUN LS-DYNA FROM INPUT FILE

This function is used for submitting an LS-DYNA analysis run from an LS-DYNA input file. The input file is not necessarily written from the current database. Therefore, it is not necessary to open any database file in order to submit the LS-DYNA run. The user selects CANCEL from the OPEN DATABASE FILE window at the start of the DYNAFORM-PC session and selects this function from the FILE menu. DYNAFORM-PC will prompt the user to select an LS-DYNA input file and specify the memory allocation for the analysis. The RUN LS-DYNA function in the ANALYSIS menu should be used to run the analysis from the current database.

3.5   SAVE

This function updates the current database or saves (writes) a copy of the DYNAFORM-PC database. The file will write over (erase) any previously saved version with the same filename. The date and time the file was last saved are displayed in the PROMPT WINDOW.

3.6   SAVE AS

Saves (writes) the specified file from the DYNAFORM-PC database. The user can also output the content of the database to several different files types. They include DYNAFORM database, DYNAFORM line data file, LS-DYNA database and others.

1. Once this function is selected, the SAVE DATABASE AS window appears. The user must specify where the file will be written, the file type and the filename.

2. The user selects a file type under FILE TYPE in the window and then select SAVE or CANCEL.

3.6.1   DYNAFORM DATABASE (*.df)

1. DYNAFORM-PC prompts:

    SAVE AS A DYNAFORM DATABASE

    ?   YES   Saves as a *.df database.
         NO   Returns the user to the current database.

3.6.2   DYNAIN (*.din)

Outputs the content in the current database as a DYNAIN input file for spring back, multi-stage tooling and mapping. This current database is not saved by this operation.

1. DYNAFORM-PC prompts:

    SAVE AS A DYNAIN FILE

    ?   YES   Continue to Step 2.
        NO   Returns the user to the current database.

2. Select OK to save as a DYNAIN database.

3.6.3   NASTRAN (*.dat)

Outputs the finite element model in the current database to a NASTRAN input file. The current database is not saved by this operation.

1. DYNAFORM-PC prompts:

    SAVE AS A NASTRAN INPUT FILE?

    ?   YES   Continue to Step 2.
        NO   Returns the user to the current database.

2. DYNAFORM-PC prompts:

    OUTPUT ONLY IN ACTIVE PARTS (YES/NO)?

    ?   YES   Only the active part(s) are saved as a NASTRAN file.
        No   All part(s) are written to the NASTRAN input file.
        Review   The user reviews the active parts in the display window.

3. DYNAFORM-PC prompts a window to define the NASTRAN file title.

4. ACCEPT, the NASTRAN file will be output.

3.6.4   DYNAFORM LINE (*.lin)

Outputs the line and surfaces in the current database to a DYNAFORM line data file. The database is not saved by this operation.

1. DYNAFORM-PC prompts:

    SAVE AS A DYNAFORM LINE DATA FILE?

    ?   YES   Continue to Step 2.
        NO   Returns the user to the current database.

2. DYNAFORM-PC prompts:

    WRITE LINES/SURFACES ONLY IN ACTIVE PARTS?

     ?   YES   Writes the line data only from parts currently turned on.

          NO   Writes all the line data from the current database.

3.6.5   IGES (*.igs)

The program outputs the lines and surfaces in the current database to an IGES file.   The current database is not saved by this operation.

    1. DYNAFORM-PC prompts:

       SAVE AS AN IGES DATA FILE?

       ?   YES   The program prompts the user through several Pop-Up windows to determine to write on lines and surfaces whether in the current part or not, enter the title for the file, ratio of model space to real world, select the measuring unit, and input the author's name and organization.

           NO   The program returns to the current database.

    2. DYNAFORM-PC prompts:

       WRITE LINES/SURFACES ONLY IN ACTIVE PARTS?

       ?   YES   Writes the IGES data only from parts currently turned on.

           NO   Writes all the line data from the current database.

    3. DYNAFORM-PC prompts a window as Figure 3.6 to let user select unit.
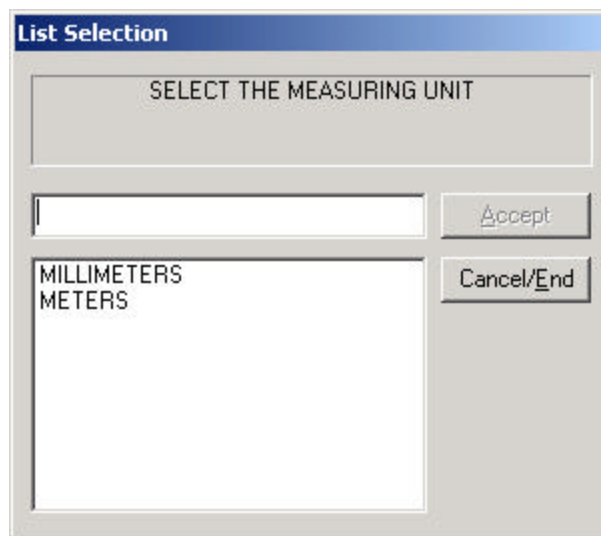
Figure 3.6 IGES File Unit

    4. DYNAFORM-PC prompts a window as Figure 3.7 to let user to select the line

type
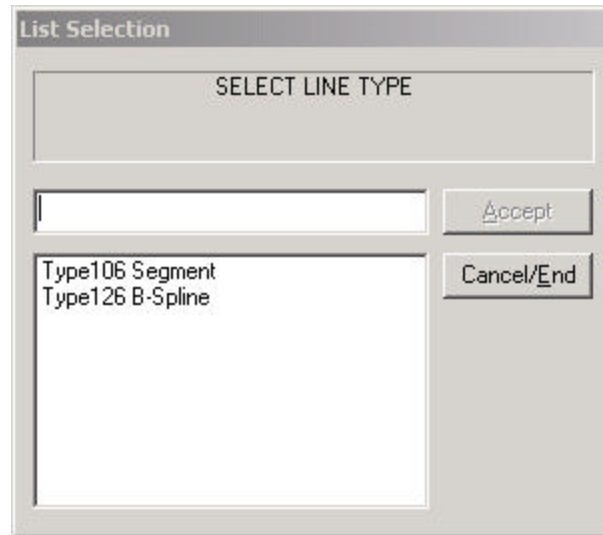
Figure 3.7 Line Type in IGES File

3.6.6   BITMAP (*.bmp)

The program saves the contents in the DISPLAY AREA as a bitmap file.   The file name should be given without .bmp extension.

3.6.7   JPEG (*.jpg, *. jpeg)

The program saves the contents in the DISPLAY AREA as a JPEG file. The file name should be given without the .jpg extension.

3.7   COPY TO CLIPBOARD

This function copies the currently displayed graphics from the DISPLAY AREA to the clipboard of the Windows operating system. The user can then paste it in other applications or documents.

3.8   PRINT SETUP

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 DYNAFORM-PC software. The user can define the printing defaults in Section 3.8.

3.9   PRINT

This function provides options for choosing the destination printer, paper size and paper orientation. See your operating system manual for more details.

3.10   EXIT

This function allows the user to exit the program. DYNAFORM-PC will prompt to save the current database.

LINES/POINTS

The functions of the LINES/POINTS menu are used to create a new set of line data or modify existing line data. DYNAFORM-PC has a 100,000 lines or 200,000 points per database limitation. This menu may be accessed through the PRE-PROCESS selection card located in the CONTROL WINDOW. The functions in Lines/Points Option menu are shown in Figure 4.1.

Figure 4.1 Lines/Points Option

A detailed description of each function is provided in the following pages.

4.1   CREATE WELD/CONTROL POINTS ✳

Creates weld/control points. The created point is included in the current part. There are two options as Figure 4.2 the user may select as follows:
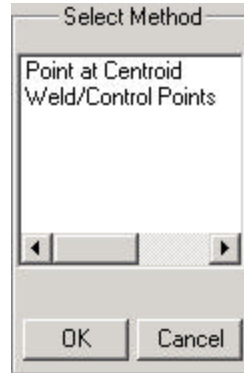
Figure 4.2 Create Points Options

4.1.1   POINT AT CENTROID

Creates a point at the line center, the created point is included in the current part.

    1.   DYNAFORM-PC prompts:

    SELECT LINE(S)

SELECT LINE OPTIONS TO CREATE POINT AT CENTROID:

    Select line by Cursor (default).

    Select line by Part. Please reference SELECT PART OPTION to select part.

    Select line by Drag Window.

    Select line by Muti-Point Region.

    Reject last Selection.

    Reject all selected line(s).

2.   Once the desired lines are selected and a point is created in the current part.

3.   END SELECT ⏎ will exit the function.

4.1.2   WELD/CONTROL POINTS

Creates weld/control points. The created point is included in the current part.

1.   DYNAFORM-PC prompts:

     SELECT A NODE FOR POINT LOCATION

2.   Select **abc** KEY IN XYZ and DYNAFORM-PC prompts via Pop-Up window as in Figure 4.3:

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ENTER X , Y, Z COORDINATE

Variable 1:	0.00000
Variable 2:	0.00000
Variable 3:	0.00000

Accept   Cancel/End
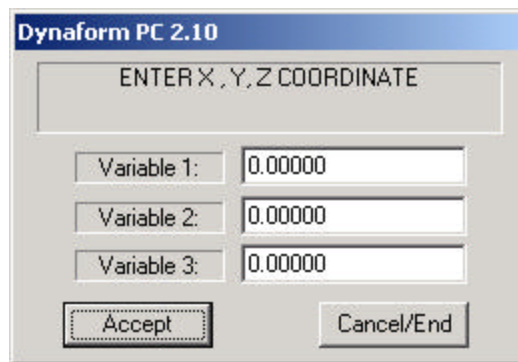
Figure 4.3 Key In XYZ

3.   Input the X, Y, Z coordinates, select ACCEPT and a point is created in the current part.

4.   END SELECT will exit the function.

OTHER SELECT NODE OPTIONS:

   Snap the Grid point(s).

   Select Surface Boundary Point.

   Select Surface Interior Point.

   Reject last Selection.

4.2   CREATE LINES

Draws lines between nodes or points through a combination of keyboard and mouse selection. The created lines will be included in the current part.

1. DYNAFORM-PC prompts:

    SELECT A NODE

2. Select **abc** KEY IN XYZ and DYNAFORM-PC prompts via Pop-Up window as in Figure 4.4:
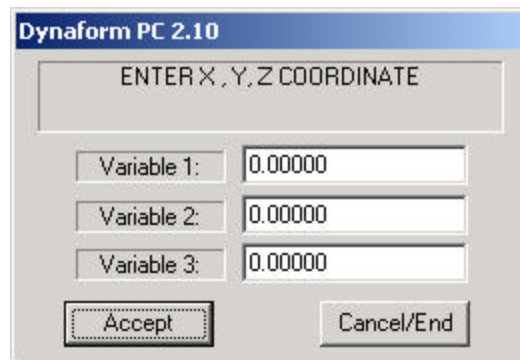
Figure 4.4 Key In XYZ

3. Input values and select ACCEPT to continue to the next window.

4. DYNAFORM-PC prompts:

    TO EXIT KEY ENTRY LOOP SELECT CANCEL/END

    •   Returns to Step 2.

5. Select CANCEL/END to close the dialog window.

6. DYNAFORM-PC prompts:

    SELECT A NODE

7. A line is created in the current part. END SELECT will exit the function.

OTHER METHODS TO CREATE LINES

1. **KEY IN XYZ BY BLOCK**

   Select KEY IN XYZ BY BLOCK from the OPTION LIST ICONS and a dialog window displays. A line can be generated through a list of points. Please see Figure 4.5.
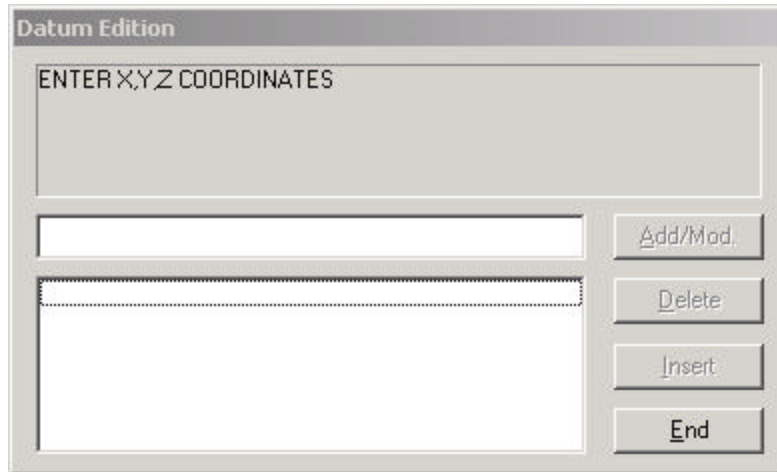
Figure 4.5 Enter XYZ Coordinates

- Once a point coordinate is inputted, the user can select ADD/MOD to add the record into the list box. The user may also input new data to update the record using this function.

- The user can delete a record by choosing one from the list box and selecting DELETE.

- Once a record is selected from the list box, the user can input a new record and select INSERT to insert the record into the list box.

- Select END to close the dialog window. A line is generated automatically in the current part.

2. **INCREMENT XYZ**
   This function is used to create a line by constructing a point of which the XYZ values are increments of the previously created point. Select INCREMENT XYZ and a dialog window displays as in Figure 4.6.
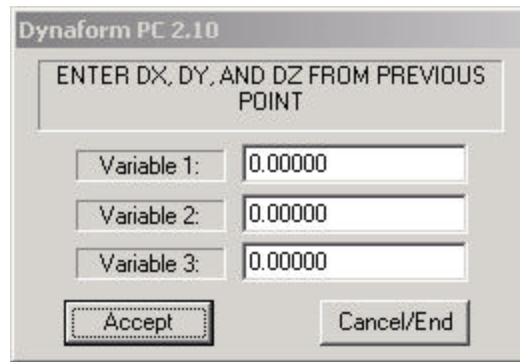
Figure 4.6 Enter Increment XYZ

3. **INCREMENT XYZ BY BLOCK**
   This function is used to create a line by constructing a set of points of which the XYZ values are increments of the previously created point. Select INCREMENT XYZ BY BLOCK and a dialog window is displayed:
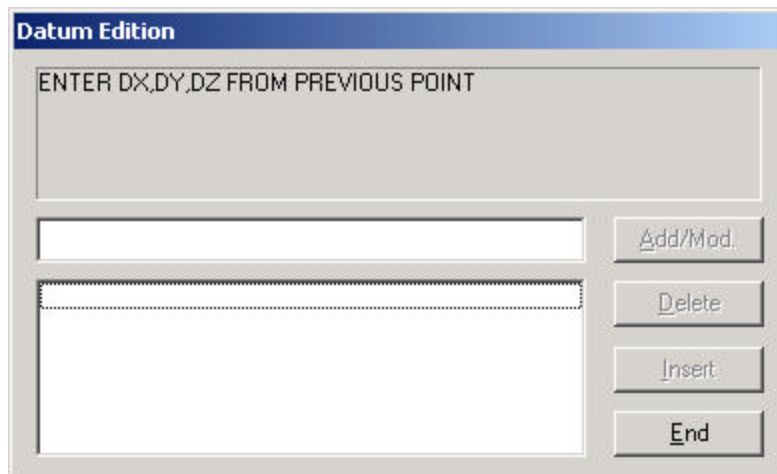
Figure 4.7 Increment XYZ by Block

4. **SELECT NODE**
   This function is used to create a line through node selection.

5. **SELECT POINT**
   This function is used to create a line through point selection.

6. **SNAP TO GRID**
   This function is used to create a lines through selected grid points. If no work plane is defined, the function is invalid.

7. **NEW LINE**

This function is used to finish the generation of a line and to create a new line.

8. **REJECT LAST**

This function is used to reject the previously created point.

4.3   CREATE CIRCLE ON WORK PLANE

Creates a circle through the center/point on the edge, center/radius, or through three points. The created circles will be included in the current part. The current work plane must be defined and displayed. The option window is displayed as Figure 4.8.
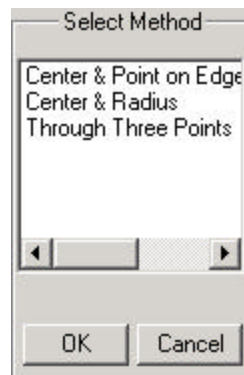
Figure 4.8 Options for Create Circle on Work Plane

4.3.1   CENTER & POINT ON EDGE

Creates a circle by selecting a center and edge point.

1. DYNAFORM-PC prompts:

   SELECT A CENTER GRID COORDINATE AND A GRID COORDINATE ON THE ARC

2. Repeat Step 1 to select another grid point as a point on the circle.

3. Select a grid point as the center point and the point coordinate is displayed.

4. Once a circle is created, END SELECT will exit this function.

4.3.2   CENTER & RADIUS

Creates a circle by establishing a center and radius. The created circles will be included in the current part.

1. DYNAFORM-PC prompts:

SELECT A CENTER GRID COORDINATE AND A GRID COORDINATE ON
THE ARC

2. Select a grid point on the current work plane and the point coordinate is
    displayed.

3. DYNAFORM-PC via Pop-Up window as Figure 4.9

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ENTER RADIUS OF CIRCLE

Variable 1:   0.00000

Accept   Cancel/End
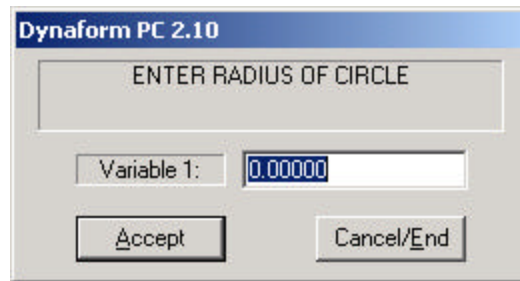
Figure 4.9 Radius of Circle

4. Input the radius value and select ACCEPT.

5. Once a circle is created in the current part, END SELECT will exit the function.

4.3.3   THROUGH THREE POINTS

Creates a circle though three nodes/points. The created circle will be included in the current
part.

1. DYNAFORM-PC prompts:

    SELECT 3 POINTS
    SELECT A POINT

2. Once the points are selected, END SELECT will create the circle and exit the
    function.

4.4   CREATE RECTANGLE ON WORK PLANE   □

Creates a rectangle by selecting two-grid points. The created rectangles will be included in the
current part. The current work plane must be defined and displayed.

1. DYNAFORM-PC prompts:

    SELECT TWO DIAGONAL GRID COORDINATES

2. Once the desired coordinates are selected, the rectangles will be created and DYNAFORM-PC will return to Step 1.

3. END SELECT will exit the function.

4.5   CREATE WORK PLANE/GRID

This function is used to create, modify and delete work plane.

See the details in Section 12.8 UTILITY/WORK PLANE.

4.6   CREATE ARC

Creates an arc through different methods as Figure 4.10. The created arc will be included in the current part.
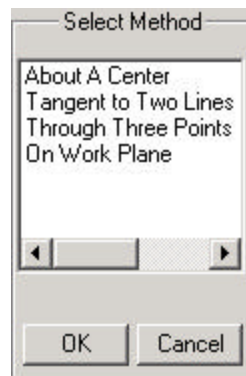
Figure 4.10 Options for Create Arc

4.6.1   ABOUT A CENTER

Generates an arc or circle around a centre point.

1. DYNAFORM-PC prompts:

    DEFINE A LOCAL COORDINATE SYSTEM
    PICK POINT FOR THE ORIGIN

    DYNAFORM-PC prompts the user to create a local coordinate system. A thorough explanation of how to create a local coordinate system is covered in Chapter 1 *General Discussion* section under "Local Coordinate System." The

    user also has the option to select the ⊕ Global Coordinate System and the

    option to select the Last Coordinate System.

2. Once the desired coordinate system is created, it will be displayed on the screen and the prompt will read via Pop-Up window:

DO YOU ACCEPT THE DISPLAYED COORDINATE SYSTEM?

? YES   Go to Step 3.
   NO   Returns to Step 1.

3. DYNAFORM-PC prompts via Pop-Up window as Figure 4.11.
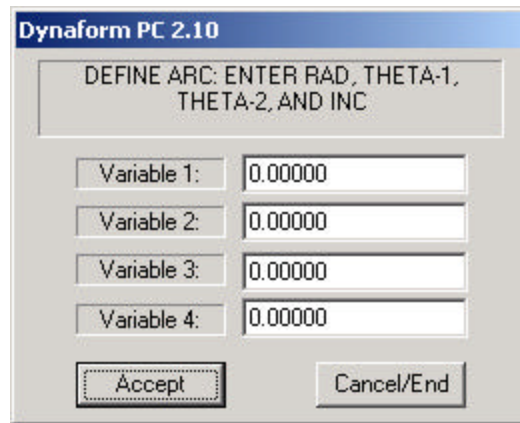
Figure 4.11 Define Arc about Centre

- RAD = radius of arc.
- THETA-1 = beginning angle of an arc or circle.
- THETA-2 = ending angle of an arc or circle.
- INC = angle increment between points (default is 5 degrees).
- Any real value greater than zero is valid. i.e., 100,0,360,10 is a circle with a radius of 100 units and an increment of 10 degrees between points.

Note: Arcs and circles are generated about the local W or global Z-axis.

4.6.2   TANGENT TO TWO LINES

Creates an arc tangent to two lines. The created arc will be included in the current part.

1. DYNAFORM-PC prompts:

SELECT LINE 1

2. After the point (or node) has been selected, DYNAFORM-PC prompts:

SELECT THE INTERSECTING LINE

3. DYNAFORM-PC prompts via Pop-Up window as Figure 4.12.
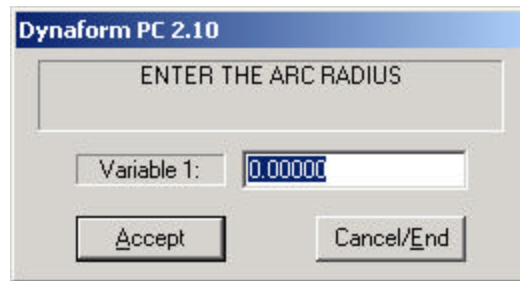
Figure 4.12 Enter the Arc Radius

> **Note: Any real value greater than zero is valid.**
4. An arc tangent to two lines is drawn. The lines do not need to be long enough to actually be in contact with the arc nor do the lines need to exist for this function to work (i.e. the user may select existing nodes to generate an arc).

4.6.3   THROUGH THREE POINTS

Creates an arc through three points/nodes. These three points cannot be co-linear. The created arc will be included in the current part.

1. DYNAFORM-PC prompts:

   SELECT 3 POINTS
   SELECT A POINT

2. After three nodes have been selected, the arc will be created.

3. END SELECT will exit the function

4.6.4   ON WORK PLANE

Creates an arc on the current work plane. The created arc will be included in the current part.

1. DYNAFORM-PC prompts:

   SELECT GRID COORDINATE FOR ARC CENTER

2. Once the desired coordinate is selected, DYNAFORM-PC prompts:

   SELECT GRID COORDINATE ON ARC

3. After selecting grid coordinate, DYNAFORM-PC prompts via Pop-Up window as Figure 4.13
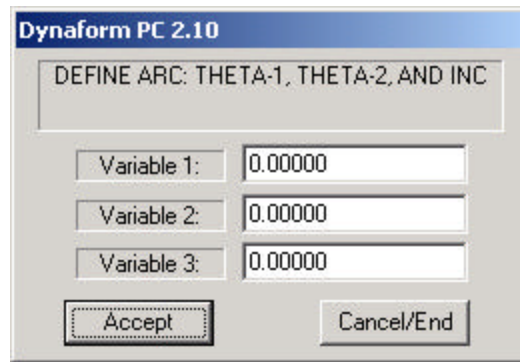
Figure 4.13 Define Arc on Work Plane

- THETA-1 = beginning angle of an arc or circle.
- THETA-2 = ending angle of an arc or circle.
- INC = angle increment between points (default is 5 degrees).
- Any real value greater than zero is valid. i.e., 100,0,360,10 is a circle with a radius of 100 units and an increment of 10 degrees between points.

Note: Arcs and circles are generated on the current work plane.

4.7   CREATE SPLINE

Creates a spline curve through multiple points, nodes or any combination of points and nodes. A minimum of three nodes is required to generate a spline curve.

1. DYNAFORM-PC prompts:

   SELECT A NODE

   Note: A thorough explanation of how to create a list of lines is covered in the CREATE LINES.

2. Repeat Step 1 until the desired points are selected and END SELECT will complete the spline and exit the function.

   - A minimum of three points/nodes is required to create the spline.
   - NEW LINE completes the spline and returns the user to Step 1.

4.8   ADD POINTS TO A LINE

Adds points to an existing line between two points that already exist on the line or at a selected cursor location. The options menu is displayed as Figure 4.12.
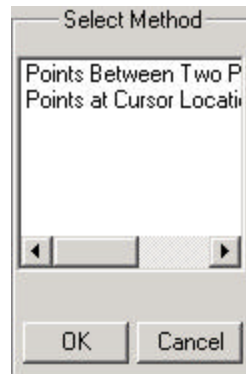
Figure 4.14 Methods for Adding Points

4.8.1   POINTS BETWEEN TWO POINTS

Adds points to an existing line between two points that already exist on the line.

1. DYNAFORM-PC prompts:

    SELECT A LINE TO ADD POINTS

    - If a line with only the two defined endpoints is selected, see Step 2.
    - If the chosen line has more than two defined points, see Step 3.
2. A dialog window displays as Figure 4.15, DYNAFORM-PC prompts.
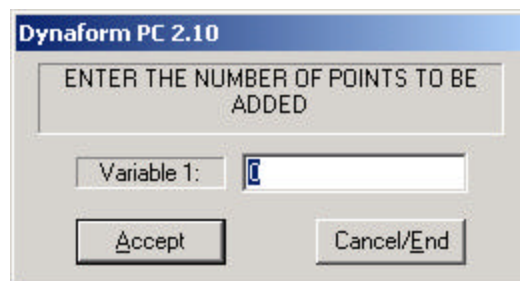
Figure 4.15 Input the Number of Points

    - Points are added to the selected line. DYNAFORM-PC returns the user to Step 2.

3. DYNAFORM-PC prompts:

    SELECT POINT (1) ON THE HIGHLIGHTED LINE

4. After point 1 is selected, DYNAFORM-PC prompts:

    SELECT POINT (2) ON THE HIGHLIGHTED LINE

5. DYNAFORM-PC prompts via Pop-Up window (see Figure 4.15):

ENTER THE NUMBER OF POINTS TO BE ADDED

- Points are added to the selected line and DYNAFORM-PC returns to step 1.

6. END SELECT will exit the function.

4.8.2   POINTS AT CURSOR LOCATION

Adds points to an existing line at selected cursor locations.

1. DYNAFORM-PC prompts:

   SELECT A LINE TO ADD POINTS

2. After selecting a line, DYNAFORM-PC prompts:

   SELECT A CURSOR LOCATION ON THE LINE

3. After the point is created, DYNAFORM-PC returns the user to Step 2.

4. END SELECT will exit the function.

4.9   REVERSE LINE DIRECTION

Each line has a direction defined from the first point to the last point. This function allows the user to reverse the line direction. Arrowheads on the line (presented during SHOW LINE, SPLIT LINE, etc.) which are located at existing points also indicate the line direction.

1. DYNAFORM-PC prompts:

   SELECT A LINE TO REVERSE

2. Once a line is selected, the direction is automatically reversed.

3. END SELECT will exit the function.

4.10   MIRROR LINES

Reflects a copy of desired lines across a local or global coordinate plane.

1. DYNAFORM-PC prompts:

   PICK LINES TO MIRROR
   SELECT LINE(S)

2. Once the desired lines have been selected, END SELECT will continue the function.

3. DYNAFORM-PC prompts via Pop-Up window:

    DEFINE A LOCAL COORDINATE SYSTEM
    PICK POINT FOR THE ORIGIN

    •   DYNAFORM-PC prompts the user to create a local system. A thorough explanation of how to create a local coordinate system is covered in Chapter 1 *General Discussion* section under "Local Coordinate System." The user also has the option to select the Global Coordinate System and the option to select the Last Coordinate System.

4. Once the desired coordinate system is selected, it will be displayed on the screen and the prompt will display via Pop-Up window:

    DO YOU ACCEPT THE DISPLAYED COORDINATE SYSTEM?

    ?   YES   Continues to Step 5.
        NO   Returns to Step 3.

5. DYNAFORM-PC prompts via Pop-Up window as Figure 4.16 to select a plane.
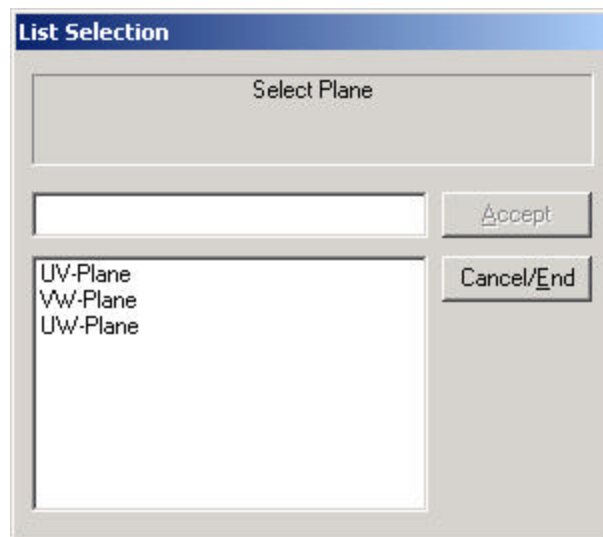
Figure 4.16 Select Plane

6. DYNAFORM-PC prompts whether or not to keep the lines in their original parts, please see Figure 4.17.
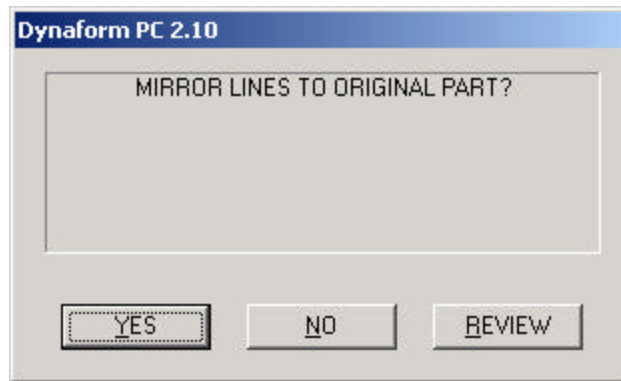
Figure 4.17 DYNAFORM-PC Question Window

?	YES	Places all selected lines in their source part.
	NO	Places all selected lines into the current part.
	REVIEW	Allows the user to review the model.

4.11 EXTEND LINE

Extends a line to a specified length, along the axis of the line at either end of the line.

1. DYNAFORM-PC prompts:

    SELECT A LINE TO EXTEND

2. Once the desired line has been selected, DYNAFORM-PC prompts:

    SELECT AN ENDPOINT TO EXTEND THE LINE

3. After selecting the endpoint, DYNAFORM-PC prompts via Pop-Up window as Figure 4.18.
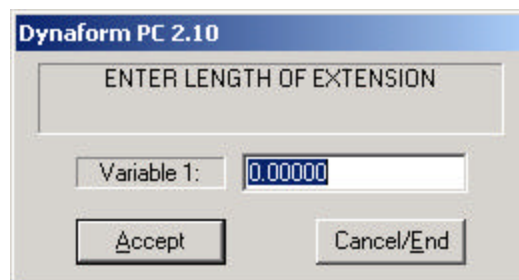
Figure 4.18 Enter Length of Extension

- The line will be extended at the endpoint selected to the specified length and will return to Step 1.

4. END SELECT will exit the function.

4.12   SPLIT LINE

Splits a line at the desired cursor location, nearest point or at an intersection with another line.

1.   DYNAFORM-PC prompts:

     SELECT A LINE TO SPLIT

2.   Once the desired line is selected, DYNAFORM-PC prompts:

     SELECT A POINT LOCATION TO SPLIT LINE

3.   After the endpoint is selected, DYNAFORM-PC returns the user to Step 1.

4.   END SELECT performs the operation and exits the function.

4.13   COMBINE LINES

Combines multiple lines into a single line with a new line number. The original lines will be deleted after they are combined. The line segments should be selected in a logical sequence to form the new line.

1.   DYNAFORM-PC prompts:

     SELECT LINES TO COMBINE

2.   Once the desired line is selected, END SELECT combines the selected lines and exits this function.

4.14   DELETE

Deletes the selected lines and weld/control points. Please see Figure 4.19.

Figure 4.19 Delete Options

4.14.1 DELETE LINES

Deletes selected lines.

> 1. DYNAFORM-PC prompts:
>
>   SELECT LINE(S)
>
> 2. Once the desired lines are selected, END SELECT deletes the selected lines and exits this function.

4.14.2 DELETE POINTS

Deletes selected points.

> 1. Once the desired points are selected, END SELECT deletes the selected points and exits this function.

4.15 MODIFY LINE

This function allows the user to modify a selected line by moving its points. Please see Figure 4.20 to get the methods of Modify Line.
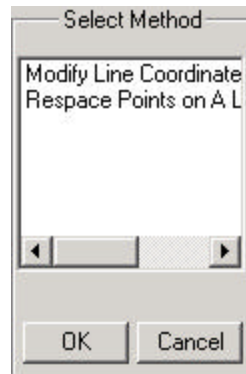
Figure 4.20 Method for Modify Line

4.15.1 MODIFY LINE COORDINATE DATA

Modify coordinate data of the selected line.

1. DYNAFORM-PC prompts:

    SELECT A LINE TO MODIFY

2. Once the desired line is selected, a Pop-Up window displays as Figure 4.21.
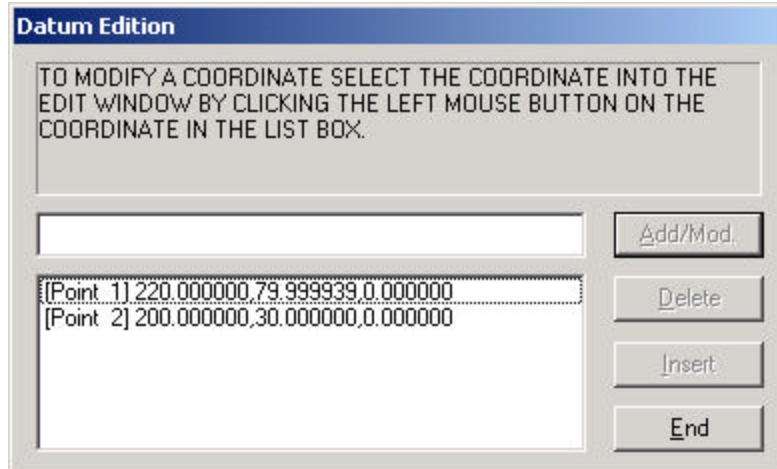
Figure 4.21 Datum Edition For Modify a Coordinate

3. Modify data through ADD/MOD, DELETE, INSERT and use END to close the dialog window and return Step 1.

4.15.2 RESPACE POINTS ON A LINE

Changes the number of given points on the selected line.

1. DYNAFORM-PC prompts:

SELECT A LINE FOR RE-SPACING POINTS

2. Once the desired line is selected, DYNAFORM-PC prompts via Pop-Up window as Figure 4.22.
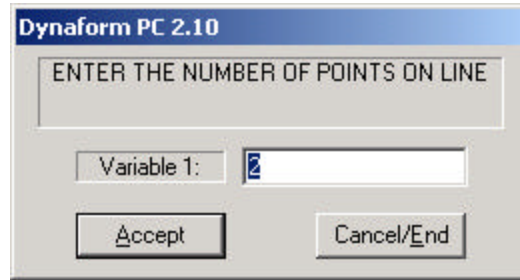
Figure 4.22 Enter the Number of Points on Line

3. Input the number of points from 3 to 500 on the selected line and DYNAFORM-PC returns to Step 1.

- Any integer greater than two must be entered. DYNAFORM-PC includes the end points of the line as part of the total count of points on the line.

4. END SELECT will exit the function.

4.16   RE-SCALE LINES

Scales selected lines or points with respect to any axis (global or local) using magnification factors in X, Y, Z (U, V, or W). A scale factor greater than 1 will magnify the line, whereas a scale factors less than 1 will reduce the line.

1. DYNAFORM-PC prompts:

   DEFINE A LOCAL COORDINATE SYSTEM
   PICK POINT FOR THE ORIGIN

   - DYNAFORM-PC prompts the user to create a local system. A thorough explanation of how to create a local coordinate system is covered in Chapter 1 *General Discussion* section under "Local Coordinate System." The user also has the option to select the Global Coordinate System and the option to select the Last Coordinate System.

2. Once the desired coordinate system is selected, it will be displayed on the screen and DYNAFORM-PC will prompt via Pop-Up window:

   DO YOU ACCEPT THE DISPLAYED COORDINATE SYSTEM?

? YES   Continue to Step 3.
NO   Returns to Step 1.
REVIEW   Allows the user to review the model.

3. DYNAFORM-PC prompts:

   SELECT LINES TO SCALE
   SELECT LINE(S)

4. After the desired lines have been selected, END SELECT will proceed to Step 5.

5. DYNAFORM-PC prompts via Pop-Up window:

   ENTER U, V, W SCALE FACTORS

   • User may now select desired scale factor.

     i.e., 1,-1,1 will result in lines being reflected in the negative Y-direction across the UV plane.

     i.e., 0.5,0.5,0.5 will result in the lines being shrunk by a factor of 0.5.

6. Once the scale factors are inputted, DYNAFORM-PC returns the user to Step 3. END SELECT will exit the function.

4.17   IDENTIFY NODES/POINTS $\ast_{id}$

Identifies nodes or points by cursor selection.

1. DYNAFORM-PC prompts:

   SELECT A NODE FOR IDENTIFICATION

2. When $\ast$ node is selected, the node number nearest to the cursor location will be printed on the screen. The following message will be echoed in the message window.

   NODE xxx X=x.xxx Y=y.yyy Z=z.zzz

3. If ○ point is selected instead of node, the point number nearest the cursor location will be printed on the screen. The following message will echo on the command line:

   POINT xxx X=x.xxx Y=y.yyy Z=z.zzz

4. If ⊢→ Adjust Value, DYNAFORM-PC prompts a window as figure 4.23.
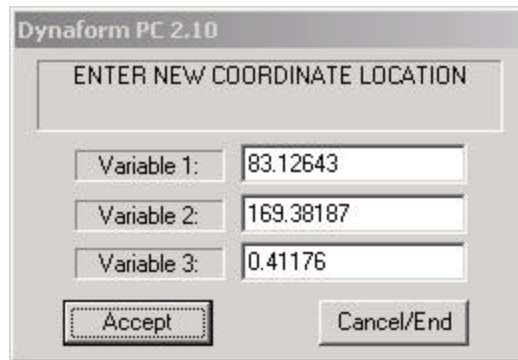
Figure 4.23 Enter New Coordinate Location

User is allowed to adjust the value of the selected node or point.

5. END SELECT will exit the function.

4.18   COPY AND TRANSFORM LINES

Generates a set of duplicate lines. The user can simultaneously translate and rotate the copied lines, there are several methods as Figure 4.24.
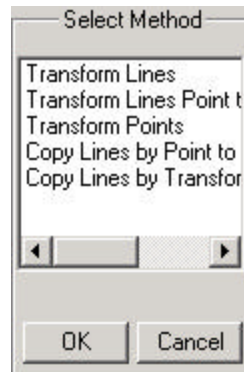
Figure 4.24 Methods for Copy and Transform Lines

4.18.1 TRANSFORM LINES

Translates or rotates selected line(s) to new locations.

1. DYNAFORM-PC prompts:

    DEFINE A LOCAL COORDINATE SYSTEM
    PICK POINT FOR THE ORIGIN

- DYNAFORM-PC prompts the user to create a local system. A thorough explanation of how to create a local coordinate system is covered in Chapter 1 *General Discussion* section under "Local Coordinate System." The user also has the option to select the Global Coordinate System and the option to select the Last Coordinate System.

2. Once the desired coordinate system is selected, it will be displayed on the screen and the prompt will display via Pop-Up window:

   DO YOU ACCEPT THE DISPLAYED COORDINATE SYSTEM?

   ?   YES   Continues to Step 3.
       NO   Returns to Step 1.

3. DYNAFORM-PC prompts via Pop-Up window;

   SELECT TRANSFORMATION OPTION

   ?   MOVE   The Pop-up window is Figure 4.25.
       ROTATE   The Pop-up window is Figure 4.26.

   ? MOVE

Figure 4.25 Enter UVW Increments

i.e., 0,0,1000 will copy the selected lines 1000 units along the W axis.

? ROTATE
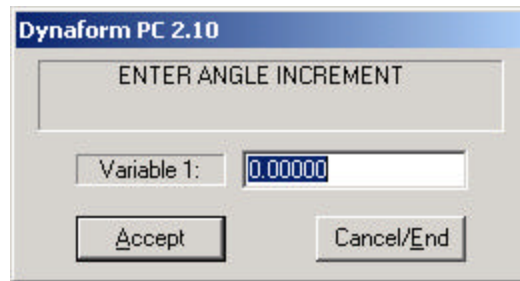
Figure 4.26 Enter Angle Increment

4. After the correct values have been entered, DYNAFORM-PC prompts:

    SELECT LINE(S)

5. Once the desired lines are selected, END SELECT will continue to the next step.

6. DYNAFORM-PC prompts:

    PICK SCREEN TO REPEAT TRANSFORMATION

    At this time user can pick screen to select a different set of lines, also can select

    options

     Select a different set of line with Cursor.

     Pop-up a window allowed user to input the Angle Increment (Rotate), or U, V, W Increments (Move).

     Define new coordinate system.

     Select a different set of lines with selecting point on the lines.

     Repeat the last operation.

     Reverse the last operation.

7. END SELECT will exit the function.

4.18.2 TRANSFORM LINES POINT TO POINT

Translates selected line(s) to new locations through selecting point to point.

---

1. DYNAFORM-PC prompts:

    DEFINE A DIRECTION VECTOR
    SELECT A NODE FOR ORIGIN

2. After the desired point is selected, DYNAFORM-PC prompts:

    SELECT A NODE FOR THE TARGET LOCATION

3. After the target point is selected, DYNAFORM-PC prompts:

    SELECT LINE (S) TO TRANSFORM
    SELECT LINE (S)

4. END SELECT will exit the function.

4.18.3 TRANSFORM POINTS

Translates or rotates selected points to new locations.

1. DYNAFORM-PC prompts:

    DEFINE A LOCAL COORDINATE SYSTEM
    PICK POINT FOR THE ORIGIN

    - DYNAFORM-PC prompts the user to create a local system. A
      thorough explanation of how to create a local coordinate system is
      covered in Chapter 1 *General Discussion* section under "Local
      Coordinate System." The user also has the option to select the Global
      Coordinate System and the option to select the Last Coordinate
      System.

2. Once the desired coordinate system is selected, it will be displayed on the screen
    and the prompt will display via Pop-Up window:

    DO YOU ACCEPT THE DISPLAYED COORDINATE SYSTEM?

    ?   YES   Continues to Step 3.
        NO   Returns to Step 1.
        REVIEW   Allows the user to review the model.

3. DYNAFORM-PC prompts via Pop-Up window:

    SELECT TRANSFORMATION OPTION

    ?   MOVE   The Pop-up window is Figure 4.27

ROTATE   The Pop-up window is Figure 4.28

? MOVE
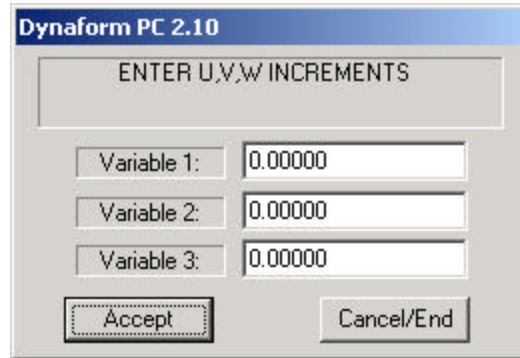
Figure 4.27 Enter UVW Increments

i.e., 0,0,1000 will copy the selected lines 1000 units along the W axis.

? ROTATE
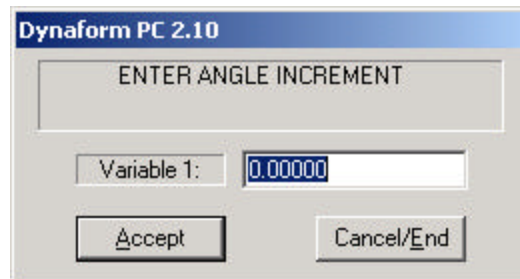
Figure 4.28 Enter Angle Increment

4. After the correct values have been entered, DYNAFORM-PC prompts:

SELECT POINT(S) TO TRANSFORM

5. Select the desired points to transform and END SELECT will exit this function.

4.18.4 COPY LINES POINT TO POINT

Copies selected line(s) to new locations through selecting point to point.

1. DYNAFORM-PC prompts:

DEFINE A DIRECTION VECTOR

SELECT A NODE FOR THE ORIGIN

2. After the desired point is selected, DYNAFORM-PC prompts:

SELECT A NODE FOR THE TARGET LOCATION

3. After the target point is selected, DYNAFORM-PC prompts:

SELECT LIN E(S) TO COPY
SELECT LINE

4. After the desired lines are selected, END SELECT will continue to the next step.

5. DYNAFORM-PC prompts via Pop-Up window as Figure 4.29.
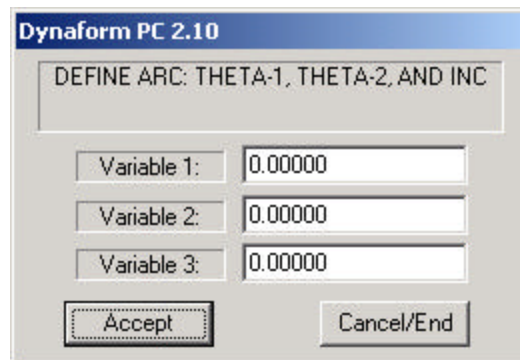
Figure 4.29 DYNAFORM-PC Question Window

?	YES	Places all copied lines in their original part.
	NO	Places all copied lines into the current part.
	REVIEW	Allows the user the review the model.

6. END SELECT will exit the function.

4.18.5 COPY LINES BY TRANSFORMATION

Generates a set of duplicate lines. The user can simultaneously translate and rotate the copied lines.

1. DYNAFORM-PC prompts via Pop-Up window as Figure 4.30.
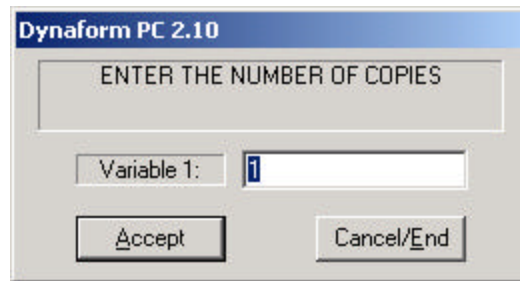
Figure 4.30 Enter the Number of Copies

2. Input a value and select ACCEPT.

3. DYNAFORM-PC prompts via Pop-Up window:

    SELECT TRANSFORMATION OPTION

    ?   MOVE
         ROTATE

4. DYNAFORM-PC prompts:

    DEFINE A LOCAL COORDINATE SYSTEM
    PICK POINT FOR THE ORIGIN

    •   DYNAFORM-PC prompts the user to create a local system. A
        thorough explanation of how to create a local a coordinate system is
        covered in Chapter 1 *General Discussion* section under "Local
        Coordinate System." The user also has the option to select the Global
        Coordinate System and the option to select the Last Coordinate
        System.

5. Once the desired coordinate system is selected, it will be displayed on the screen
    and the prompt will display via Pop-Up window:

    DO YOU ACCEPT THE DISPLAYED COORDINATE SYSTEM?

    ?   YES   Continues to Step 6.
         NO   Returns to Step 4.

6. If MOVE is selected as the transformation type, DYNAFORM-PC prompts as
    Figure 4.31

Figure 4.31 Enter UVW Increments

If ROTATE is selected as the transform type, DYNAFORM-PC prompts as Figure 4.32.
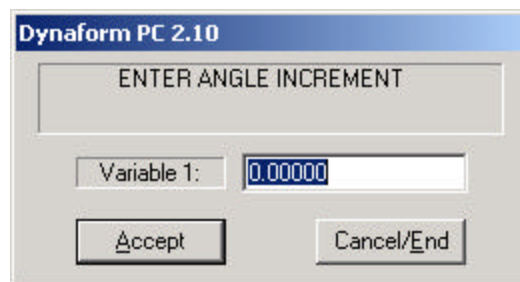
Figure 4.32 Enter Angle Increment

7. Input the desired values and select ACCEPT. DYNAFORM-PC prompts:

    SELECT LINES TO COPY
    SELECT LINE(S)

8. After the desired lines are selected, END SELECT will continue to the next step.

9. DYNAFORM-PC prompt via a Pop-Up window as Figure 4.33.
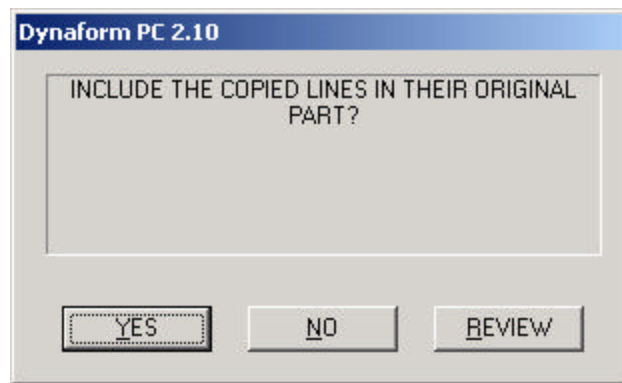
Figure 4.33 DYNAFORM-PC Question Window

?	YES	Places all copied lines in their original part.
	NO	Places all copied lines into the current part.
	REVIEW	Allows the user to review the model.

4.19   SHOW LINE AND DIRECTION

Identifies any existing line and its direction. The selected line will be highlighted (the starting point is labelled with a circle) and subsequent points are represented with arrowheads oriented in the line's direction. The number of points on the selected line is given in the PROMPT WINDOW.

     1. DYNAFORM-PC prompts:

        SELECT A LINE TO SHOW

     2. Once the desired line is selected, DYNAFORM-PC prompts:

        LINE XXX IN PART XXXX CONTAINS XXX POINTS
        XXX SEGMENTS AND IS XXXX UNITS LONG

     3. END SELECT will exit the function.

4.20   SECTION THROUGH LINE(S)

Computes and displays the intersection of a set of lines across a user defined U-V plane. The created section line will be included in the current part.
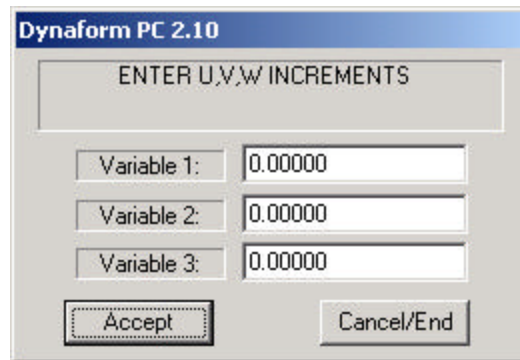
     1. DYNAFORM-PC prompts:

        DEFINE A LOCAL COORDINATE SYSTEM
        PICK POINT FOR THE ORIGIN

- DYNAFORM-PC prompts the user to create a local system. A thorough explanation of how to create a local coordinate system is covered in Chapter 1 *General Discussion* section under "Local Coordinate System." The user also has the option to select the Global Coordinate System and the option to select the Last Coordinate System.

2. Once the desired coordinate system is selected, it will be displayed on the screen and the prompt will display via Pop-Up window:

   DO YOU ACCEPT THE DISPLAYED COORDINATE SYSTEM?

? YES	Continues to Step 3.
NO	Returns to Step 1.
REVIEW	Allows the user to review the model.

3. DYNAFORM-PC prompts via Pop-Up window as Figure 4.34.

Figure 4.34 Enter Distance from Plane W

- The user may now input a distance along the W axis from the selected coordinate system's origin.

- Any real value is a valid response.

- The section line will be created at a distance 'W' along the UV plane.

4. DYNAFORM-PC prompts:

   SELECT A LINE

5. After selecting the desired lines, END SELECT completes the section creation and exits the function.

4.21   CREATE FE. BDY LINE

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

1. DYNAFORM-PC prompts via Pop-Up window as Figure 4.35.
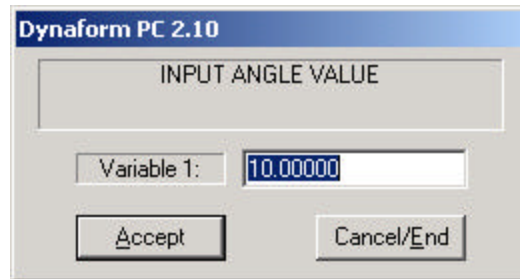
Figure 4.35 Input Angle Value

2. Input angle and select ACCEPT to finish the function. The generated line will be split at the corners whose angles are less than the split angle. The default value is 10.

4.22 PROJECT LINES

This function projects lines onto a selected surface.

1. DYNAFORM-PC prompts:

    SELECT SURFACES

2. Once the desired surfaces are selected, DYNAFORM-PC prompts:

    DEFINE PROJECTION VECTOR
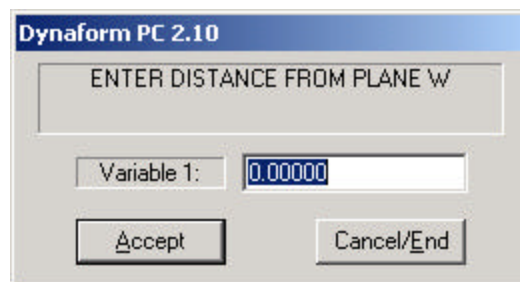   DEFINE LOCAL COORDINATE SYSTEM
    PICK POINT FOR ORIGIN

    •   DYNAFORM-PC prompts the user to create a local system. A thorough explanation of how to create a local coordinate system is covered in Chapter 1 *General Discussion* section under "Local Coordinate System." The user also has the option to select the Global Coordinate System and the option to select the Last Coordinate System.

3. Once the desired coordinate system is selected, it will be displayed on the screen and DYNAFORM-PC will prompt via Pop-Up window:

    DO YOU ACCEPT THE DISPLAYED COORDINATE SYSTEM?

    ?   YES   Continue to Step 4.
        NO   Returns to Step 2.

REVIEW   Allows the user to review the model.

4. DYNAFORM-PC prompts:

SET ENTITY FOR PROJECTION

5. The selected line will be projected on the surface automatically.

CHAPTER 5
SURFACES

The functions in the SURFACE menu allow the user to create and modify surfaces in a DYNAFORM-PC database. The IGES translator has been updated to convert IGES surface data into the DYNAFORM-PC format. DYNAFORM-PC line data file format has been expanded to include surface data. This menu may be accessed through the PRE-PROCESS selection card located in the CONTROL WINDOW. The functions are organized as Figure 5.1

Figure 5.1 Surface Menu

A detailed description of each function is provided in the following pages.

5.1   CREATE 2 LINE SURFACE

Creates a surface from two selected lines (curves). The operation is similar to 2-Line Mesh for elements.

   1. DYNAFORM-PC prompts:

   SELECT A LINE

   . Line allows the user to select lines (default).

   Line Segment allows the user to select several line segments to form a line.

   2. Upon selection of the second line, the surface will be generated.

      Note: **The direction of the surface dictates the direction of the plate elements generated from the surface.**

   3. END SELECT will exit the function.

5.2   CREATE 3 LINE SURFACE

Creates a surface from three selected lines (curves). The operation is similar to 3-Line Mesh for elements.

   1. DYNAFORM-PC prompts:

   SELECT 3 LINES AS BOUNDARY
   SELECT A LINE

      Note: Select lines in a clockwise or counter clockwise direction.

   2. Upon selection of the third line, the surface will be generated.

   3. END SELECT will exit the function.

5.3   CREATE 4 LINE SURFACE

Creates a surface from four selected lines (curves). The operation is similar to 4-Line Mesh for elements.

   1. DYNAFORM-PC prompts:

   SELECT 4 LINES AS BOUNDARY
   SELECT A LINE

Note: Select lines in a clockwise or counter clockwise direction.

2. Upon selection of the fourth line, the surface will be generated.

3. END SELECT will exit the function.

5.4   CREATE REVOLUTION SURFACE

Creates a surface by rotating a selected line (the generatrix) 360º about a vector (the axis of revolution).

1. DYNAFORM-PC prompts:

   SELECT THE AXIS OF REVOLUTION
   SELECT THE FIRST POINT
   SELECT A LINE FOR THE AXIS OF REVOLUTION

   Select Line with Two Points.

   Create a line with Two Points.

2. Upon selection of the line for the axis of revolution, DYNAFORM-PC displays a vector representing the axis of revolution. DYNAFORM-PC prompts:

   SELECT A LINE AS THE GENERATRIX OF REVOLUTION

3. Once the desired line is selected, a Pop-Up window is displayed; Figure 5.2.
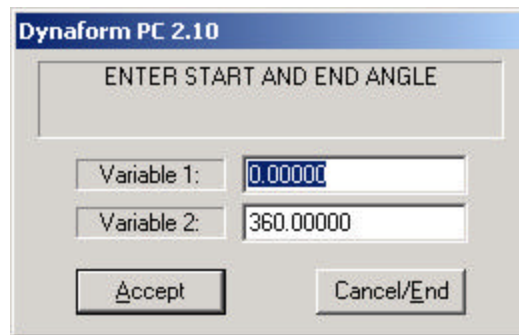
Figure 5.2 Enter Start and Angle

4. Input start angle, end angle and the surface will be generated automatically when ACCEPT is selected.

5. END SELECT will exit the function.

5.5   CREATE SURFACE BOUNDARY

Creates a boundary line around a selected surface.

1. DYNAFORM-PC prompts:

    SELECT SURFACE TO CREATE BOUNDARY LINES

2. Upon surface selection, the boundary line will be automatically generated.

3. END SELECT will exit the function.

5.6   CREATE SURFACE SECTION LINE

Creates section lines on the selected surface.

1. DYNAFORM-PC prompts:

    SELECT SURFACE

    SELECT SURFACE OPTIONS

    Select surface by Cursor on the surface.

    Select surface by Keying In Surface Number.

    Select surface by Drag Window.

    Select surface by Part. Please reference SELECT PART OPTION to select part.

    Select Displayed Surface.

    Select line by Muti-Point Region.

2. DYNAFORM-PC labels the edge numbers on the surface and prompts the user for the number of section lines along edges 1 and 2 via Pop-Up window as Figure 5.3.
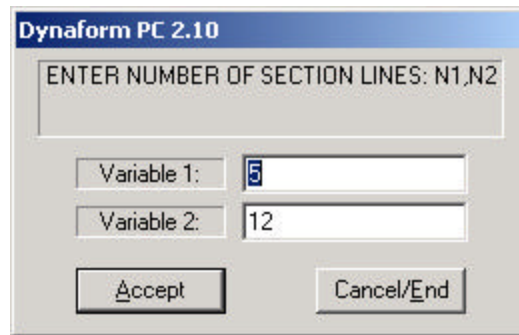
Figure 5.3 Enter Numbers of Section Lines

    3. END SELECT will exit the function.

5.7   DELETE SURFACE ✂

Deletes the selected surfaces from the database.

    1. DYNAFORM-PC prompts:

        SELECT SURFACES

    2. END SELECT will delete the surfaces and exit the function.

5.8   RESPACE SURFACE SECTION LINE

Changes the number of the section line on the selected surface.

    1. DYNAFORM-PC prompts:

        SELECT SURFACE

    2. Once the surface is selected, DYNAFORM-PC prompts via Pop-Up window as Figure 5.4.
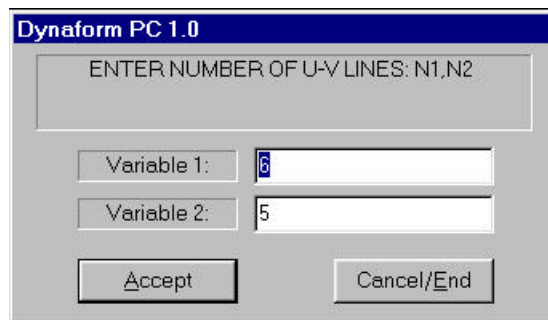
Figure 5.4 Enter Numbers of U-V Lines

- Any integer greater than two must be entered.

3. END SELECT will exit the function.

5.9 REVERSE SURFACE NORMAL

Reverses the normal direction of the selected surface. The surface normal controls the normal direction of the plate elements when meshing.

1. DYNAFORM-PC prompts:

    SELECT SURFACES

2. END SELECT will complete and exit the function.

5.10 SCALE SURFACE

Scales a selected surface with respect to any axis (global or local) using magnification factors in X, Y, Z (U, V or W). The scale factor greater than 1 will magnify the line, and the scale factor less than 1 will reduce the line.

1. DYNAFORM-PC prompts:

    SELECT SURFACES TO SCALE
    SELECT SURFACES

    - After the desired lines have been selected, END SELECT will continue to Step 2.

2. DYNAFORM-PC prompts via Pop-Up window:

    DEFINE A LOCAL COORDINATE SYSTEM
    PICK POINT FOR THE ORIGIN

    ● DYNAFORM-PC prompts the user to create a local coordinate system. A thorough explanation of how to create a local coordinate system is covered in Chapter 1 *General Discussion* section under "Local Coordinate System." The user also has the option to select the Global Coordinate System and the option to select the Last Coordinate System

3. Once the desired coordinate system has been defined, DYNAFORM-PC prompts via Pop-Up window as Figure 5.5.
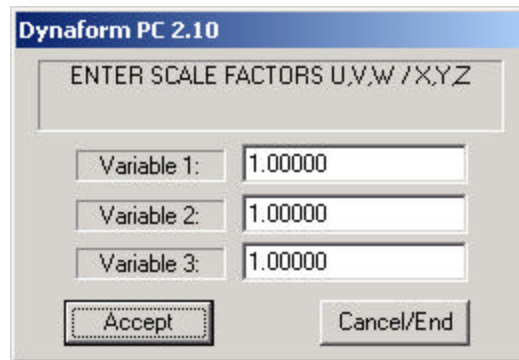
Figure 5.5 Enter Scale Factors UVW/XYZ

- The user may now select the desired scale factor:

  i.e., 1,-1,1 will result in lines being reflected in the negative Y-direction (across the UV plane).

  i.e., 0.5,0.5,0.5 will result in the lines being shrunk by a factor of 0.5.

4. END SELECT will exit the function.

5.11   CREATE SURFACE INTERSECT

Creates a line at the intersection of two selected surfaces.

1. DYNAFORM-PC prompts:

   SELECT THE FIRST SURFACE

2. After selecting the first surface, DYNAFORM-PC prompts the user for the second surface:

   SELECT THE SECOND SURFACE

3. Once the second surface is selected, END SELECT will create the intersection of the two surfaces. The newly created line is included in the current part.

5.12   SPLIT SURFACE

Splits the selected surface into two surfaces at a specified location.

1. DYNAFORM-PC prompts:

SELECT SURFACE

2. Upon surface selection, DYNAFORM-PC prompts:

SELECT SECTION LINE TO SPLIT SURFACE

SPLIT SURFACE OPTIONS

SELECT TWO BOUNDARY POINTS

- The user may split the selected surface at two-selected boundary points.

  SELECT FIRST POINT ON BOUNDARY
  SELECT SECOND POINT ON BOUNDARY

- DYNAFORM-PC splits the surface upon selection of the second boundary point.

RE-SPACE SECTION LINES

- The user may re-space the surface section lines. The user may then select a section line along which to split the surface.

  ENTER THE NUMBERS OF SECTION LINES: N1, N2

- N1 or N2 cannot be greater than 199.

- Once N1 and N2 section lines have been entered, the user may select the desired section line.

SECTION LINES

- Once the section line is selected, the surface is split.

SURFACE OF INTERSECTION

- The user may split a surface at the intersection of a second surface.

- Upon selection of the second surface, the first surface will be split at the intersection of the two surfaces. The user then has the option to split the second surface at the same intersection.

SPLINE LINE

- The user may pick up to 500 locations on the surface defining either a piecewise linear curve or a spline.

> **Note:** **The user must end selection of spline curve points with the selection of END SPLINE CLOSED or END SPLINE OPEN from the right click pull down menu.**

LINEAR SEGMENT

SELECT NEXT POINT FOR LINEAR SEGMENTS

3. END SELECT completes the split operation.

5.13   TRIM SURFACE

Defines curves for trimming edges and/or cutting holes on a selected surface. The defined curves may be poly-lines, b-spline lines, circles, piecewise linear segments or intersection lines of two surfaces.

1. DYNAFORM-PC prompts:

    SELECT A SURFACE TO TRIM

2. Upon selection of the surface, DYNAFORM-PC prompts:

    SELECT A CURSOR LOCATION TO BEGIN THE TRIM LINE

3. OPTION LIST ICONS is activated as

    SELECT POINT

    The user may select any location on the selected surface via mouse pick to define the linear segment (default). Select END TRIM LINE CLOSED or END TRIM LINE OPEN. Upon completion of point selection and then proceed to Step 4.

    CREATE SPLINE LINE

    SELECT THE FIRST CURSOR LOCATION ON THE TRIM LINE

    Cuts spline curves on the selected surface. The user may select any location on the selected surface via mouse pick to define the spline. Select END TRIM LINE CLOSED or END TRIM LINE OPEN. Upon completion of point selection, proceed to Step 4.

    CREATE CIRCLE

    SELECT THE CENTER OF THE CIRCLE

Cuts circles on the selected surface. The user may select any location on the selected surface via mouse pick to define the circle. The user may pick anywhere on the surface to define the center.

SELECT A POINT ON THE CIRCLE

The second point selected on the surface defines the radius of the circle and the circle will then be drawn. Once this is complete, proceed to Step 4.

SELECT INTERSECTION

Trims out the intersection of two surfaces.

SELECT SURFACE FOR TRIM OPERATION

Note: This surface will be called the primary surface for ease of reference in this section.

SELECT THE SECOND SURFACE

Upon selection of the second surface, a boundary line will be drawn between the two intersecting surfaces.

Note: This surface will be called the secondary surface for ease of reference in this section. Once this step is completed, see Step 4.

END TRIM LINE CLOSED

Defines a closed linear segment/spline curve passing through all the points selected.

END TRIM LINE OPEN

Defines a linear segment/spline curve passing through all the points selected with the first and last point, selected as the starting and ending points of the curve/segments.

4. DYNAFORM-PC prompts:

SELECT A CURSOR LOCATION ON THE TRIM LINE

5. Once the cursor locations are selected, click DEFINE NEW SURFACE BOUNDARY. The user must select an outer and inner boundary to complete the trimming function.

6. DYNAFORM-PC prompts:

SELECT SURFACE BOUNDARY AS THE OUTER BOUNDARY FOR THE TRIMMED SURFACE

- There is exactly one and it lies within the domain (the surface to be trimmed). In particular, it can be the boundary curve of the domain.

7. Select the outer closed surface boundary of the intended trimmed surface (cursor select). DYNAFORM-PC prompts:

   SELECT A CLOSED CURVE AS AN INNER BOUNDARY

   - There can be any number of them, including zero. The set of inner boundaries satisfies two criteria:

     A. The curves as well as their interiors are mutually disjoint.
     B. Each curve lies in the interior of the outer boundary

   Note: END SELECT is required to end this operation.

   Note: NO INNER BOUNDARY is an available option. The surface outside of the outer boundary will be trimmed.

8. DYNAFORM-PC prompts:

   DELETE THE ORIGINAL SURFACE?

   | ? | YES | Generates a trim surface. |
   | | NO | Generates a trim surface and retains the original surface. |

9. END SELECT will exit the function.

5.14   COPY SURFACES

Copies selected surface data to a new location.

1. DYNAFORM-PC prompts:

   SELECT SURFACES

   Note: END SELECT will complete surface selection.

2. DYNAFORM-PC prompts via Pop-Up window:
   DEFINE A LOCAL COORDINATE SYSTEM
   PICK POINT FOR THE ORIGIN

   - DYNAFORM-PC prompts the user to create a local coordinate system. A thorough explanation of how to create a local coordinate system is covered in Chapter 1 *General Discussion* section under "Local Coordinate System." The user also has the option to select the Global Coordinate System and the option to select the Last Coordinate System

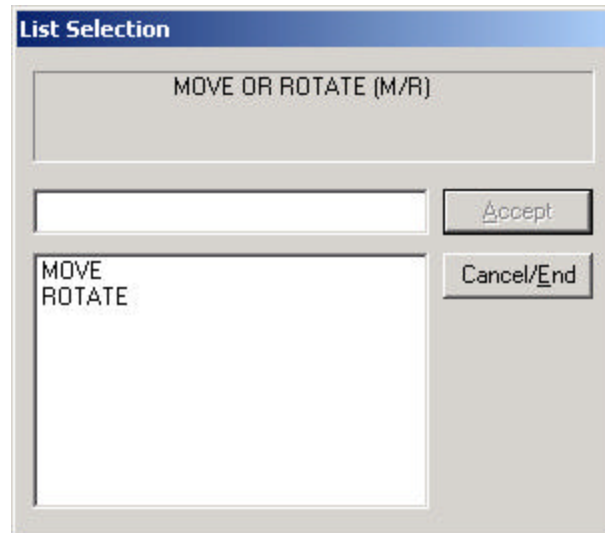3. Once the desired coordinate system has been defined, DYNAFORM-PC prompts via Pop-Up window as Figure 5.6.
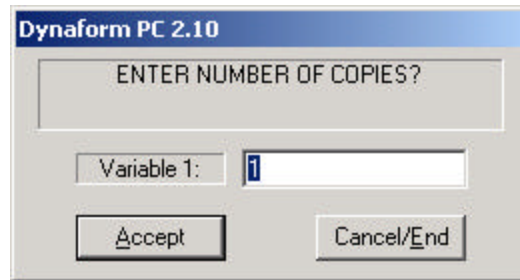
Figure 5.6 Enter the Number of Copies

4. Once the desired number of copies has been entered, DYNAFORM-PC provides different methods to copy as Figure 5.7.

Figure 5.7 Move or Rotate

?   MOVE
    DYNAFORM-PC prompts:

    ENTER U, V AND W INCREMENTS

    • i.e. 0,0,1000 will copy the selected surface(s) 1000 units along the W axis.

?   ROTATE
    DYNAFORM-PC prompts:

    ENTER ANGLE INCREMENT

• Enter angle increment (right hand rule) about the W axis.

5. DYNAFORM-PC prompts via Pop-Up window as Figure 5.9.
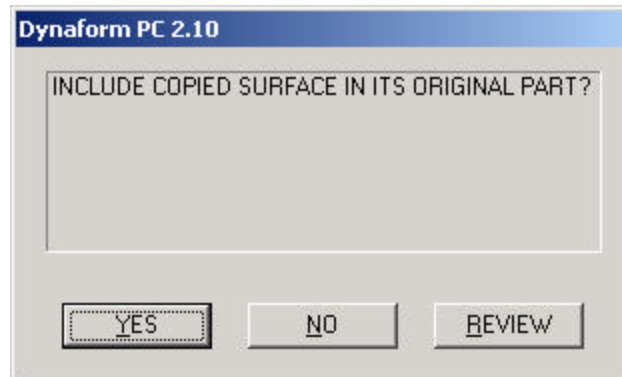
Figure 5.8 DYNAFORM-PC Question Window

?	YES	Copied surface will be in its original part.
	NO	Copied surface will be in the current part.
	REVIEW	Allows the user to review the current model.

5.15   TRANSFORM SURFACE

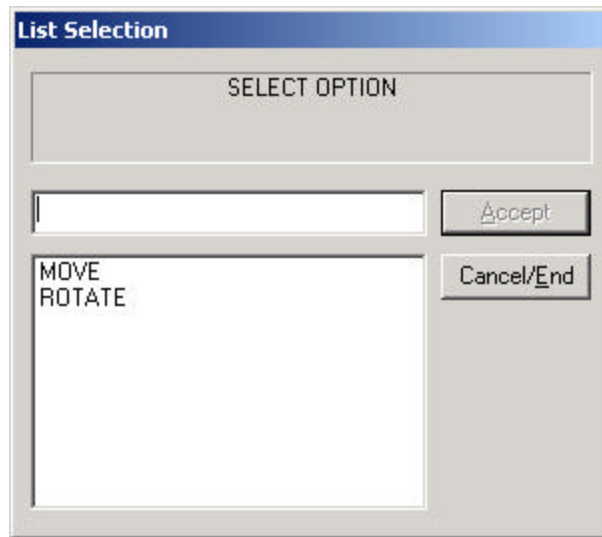Translates or rotates selected surface(s) to new locations.

1. DYNAFORM-PC prompts:

   DEFINE A LOCAL COORDINATE SYSTEM
   PICK POINT FOR THE ORIGIN

   ● DYNAFORM-PC prompts the user to create a local system. A thorough
     explanation of how to create a local coordinate system is covered in Chapter
     1 *General Discussion* section under "Local Coordinate System." The user
     also has the option to select the Global Coordinate System and the option to
     select the Last Coordinate System.

2. Once the desired coordinate system is selected, it will be displayed on the screen
   and the prompt will display via Pop-Up window:

   DO YOU ACCEPT THE DISPLAYED COORDINATE SYSTEM?

?	YES	Continue to Step 3.
	NO	Returns to Step 1.

3. DYNAFORM-PC prompts via Pop-Up window as Figure 5.9.

Figure 5.9 Methods for Transform Surface

? MOVE
DYNAFORM-PC prompts:

ENTER U, V AND W INCREMENTS

- i.e. 0,0,1000 will copy the selected surface(s) 1000 units along the W axis.

? ROTATE
DYNAFORM-PC prompts:

ENTER ANGLE INCREMENT

- Enter angle increment (right hand rule) about the W axis.

4. DYNAFORM-PC prompt:

SELECT SURFACES

Note: END SELECT will finish surface selection.

5. END SELECT will exit the function.

5.16   MIRROR SURFACE

Reflects a copy of desired surfaces across a local or global coordinate plane.

1. DYNAFORM-PC prompts:

SELECT SURFACE

2. Once the surfaces have been selected, END SELECT will continue the function.

3. DYNAFORM-PC prompts via Pop-Up window:

   DEFINE A LOCAL COORDINATE SYSTEM
   PICK POINT FOR THE ORIGIN

   ● DYNAFORM-PC prompts the user to create a local system. A thorough
   explanation of how to create a local coordinate system is covered in Chapter
   1 *General Discussion* section under "Local Coordinate System." The user
   also has the option to select the Global Coordinate System and the option to
   select the Last Coordinate System.

4. Once the desired coordinate system is selected, it will be displayed on the screen
   and the prompt will display via Pop-Up window:

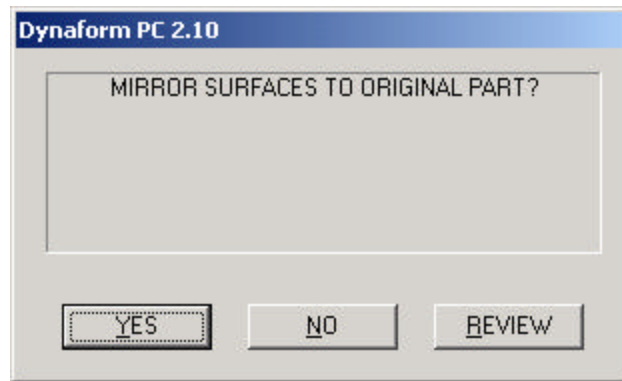   DO YOU ACCEPT THE DISPLAYED COORDINATE SYSTEM?

   ?   YES   Continues to Step 5.
       NO   Returns to Step 3.

5. DYNAFORM-PC prompts via Pop-Up window as Figure 5.10 to select a plane:
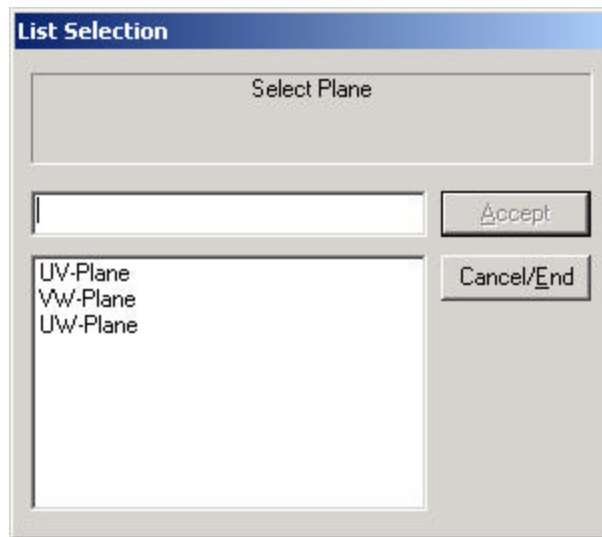
Figure 5.10 Select Plane

6. DYNAFORM-PC prompts the window as the Figure 11 whether or not to keep
   the surfaces in their original parts:

Figure 5.11 DYNAFORM-PC Question Window

?	YES	Places all selected surfaces in their source part.
	NO	Places all selected surfaces into the current part.
	REVIEW	Allows the user to review the model.

5.17   REMOVE HOLE

Removes holes from surfaces

1. DYNAFORM-PC prompts:

    SELECT A SURFACE

2. Once a desired surface is selected, DYNAFORM prompts.

    SELECT MODE TO REMOVE HOLE ON SURFACE
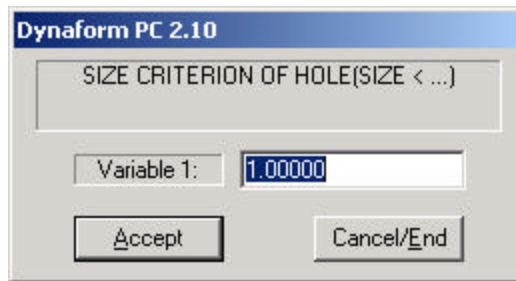
3. Options for Hole on Surface

    INNER HOLE
    There are three options to select inner hole.

    ● Select hole on surface by Cursor.

    ● Select hole by Size. DYNAFORM-PC will fill any hole less than
       the entered value.

Figure 5.12 Size Criterion

● All holes on the selected surface will be removed.

Remove hole on surface outer boundary by selecting two points on either
        edge of the gap.

5.18   LABEL SURFACE

Displays the surface ID number. Select LABEL SURFACE and the surface ID number is
displayed on the screen.

5.19   SHOW SURFACE

Highlights selected surfaces by cursor.

1. DYNAFORM-PC prompts:

    SELECT SURFACE

2. END SELECT will exit this function.

CHAPTER 6
ELEMENTS

The ELEMENTS menu contains functions used to create and modify elements. Elements are the basis for finite element modelling and are created from within this menu. This menu may be accessed through the PRE-PROCESS selection card located in the CONTROL WINDOW. The available functions are shown in Figure 6.1.

Figure 6.1 Element Option

A detailed description of each function is provided in the following pages.
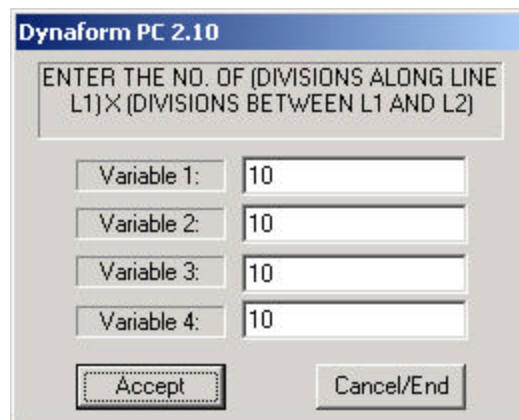
6.1   2-LINE MESH 🟫

Generates elements between two selected lines. A rectangular pattern of L1, L2, L3 by L4 quadrilateral elements will be generated. For a transition mesh, it is possible to enter L3 and L4 parameters for the mesh (i.e., each side has a different number of elements, such as 10, 5, 8, 7). The node sequence starts at the starting point of line 1 and follows the direction of line 1 with the right hand rule. The created elements will be included in the current part. The current part should be verified before proceeding with this function.

1. DYNAFORM-PC prompts:

    SELECT A LINE

    - LINE SEGMENT allows the user to combine a broken line while selecting lines for the 2-Line Mesh function as in Section 4.13 COMBINE LINES of LINE/POINTS. The user must select this option before selecting the desired lines for a 2-Line Mesh.

    - The user may check the line(s) prior to executing this function by using SHOW LINE in LINES/POINTS.

2. Once the desired line is selected, DYNAFORM-PC prompts via Pop-Up window as Figure 6.2.



Figure 6.2 2-Line Mesh Parameters

- L1 is any positive integer greater than one and represents the number of elements along the selected lines.

- L2 is any positive integer and represents the number of elements between the 2 lines.

- L3 (optional) is an integer value for the side opposite L1 where L1 is not greater than 2*L3 or L3 is not greater than 2*L1.

- L4 (optional) is an integer value for the side opposite L2 where L2 is not greater than 2*L4 or L4 is not greater than 2*L2.

- i.e.,10,10 or 10,10,11,14 = ACCEPTABLE

- i.e., 5,5,12,6 = NOT ACCEPTABLE

3. Once the desired element L1, L2 is entered, DYNAFORM-PC prompts via Pop-Up window:

   DO YOU ACCEPT THE GENERATED MESH?

   ? YES   Accepts mesh, places newly created elements in the current part and returns the user to Step 1.
     NO   Does not accept mesh and prompts the user to remesh the region.

4. END SELECT will exit the function.

6.2   3-LINE MESH

Generates elements in an area defined by 3 lines. The lines defining the area do not have to intersect. Elements will be included in the current part. The current part should be verified before proceeding with this function. Two mesh patterns are available for 3-Line Mesh. Due to the geometry of 3-Line Mesh, DYNAFORM-PC will generate some triangular elements and the user can choose the location of the triangular elements.

1. DYNAFORM-PC prompts:

   USING THE RIGHT HAND RULE, SELECT 3 LINES TO DEFINE THE MESH REGION

   - LINE SEGMENT allows the user to combine a broken line while selecting lines for the 3-Line Mesh function as in Section 4.13 COMBINE LINES of LINE/POINTS. The user must select this option before selecting the desired lines for a 3-Line Mesh.

   - The user may check the line(s) prior to executing this function by using SHOW LINE AND DIRECTION described in Section 4.19 LINES/POINTS.

2. Once the user selects the desired lines, DYNAFORM-PC prompts via Pop-Up window as Figure 6.3.
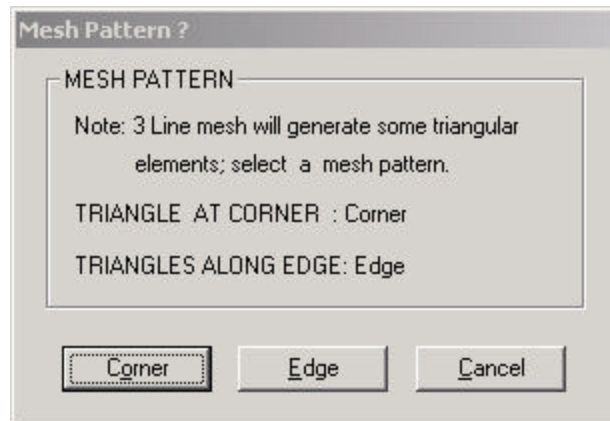
Figure 6.3 Mesh Pattern

    ? CORNER
- Triangles are concentrated in the vertex of the first two selected lines.

    DYNAFORM-PC prompts:

    ENTER THE NO. OF DIVISIONS ALONG EACH LINE: L1 AND L3

    ? EDGE
- Triangles are concentrated along the third selected line.

    DYNAFORM-PC prompts:

    ENTER NUMBER OF DIVISIONS ALONG EACH SIDE?

- DYNAFORM-PC accepts only one entry since each side has the same number of elements.

3. DYNAFORM-PC prompts via Pop-Up window:

    DO YOU ACCEPT THE GENERATED MESH?

    ? YES   Accepts mesh and places newly created elements in the current part.
       NO   DYNAFORM-PC prompts to remesh the region.

6.3   4-LINE MESH

Generates elements in an area defined by 4 selected lines. A rectangular pattern of L1, L2, L3 by L4 quadrilateral elements will be generated. For transition mesh, it is possible to enter L3 and L4 parameters for the mesh (i.e., each side has a different number of elements, such as 10, 5, 8, 7). The nodes in L2 direction are generated by linear interpretation between the opposing nodes on the two lines. A row of L2 quadrilateral elements is generated first from the starting end of line 1 to the starting end of line 2. The pattern will be repeated L1 times along line 1. The node sequence starts at the starting point of line 1 and follows the direction

of line 1 with the right hand rule. The created elements will be included in the current part. The current part should be verified before proceeding with this function.

1. DYNAFORM-PC prompts:

    USING THE RIGHT HAND RULE: SELECT 4 LINES TO DEFINE THE MESH REGION

    *   LINE SEGMENT allows the user to combine a broken line while selecting lines for the 4-Line Mesh function as COMBINE LINE of LINES/POINTS. The user must select this option before selecting the desired lines for a 4-Line Mesh.

    *   The user may check the line(s) prior to executing this function by using SHOW LINE in LINE/POINTS.

2. Once the user selects the desired lines, DYNAFORM-PC prompts via Pop-Up window as Figure 6.4.

Figure 6.4 4-Line Mesh Parameters

*   L1 is any positive integer, greater than one for the number of elements along the first selected lines.

*   L2 is any positive integer, for the number of elements along the second selected line.

*   L3 (optional) is an integer value for the side opposite L1 where L1 is not greater than 2*L3 or L3 is not greater than 2*L1.

*   L4 (optional) is an integer value for the side opposite L2 where L2 is not greater than 2*L4 or L4 is not greater than 2*L2.

*   i.e.,10,10 or 10,10,11,14 = ACCEPTABLE

*   i.e., 5,5,12,6 = NOT ACCEPTABLE

3. Once the desired element L1, L2, L3, L4 is entered, DYNAFORM-PC prompts via Pop-Up window:

   DO YOU ACCEPT THE GENERATED MESH?

   ?   YES   Places the elements in the current part.
       NO   Prompts to remesh the region.

4. END SELECT will exit the function.
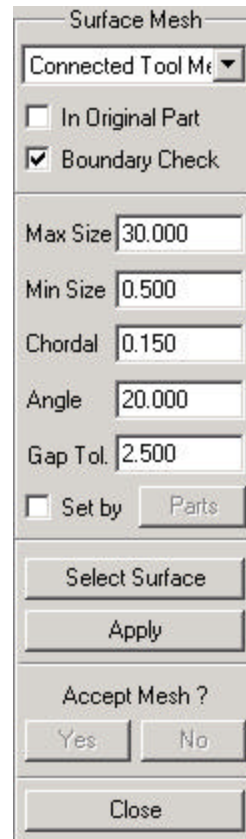

6.4   SURFACE MESH

Automatically meshes surface data. Currently DYNAFORM-PC auto meshes surfaces using 2D elements (plate elements). The user has the option to auto mesh either by selecting one surface, multiple surfaces or all displayed surfaces.

1. DYNAFORM-PC prompts:

   SELECT SURFACES

   • Once the surfaces are selected, END SELECT will continue the function.

2. DYNAFORM-PC displays the surface mesh criteria window, Figure 6.5.

Figure 6.5 Surface Mesh

3. Select a type of tool (connected or disconnected) or blank (fixed or variable), please see Figure 6.6.

Figure 6.6 Surface Mesh Type

4. Toggle on/off MESH IN ORIGINAL PART. This function includes the created elements in original part or in current part.

5. Enter the data for maximum size, minimum size, chordal deviation and gap tolerance in the field.

6. Click SELECT SURFACE to select additional surfaces, DYNAFORM-PC prompts:

    SELECT SURFACE

- Once the surfaces are selected, END SELECT will continue to the next step.

7. Select APPLY.

TOOL MESH
The tool mesh function will create the mesh on the selected surfaces based on the control parameters in Figure 6.5.

|	
MAX. SIZE	Maximum element size allowed in the mesh.
MIN.SIZE	Minimum element size allowed in the mesh.
CHORDAL	Largest chordal deviation allowed between the element edge and the surface in actual length unit.
ANGLE	Largest feature line angle allowed which is the angle of the planes between two adjacent plate elements.
Gap Tol.	The gap will be avoided if the distance between two surface boundaries is less than the Gap Tol..

The Connected Tool Mesh option will generate the mesh with the coincident nodes merged automatically along the boundaries of the adjacent surfaces. While the Unconnected Tool Mesh does not guarantee the conformity between adjacent surfaces, the speed of the mesh is faster. However, it will take a longer solution time in LS-DYNA for analysis.

- If the number of selected surfaces is greater than 2, the status window will be displayed. Please see Figure 6.7.
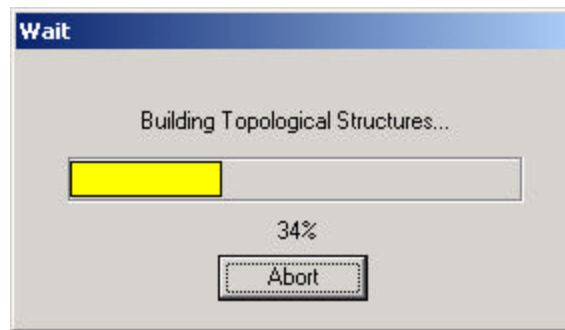

Figure 6.7 Building Topological Structures

After building topological structures, the Boundary Check window will be displayed allowing the user to change the Gap Tol.. Please see as Figure 6.8.
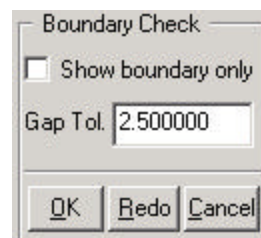

Figure 6.8 Boundary Check

SHOW BOUNDARY ONLY If check is on, only the boundary is displayed in the display window, or surfaces and boundaries will be displayed.

OK   Accepts the default Gap Tolerance and continues to the next step.

REDO   Change the Gap Tolerance; select REDO to build the topological structures again.

CANCEL   Cancels the surface mesh.

- If the number of selected surfaces is greater than 2, the status window will be displayed and DYNAFORM-PC will mesh the surface boundaries. Please see Figure 6.8.
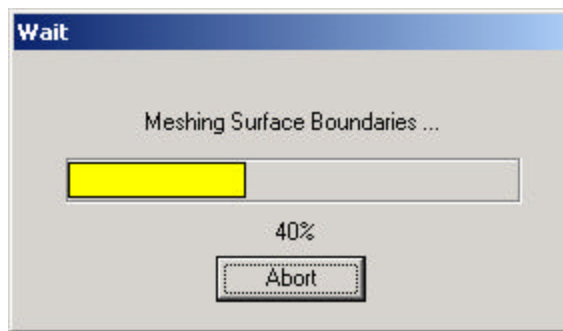
Figure 6.8 Meshing Surface Boundaries

After meshing surface boundaries, DYNAFORM-PC will create the mesh on the selected surface. Please see Figure 6.9.
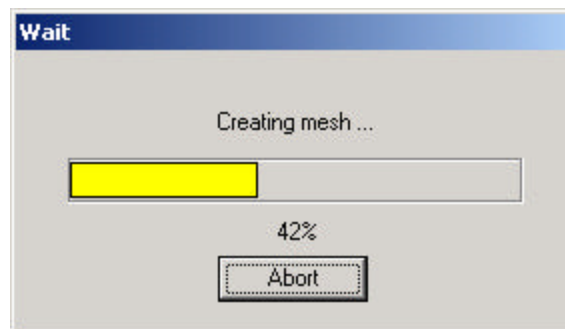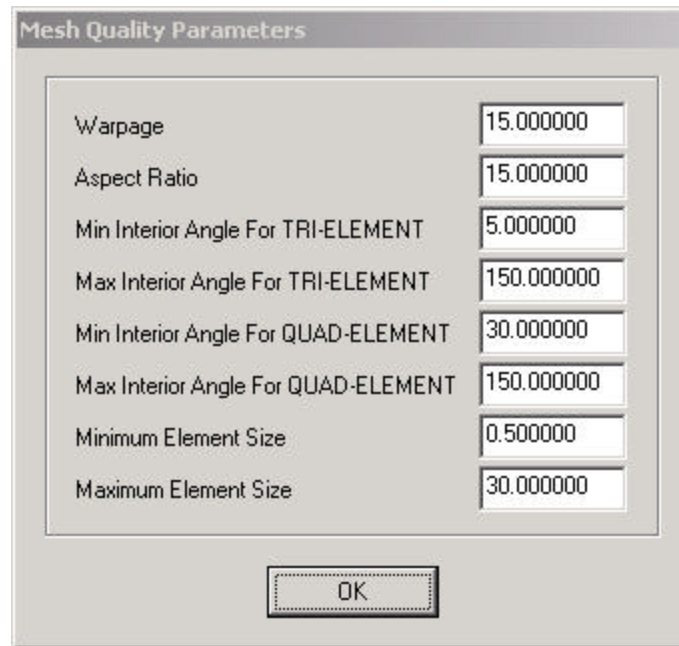

Figure 6.9 Creating Mesh

- Once the mesh operation is finished, select YES to accept or NO to reject the created mesh. And select CLOSE to exit the function.

FIXED BLANK MESH/VARIABLE BLANK MESH
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 only parameter required to control the Fixed Blank Mesh pattern is the element size. The Variable Blank Mesh requires

three parameters: maximum size, minimum size and chordal ratio to control the mesh pattern.

A parameter definition dialog window will be displayed for the user to define the quality criteria for the mesh. Please see the figure 6.10.
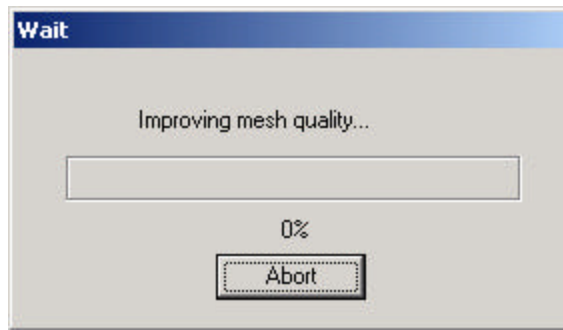

Figure 6.10 Mesh Quality Parameters

- The user can modify the mesh quality parameter or accept the default values.

- After selecting OK, the status windows, Figure 6.11 and Figure 6.12 appear successively if the number of blank surfaces is greater than 2.
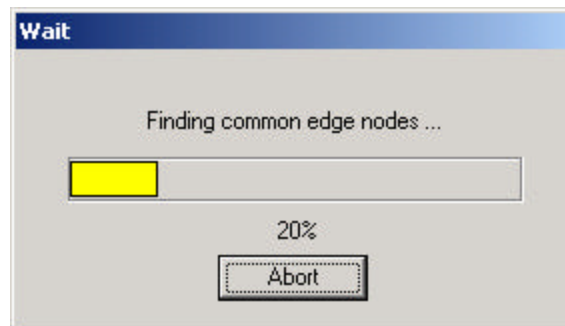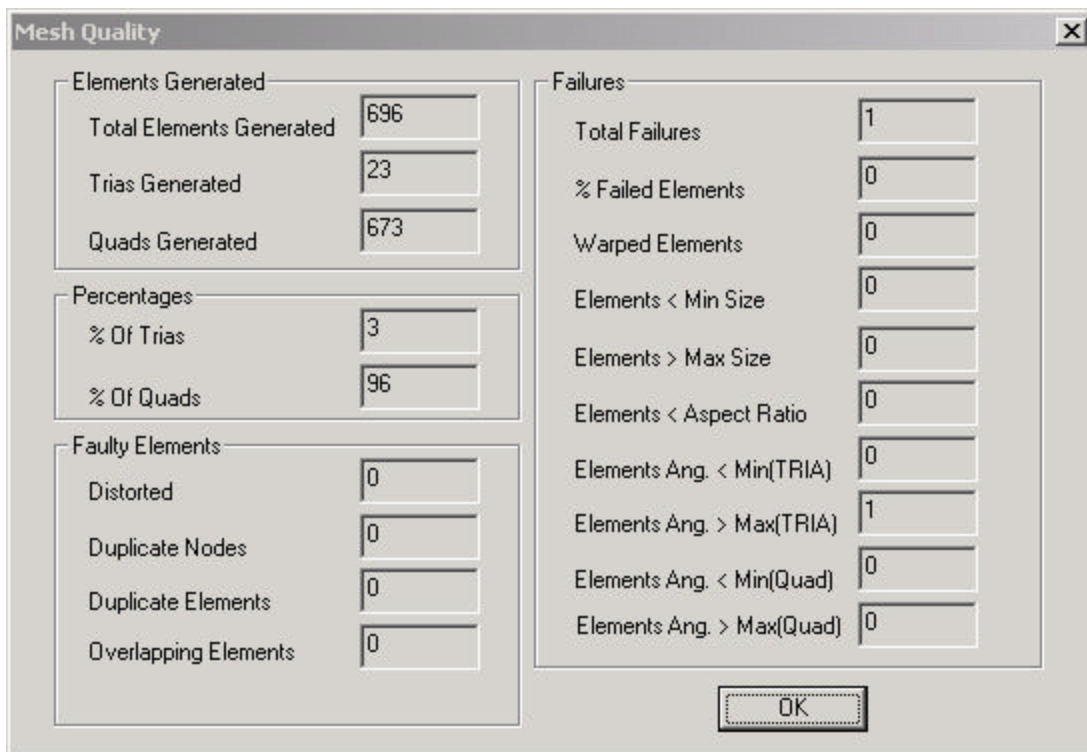

Figure 6.11 Finding Common Edge Nodes

Figure 6.12 Improving Mesh Quality

- After meshing, a mesh quality window is displayed, Figure 6.11.

Figure 6.13 Mesh Quality

- Select OK to close the dialog window.

- Once the mesh operation is finished, select YES to accept or NO to reject the created mesh, and select CLOSE to exit the function.

6.5   LINE MESH

This function allows the user to generate one-dimensional beam elements. The created elements will be included in the current part. The current part should be verified before proceeding with this function.

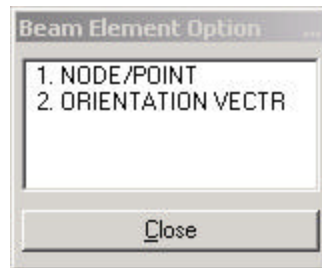1. DYNAFORM-PC prompts for the number of elements:
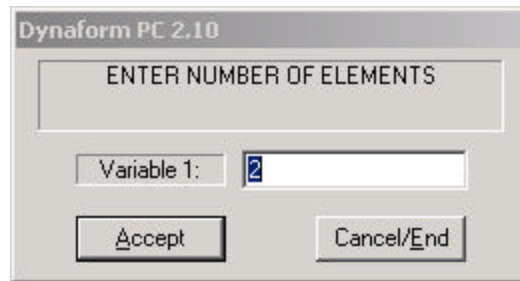
Figure 6.14

2. Once number of elements has been input and the lines selected, the Beam Element Option window will be displayed as in Figure 6.15.
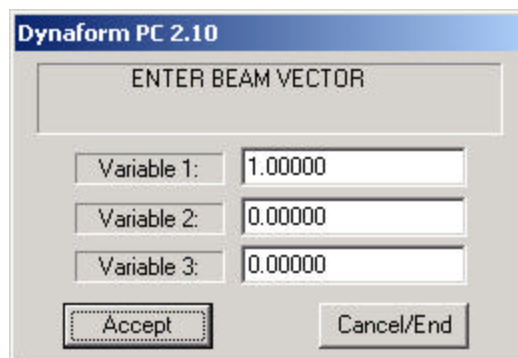
    DYNAFORM-PC prompts,

    PICK NODE/PT OR VECTOR FOR BEAM ORIENTATION

Figure 6.15

3. If the user selects NODE/POINT, the user can select a node or point to define the beam element orientation. If ORIENTATION VECTOR is selected, a window as in Figure 6.16 will be displayed, allowing the user to input the vector.

Figure 6.16 Enter Beam Vector
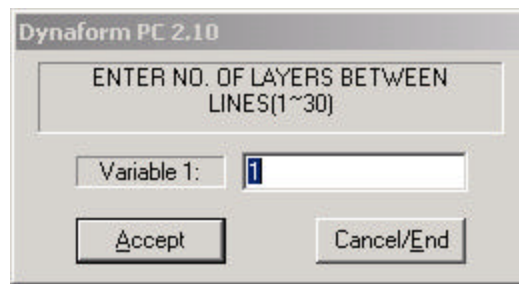
4. DYNAFORM-PC prompts:

ACCEPT MESH?

?   YES   Accepts mesh and prompts for more lines.
    NO   User can decide whether to re-mesh or cancel.
    REVIEW   User can review the model, then decide whether accept the mesh
               or not.

6.7   2-LINE POINT MESH

This function creates shell elements by dividing the area between two lines into layers (1-30). Once two desired lines are selected, the window as Figure 6.17 will be displayed allowing the user to input the number of layers.

Figure 6.17 Enter Numbers of Layers between Lines

DYNAFORM-PC prompts:

DO YOU ACCEPT THE GENERATED MESH?
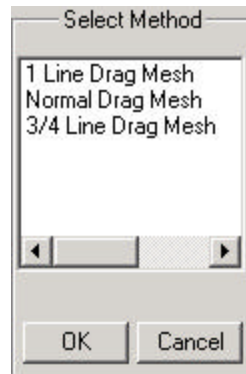
?   YES   Accepts mesh and prompts for more lines.
    NO   User can decide whether to re-mesh or cancel.
    REVIEW User can review the model, then decide whether accept the mesh or
               not.

6.7   DRAG MESH

Extrudes 1D elements into 2D elements (beam to plate) or 2D elements into 3D elements (plate to solid). Please see the Figure 6.18.

Figure 6.18 Drag Mesh Options

6.7.1   1-LINE DRAG MESH

Extrudes 1D elements into 2D elements (beam to plate) or 2D elements into 3D elements (plate to solid) along a selected line.

    1. DYNAFORM-PC prompts:

       SELECT ELEMENTS

- END SELECT completes this step.

    2. DYNAFORM-PC prompts:

       SELECT CONTROL LINE FOR DRAG MESH

- The number of points on the selected dragline dictates the number of elements created.

- Line direction must point in the direction of the intended mesh.

    3. Upon line selection, the mesh is created. DYNAFORM-PC prompts via Pop-Up window:

       DO YOU ACCEPT THE GENERATED MESH?

    ?   YES   Places the newly created elements in the current part.
         NO   Exits function.
         REVIEW User can review the model, then decide whether accept the mesh or not.

    Note: **Plate element normal should be consistent prior to the execution of 1-LINE DRAG MESH.**
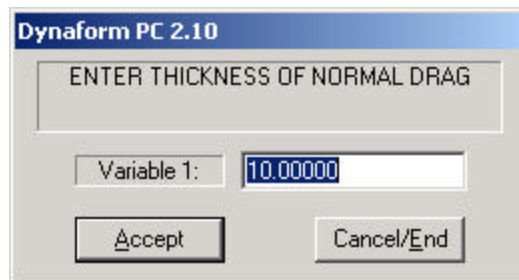
6.7.2   NORMAL DRAG MESH

Extrudes plate elements into solid elements in the normal direction of the selected plate elements.

     1. DYNAFORM-PC prompts:

        SELECT PLATE ELEMENT(S) FOR PATTERN
        SELECT ELEMENTS

       •   END SELECT completes this step.
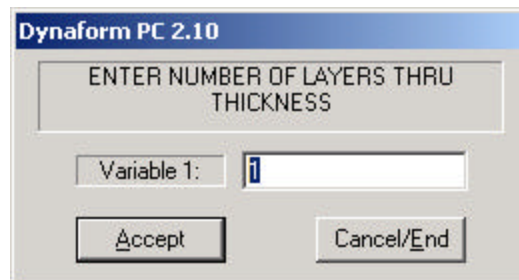
     2. DYNAFORM-PC prompts via Pop-Up window as in Figure 6.19:

Figure 6.19 Enter Thickness of Normal Drag

       •   Enter the total thickness of the drag operation.

       DYNAFORM-PC prompts via Pop-Up window as in Figure 6.20:

Figure 6.20 Enter Number of Layers

       •   Enter the number of layers through thickness.

       DO YOU ACCEPT THE GENERATED MESH?

      ? YES   Places newly created elements into the current part.
          NO   Exits function.
          REVIEW   User is allowed to rotate the display window first, then decide
                       whether accept the mesh or no

      Note: **Plate element normal should be consistent prior to the execution of
                 NORMAL DRAG MESH.**

6.7.3   3 - 4 LINE DRAG MESH

Extrudes beam elements into plate elements or plate elements into solid elements along three or four selected lines.

1. DYNAFORM-PC prompts:

   SELECT PLATE ELEMENT(S) FOR PATTERN
   SELECT ELEMENTS

   - END SELECT completes this step.

2. DYNAFORM-PC prompts:

   SELECT THE FIRST CONTROL LINES

   - The user must select the 3-4 control lines in a clockwise or counter-clockwise order to form a solid volume.

   - The user must use END SELECT after selecting the last line for a 3 Line Drag Mesh.

3. Once the desired lines are selected, DYNAFORM-PC prompts via Pop-Up window as in Figure 6.21:
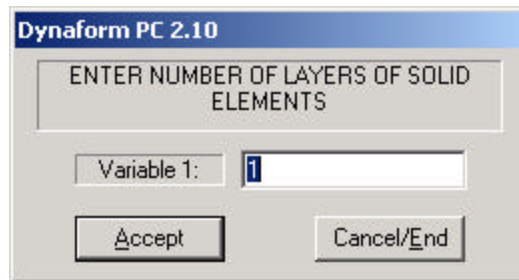
Figure 6.21 Enter Number of Layers of Solid Elements

4. After the number of layers is specified, DYNAFORM-PC creates the mesh and prompts via Pop-Up window:

   DO YOU ACCEPT THE GENERATED MESH?

   ?   YES   Accepts mesh and places it in current part.
        NO   Exits function.
          REVIEW   User is allowed to rotate the display window first, then decide whether accept the mesh or no

- Line direction is not important during the 3 - 4 Line Drag Mesh, but the lines should be of similar length to form a reasonable volume.

Note: **Plate element normal should be consistent prior to the execution of 3 - 4 LINE DRAG MESH.**

6.8   CREATE ELEMENTS

This function is used to create elements via the connection of selected nodes or points. The element types supported are those recognized by LS-DYNA. Please see Figure 6.22.

Figure 6.22 Select Element Type

User can select Change Element Type to change element type.

6.8.1   CREATE BEAM

Creates beam elements. The type of beam is controlled by the beam property defined for the beam.

1. DYNAFORM-PC prompts:

    PICK NODES/POINTS
    PICK POINT FOR ELEMENT

    - To create the elements the user may select a node, point, keyboard entry or any combination of the three.

2. Once the desired nodes/points have been selected, DYNAFORM-PC prompts:

    PICK NODE FOR BEAM ORIENTATION

3. Once the vector is determined, the element is created automatically in the current part.

4. END SELECT will exit the function.

6.8.2   NODAL RIGID BODY

Creates rigid link elements.

1. DYNAFORM-PC prompts for nodes/points for element:

    PICK NODES/POINTS
    AT INDEPENDENT NODE

    *   To create the elements the user may select a node, point, keyboard entry or any combination of the three.

2. Once the desired node or point has been selected, DYNAFORM-PC prompts:

    AT DEPENDENT NODE

    *   The user may select up to 6 dependent nodes.

3. Once the dependent node(s) is selected, END SELECT completes the process and returns to Step 1.

6.8.3   SPOT WELD

Creates spot weld in the current part.

1. DYNAFORM-PC prompts:

    PICK NODES/POINTS
    PICK POINT FOR ELEMENT

    *   The user may select a node, point, keyboard entry or any combination of the three to create an element.

    *   The SPOT WELD is created automatically in the current part after two nodes or points are selected

2. END SELECT will exit the function.

6.8.4   CREATE SHELL

Creates three and four node plate elements.

1. DYNAFORM-PC prompts:

    PICK NODES/POINTS
    PICK POINT FOR ELEMENT

    •   The user may select a node, point, keyboard entry or any combination of the
        three to create an element.
    •   A 3-node element requires 3 nodes or points to be selected followed by END
        SELECT.

2. END SELECT will exit the function.

6.8.5   CREATE SOLID

Creates four, six and eight node solid elements.

1. DYNAFORM-PC prompts:

    PICK NODES/POINTS
    PICK POINT FOR ELEMENT

    ●   To create the elements the user may select a node, point, keyboard entry or
        any combination of the three.

        -   Pick 4 node points and END SELECT to complete a 4-node tetrahedron
            element.

        -   Pick 6 nodes/points (pick the second set of 3 nodes/points in the same
            direction as the first set of 3 nodes/points) and then END SELECT to
            complete a 6-node wedge element.

        -   Pick 8 nodes/points (pick the second set of 4 nodes/points in the same
            direction as the first set of 4 nodes/points). An 8-node cube element will
            be created.

2. END SELECT will exit the function.

### 6.8.6   CREATE SPRING/DAMPER ELEMENT

Creates spring/damper elements.

1. The user should define the spring/damper element property first.

    ●   Select Property by Name [icon] is the default type. DYNAFORM-PC
        prompts:

        PICK AN ELEMENT TO IDENTIFY THE ELEMENT PROPERTY

The user can select an element with properties to define the spring/damper property.
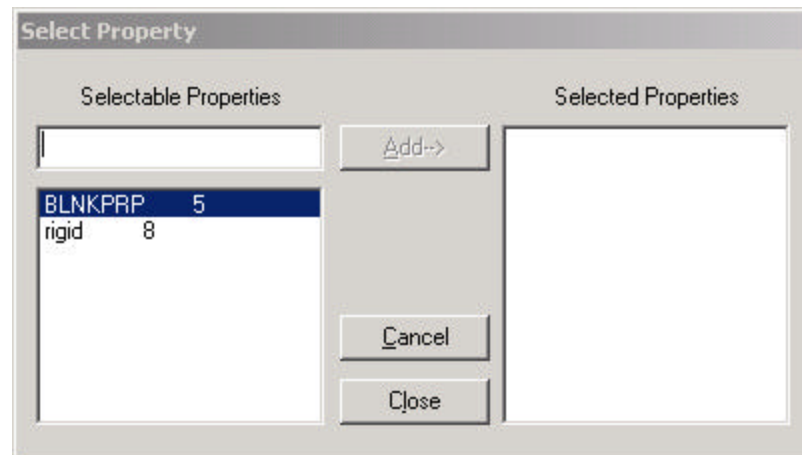
● The user can also select Create New Property ⏎ to define a new property. DYNAFORM-PC prompts via Pop-Up window as in Figure 6.23.
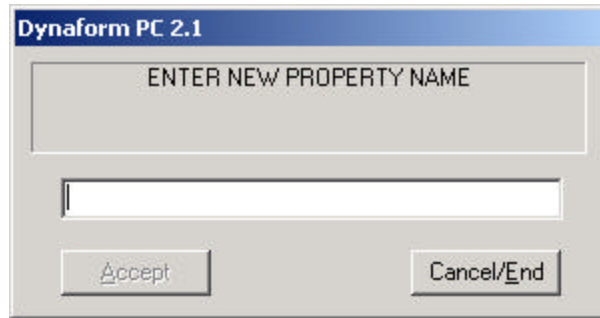
Figure 6.23 Enter New Property Name

Note: The definition of property can be found in the *LS-DYNA User's Manual.*

● The user can also select Select By Name **abc** to select an existing property. The Select Property Window is displayed as in Figure 6.24:

Figure 6.24 Select Property

2. Once the property is defined, DYNAFORM-PC prompts:

PICK NODES/POINTS
PICK POINT FOR ELEMENT

● To create the elements the user may select a node, point, keyboard entry or any combination of the three.
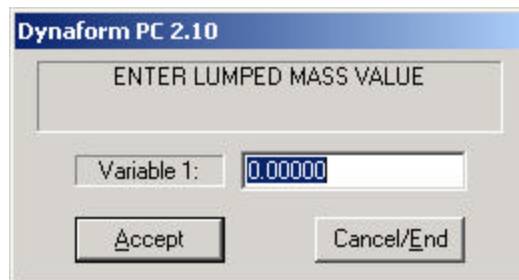
- Once two nodes/points are selected, the element is created automatically in the current part.

3. END SELECT will exit the function.

6.8.7 MASS

Creates concentrated mass elements.

1. DYNAFORM-PC prompts via Pop-Up window as in Figure 6.25.

Figure 6.25 Enter Lumper Mass Value

2. DYNAFORM-PC prompts:

PICK NODES/POINTS
PICK POINT FOR ELEMENT

- To create the elements the user may select a node, point or keyboard entry.

- The element will be created simultaneously upon selection of the desired node/point.

3. END SELECT will exit the function.

6.8.8 THICK SHELL

Creates thick shell elements.

1. DYNAFORM-PC prompts:

PICK NODES/POINTS
PICK POINT FOR ELEMENT

- To create the elements the user may select a node, point or keyboard entry.

- The elements will be created simultaneously upon selection of the desired node/points.

- • 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).

2. END SELECT will exit the function.

6.9   COARSE ELEMENTS

Coarsens up to four plate elements into one plate element.

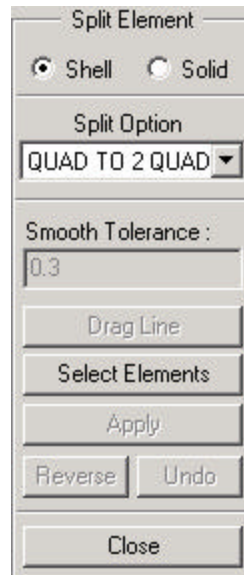1. DYNAFORM-PC prompts the user to select 2, 3, or 4 elements:

   SELECT ELEMENTS

2. END SELECT coarsens the selected elements and prompts for more. Select the function again to exit the function.
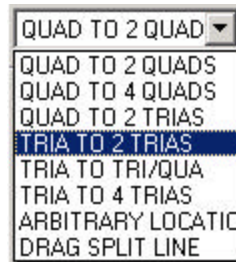
6.10   SPLIT ELEMENTS

This function divides SHELL elements into multiple elements.

1. DYNAFORM-PC prompts:

Figure 6.26 Split Element

2. Select SPLIT OPTION
   User can select from the following types:

Figure 6.27 Split Option

QUAD TO 2 QUADS
Splits a quadrilateral into two quadrilaterals by dividing the longest side and its opposite side. If select REVERSE, dividing two other sides.

QUAD TO 4 QUADS
Splits a quadrilateral into four quadrilaterals.

QUAD TO 2 TRIAS
Splits a quadrilateral into two triangular elements with the diagonal line in the corner of the greatest angle. If REVERSE is selected, the element is split from the other diagonal line.

TRIA TO 2 TRIAS
Splits a quadrilateral to two triangular elements with the midline of the greatest angle. If select REVERSE, splits with the midline of the smallest angle.

TRIA TO TRI/QUA
Splits a triangular element into a triangular element and a quadrilateral element with the line between the two midpoints of the two longest lines. The user can select REVERSE to change splitting position.

TRIA TO 4 TRIAS
Splits a quadrilateral to four triangular elements with three lines between three midpoints of three sides.

ARBITRARY LOCATION
Select an element first and select two nodes/points arbitrarily on the boundary of the element to split the element to two elements.

DRAG SPLIT LINE
Select DRAGLINE to drag a line to split elements that the drag line passes. The user can enter a number into the field to change smooth tolerance.

3. Click SELECT ELEMENTS, DYNAFORM-PC prompts:

SELECT ELEMENTS

*   END SELECT will continue to the next step.

4. Select APPLY.

    •   After the elements are split, the user can select UNDO to cancel the split or select REVERSE to change the split direction.

5. Select CLOSE to exit the function.

6.11 REVERSE NORMAL

Reverses the element orientation (normal) for all elements except MASS elements.
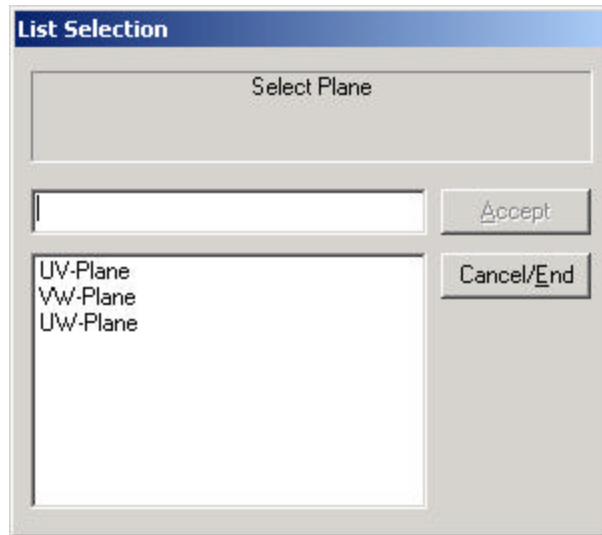
1. DYNAFORM-PC prompts:

    SELECT ELEMENTS

2. Once the desired elements have been selected, END SELECT reverses the element normal and exits the command.

6.12 MIRROR ELEMENTS

Reflects a copy of desired elements across a local or global coordinate plane.

1. DYNAFORM-PC prompts:

    SELECT ELEMENTS

        •   END SELECT will continue to the next step.

2. Once the desired elements have been selected, DYNAFORM-PC prompts:

    DEFINE A LOCAL COORDINATE SYSTEM
    PICK POINT FOR THE ORIGIN

        •   DYNAFORM-PC prompts the user to create a local system. A thorough explanation of how to create a local coordinate system is covered in Chapter 1 *General Discussion* section under "Local Coordinate System."

3. Once the desired coordinate system is acquired, it will be displayed on the screen and DYNAFORM-PC prompts via Pop-Up window:

    DO YOU ACCEPT THE DISPLAYED COORDINATE SYSTEM?

    ? YES   Continues to Step 4.
      NO   Return to Step 2.
        REVIEW   User is allowed to rotate the display window first, then decide whether accept the displayed coordinate system or not.

4. DYNAFORM-PC prompts via Pop-Up window as Figure 6.28.

Figure 6.28 Select Plane

5. DYNAFORM-PC prompts via Pop-Up window:

INCLUDE MIRRORED ELEMENTS IN ORIGINAL PART?

?   YES   Places the selected elements in their respective source parts.
    NO   Places all selected elements in the current part.
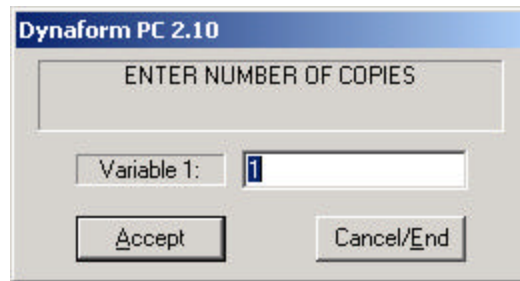   CANCEL Cancel Mirror operation.

6.13   COPY ELEMENTS

Generates a set of duplicate elements. The user may simultaneously translate, rotate or offset in the normal direction the copied elements.
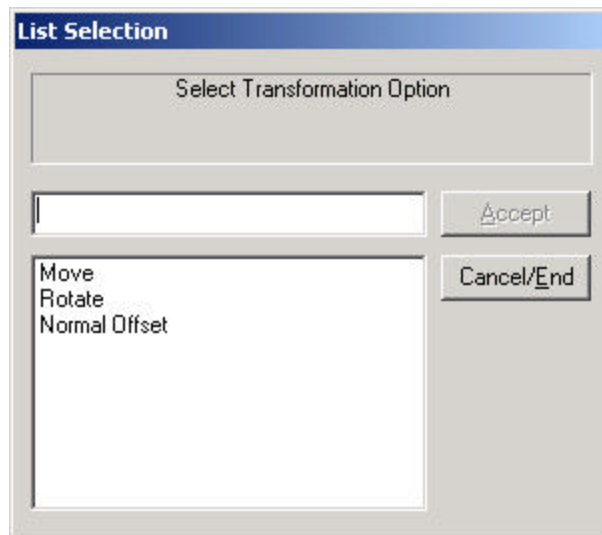
1. DYNAFORM-PC prompts:

SELECT ELEMENTS

•   END SELECT will continue to the next step.

2. DYNAFORM-PC prompts via Pop-Up window as Figure 6.29.

Figure 6.29 Enter Number of Copies

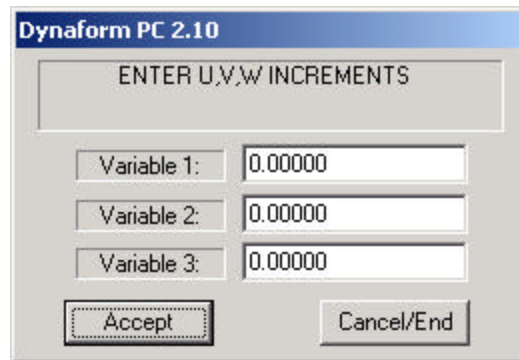3. DYNAFORM-PC prompts a window as Figure 6.30.

Figure 6.30 Select Transformation Option

4. Select a transformation option and DYNAFORM-PC prompts:

    DEFINE A LOCAL COORDINATE SYSTEM
    PICK POINT FOR THE ORIGIN

    - DYNAFORM-PC prompts the user to create a local system. A thorough explanation of how to create a local coordinate system is covered in Chapter 1 *General Discussion* section under "Local Coordinate System."

5. DYNAFORM-PC prompts for the magnitudes of transformation (respective to the selections made in Step 4):

    ?   MOVE
        DYNAFORM-PC prompts a window as Figure 6.31 allowed user to put the position of moved element (s).
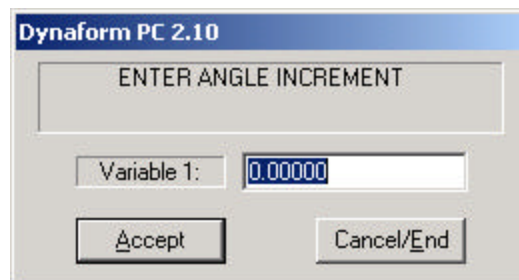
Figure 6.31 Enter UVW Increments

- i.e., 0,0,1000 will copy the selected element(s) 1000 units along the Z axis.

? ROTATE
    DYNAFORM-PC prompts a window as Figure 6.32 allowed user to put the position of rotated element (s).
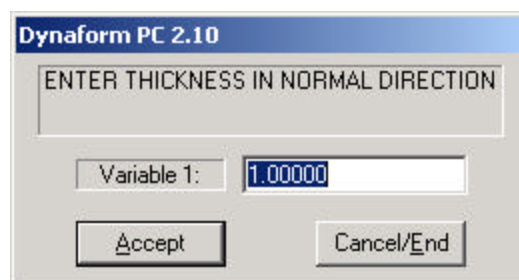

Figure 6.32 Enter Angle Increment

- Enter angle increment (right hand rule) about the W axis.
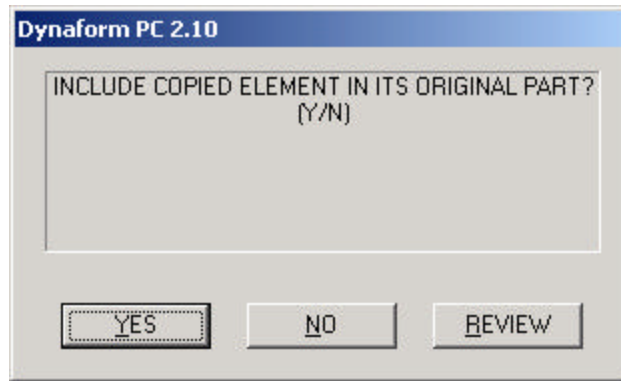
? NORMAL OFFSET
    DYNAFORM-PC prompts a window as Figure 6.33 allowed user to put the position of offset element (s).

Figure 6.33 Enter Thickness in Normal Direction

- Thickness is the distance between the plane of the elements and the direction of the element's normal.

6. DYNAFORM-PC prompts a DYANFORM Question window as Figure 6.34.

Figure 6.34 DYNAFORM-PC Question Window

? YES   Places all selected elements in their original part.

NO   Places all selected elements in the current part.

REVIEW   User is allowed to rotate the display window first, then decide whether accept the normal offset or not.

6.14   MODIFY ELEMENT

Deletes and recreates any displayed elements. The displayed elements to be modified need not be in the current part. DYNAFORM-PC will automatically identify the type of the selected element and prompt the user to recreate it:

1. DYNAFORM-PC prompts:

   PICK ELEMENT TO MODIFY

2. Once the desired element is selected, DYNAFORM-PC prompts a Pop-Up window that corresponds to the type of element selected. Enter the new data and DYNAFORM-PC prompts:

   SELECT NODE

   Details are discussed earlier in this chapter in Section 6.8 CREATE ELEMENTS

   Note: **In the case of one-dimensional elements (beams), DYNAFORM-PC displays the selected beam's property ID and allows the user to modify the beam's property ID in addition to the beam's connectivity.**

6.15   DELETE ELEMENTS ✂

Deletes elements from the model.

1. DYNAFORM-PC prompts:

    SELECT ELEMENTS

2. Once the desired elements are selected, END SELECT deletes the chosen elements and exits the command.

6.16   CHANGE ELEMENT NUMBER 1→2

Changes an existing element number. Parts do not need to be on or current.

1. DYNAFORM-PC prompts the user to select elements:

    SELECT ELEMENT FOR NEW ELEMENT NUMBER

2. Once the user selects a desired element, DYNAFORM-PC prompts via Pop-Up window as Figure 6.35.
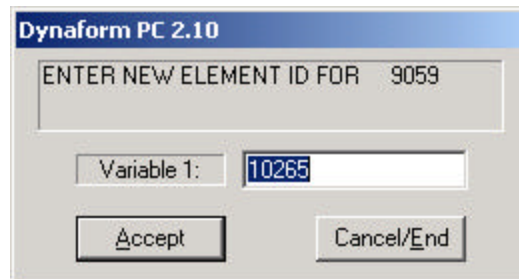
**Figure 6.35 Enter New Element ID**

-   Where xxxx is the original element number.

3. If the user enters a number that already exists, DYNAFORM-PC prompts:

    ELEMENT ID yyyy ALREADY EXISTS, REQUEST DENIED

6.17   RENUMBER ELEMENTS 1→11 2→12

Renumbers all elements in the existing database. The user may renumber elements by sequence or parts.

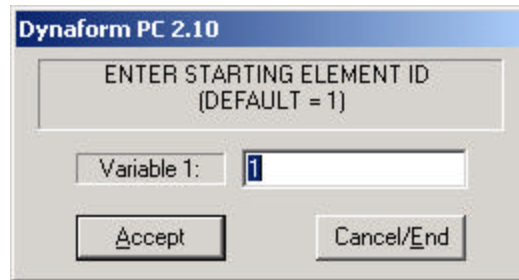1. DYNAFORM-PC prompts via Pop-Up window as Figure 6.36.
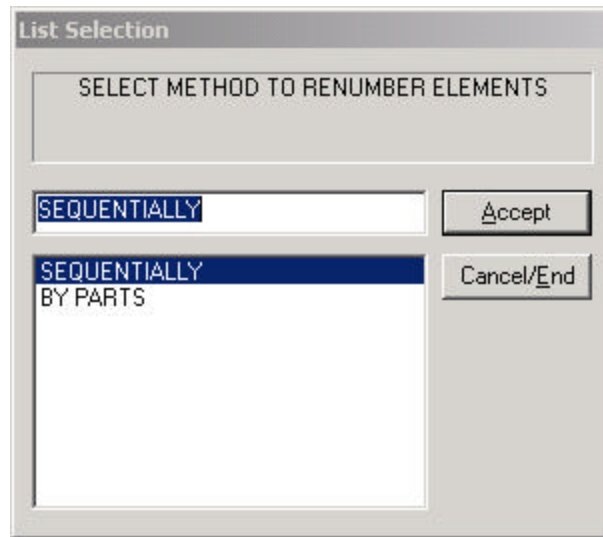
Figure 6.36 Select Method to Renumber Elements

SEQUENTIALLY
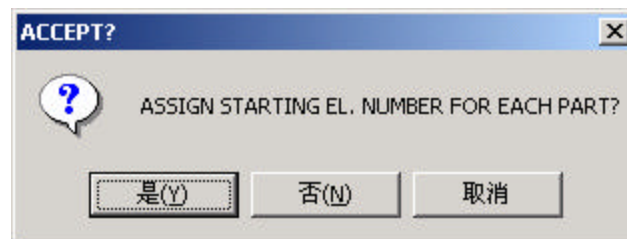DYNAFORM-PC prompts via Pop-Up window as Figure 6.37.

Figure 6.37 Renumber Element Sequentially

- Select ACCEPT to complete the operation.

BY PARTS
DYNAFORM-PC prompts via Pop-Up window as Figure 6.38.

Figure 6.38 Renumber Element Sequentially By Parts

? YES   Continue to Step 2.

NO   Continue to Step 4.

2. DYNAFORM-PC prompts via Pop-Up window:

    ENTER STARTING ELEMENT ID FOR PART XXXX

3. Once a number has been entered, DYNAFORM-PC automatically renumbers the element in the specified part. Step 3 is repeated for each additional part. If the starting number has already been used, DYNAFORM-PC will prompt via Pop-Up window:

    EL. ID HAS BEEN USED, NEXT ELEM NO.: XXXX

    Once a number is entered, DYNAFORM-PC will prompt:

    ELEM ID xx TO xx ASSIGNED, NEXT ELEM NO.: yyyy

    *   If "blank" or "zero" is entered, yyyy will be used as the starting element number for the next part.

4. DYNAFORM-PC prompts via Pop-Up window:

    ENTER STARTING ELEMENT ID NUMBER

    *   Any positive integer may be entered.
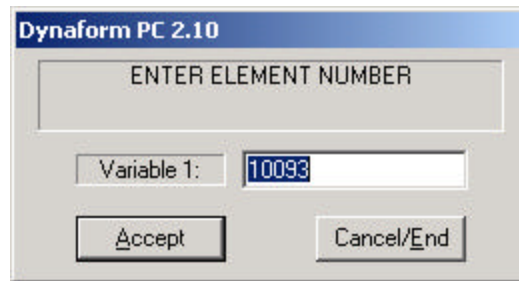
5. DYNAFORM-PC prompts via Pop-Up window:

    ENTER ELEMENT INCREMENT BETWEEN PARTS (1000)

    *   Any positive integer may be entered.

    *   Once the integer is entered, DYNAFORM-PC automatically renumbers the elements.

6.18   FIND ELEMENT

Finds and identifies an element by entering its element number.

1. DYNAFORM-PC prompts a window as Figure 6.39.

Figure 6.39 Enter Element Number

- Once the element number has been entered, it will be highlighted with its element and node numbers labelled. The node numbers will also be displayed in the prompt area of the screen.

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

   ELEMENT X IN TURNED OFF PART: YY

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

   ELEMENT X NOT FOUND IN DATABASE

6.19   IDENTIFY ELEMENTS

Identifies an element and its nodes by cursor selection.

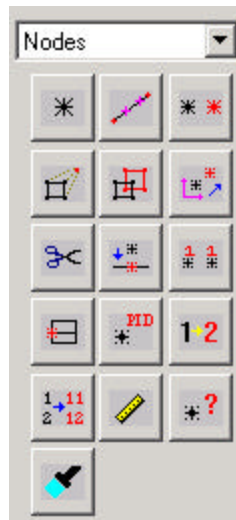1. DYNAFORM-PC prompts:

   SELECT ELEMENT BY CURSOR

   - DYNAFORM-PC identifies the element type, number, part to which it belongs and respective nodes. The selected element will be highlighted and the element and node numbers will be displayed on the screen and in the PROMPT WINDOW.

   - It will also identify:

     RIGID LINK   Degree of freedom for the independent node.
     SPRING ELEMENT   Spring rate and direction.
     RIGID BAR ELEMENT   Degrees of freedom.
     MASS ELEMENT   Mass at node number.
     BEAM ELEMENT   Bar ID and vector.

CHAPTER 7
NODES

In DYNAFORM-PC, 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 (free) nodes. A referenced node is a node that used by elements and is represented by a dot. A free node is a node that is not used by any elements and is represented by an asterisk. This menu may be accessed through the PRE-PROCESS selection card located in the CONTROL WINDOW. The functions in this menu are displayed as Figure 7.1.

Figure 7.1 Node Option

A detailed description of each function is provided in the following pages.

7.1   CREATE NODE   ✳

Generates unreferenced nodes at global locations or at any selected point on a line.

1. DYNAFORM-PC prompts:

    SELECT POINT FOR NODE LOCATION

2. Once the point is selected, a node is generated automatically in the current part.

3. END SELECT will exit the function.

7.2   CREATE NODES BETWEEN POINTS/NODES   ⬜

Generates nodes equally spaced between two existing nodes or points. The new nodes are displayed as unreferenced (free) nodes.

1. DYNAFORM-PC prompts:

    SELECT POINT 1 (NODE 1)

2. Once a point/node is selected, DYNAFORM-PC prompts:

    SELECT POINT 2 (NODE 2)

3. After the second point/node is selected, DYNAFORM-PC prompts:

    ENTER NUMBER OF NODES TO CREATE

4. DYNAFORM-PC prompts via Pop-Up window as Figure 7.2.
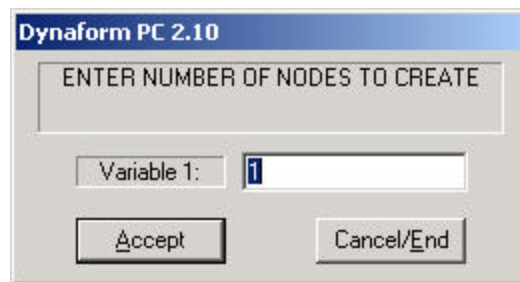
Figure 7.2 Enter Number of Nodes to Create

- Any integer, one or greater, is valid.

5. Input a valid value and select ACCEPT. A set of nodes is generated automatically in the current part.

6. END SELECT will exit the function.

7.3   COPY NODES

Generates a set of duplicate nodes (referenced or unreferenced) at a user-specified distance or rotation from the original nodes.

1. DYNAFORM-PC prompts via Pop-Up window:
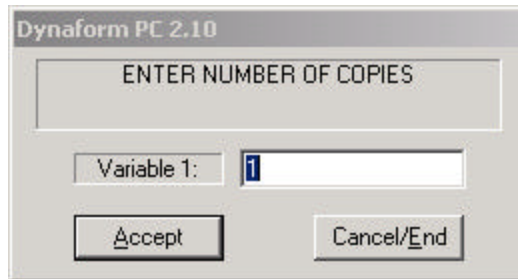
Figure 7.3 Enter Number of Copies

- Input a positive integer.

2. DYNAFORM-PC prompts:

   DEFINE A LOCAL COORDINATE SYSTEM
   PICK POINT FOR THE ORIGIN

   - DYNAFORM-PC prompts the user to create a local system. A thorough explanation of how to create a local coordinate system is covered in Chapter 1 *General Discussion* section under "Local Coordinate System."

3. Once the desired coordinate system is acquired, it will be displayed on the screen and the prompt will read via Pop-Up window:

   DO YOU ACCEPT THE DISPLAYED COORDINATE SYSTEM?

   ? YES   Continue to Step 4.
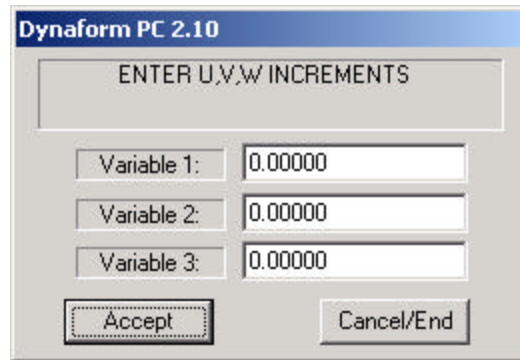   NO   Returns to Step 2.

4. DYNAFORM-PC prompts via Pop-Up window:

   SELECT TRANSFORMATION OPTION

   ?   MOVE
       ROTATE

?   MOVE
     DYNAFORM-PC prompts a window as Figure 7.4 allowed user to put the
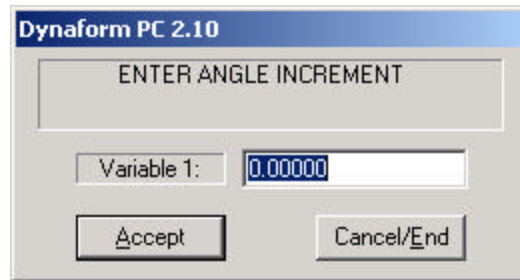   position of moved element (s).

Figure 7.4 Enter UVW Increments

- i.e. 0,0,1000. This will copy the selected node(s) 1000 units along the W axis.

?   ROTATE
     DYNAFORM-PC prompts a window as Figure 7.5 allowed user to put the
      position of rotated element (s).

Figure 7.5 Enter Angle Increment

- Enter an angle increment (right hand rule) about the local W or global Z.

5. DYNAFORM-PC prompts:

    SELECT NODES

- Once the desired nodes are selected, END SELECT will complete and exit
  the function.

7.4   MOVE NODES

Moves referenced or unreferenced nodes to any location on the display screen.

1. DYNAFORM-PC prompts:

    SELECT NODE TO MOVE

2. After selecting a node, DYNAFORM-PC prompts for a new location for the nodes:

    SELECT DESTINATION NODE

    •   END SELECT will complete and exit the function.

3. If multiple or coincident nodes are found near the cursor, DYNAFORM-PC prompts via Pop-Up window:

    MULTIPLE NODES FOUND
    SELECT ELEMENT FOR NODE

    •   If an element centre is selected by cursor and it does not connect to the target node, the following message will echo to the command line area of the DYNAFORM-PC screen:

        SELECTED ELEMENT IS NOT CONNECTED TO DUPL. NODE
        SELECT ELEMENT FOR NODE

    •   The user is now returned to Step 1.

7.5   SCALE NODES ⊞

Applies a scale factor to any or all X, Y, and Z components for some or all displayed nodes in a global or local direction. The scale factor may be positive or negative (change the sign on any or all of the X, Y, and Z components). The parts will be magnified if the scale factor is greater than 1 and shrunk if the scale factor is less than 1. The user may reflect, enlarge or shrink a part without leaving a copy of the original elements behind.

1. DYNAFORM-PC prompts:

    PICK NODES TO SCALE
    SELECT NODES

    •   END SELECT will continue to the next step.

2. DYNAFORM-PC prompts:

   DEFINE A LOCAL COORDINATE SYSTEM
    PICK POINT FOR THE ORIGIN

- DYNAFORM-PC prompts the user to create a local system. A thorough explanation of how to create a local coordinate system is covered in Chapter 1 *General Discussion* section under "Local Coordinate System."

3. Once the desired coordinate system is acquired, it will be displayed on the screen and the prompt will read via Pop-Up window:

    DO YOU ACCEPT THE DISPLAYED COORDINATE SYSTEM?

    ? YES   Continue to Step 4.
      NO   Exits function.

4. DYNAFORM-PC prompts via Pop-Up window as Figure 7.6.
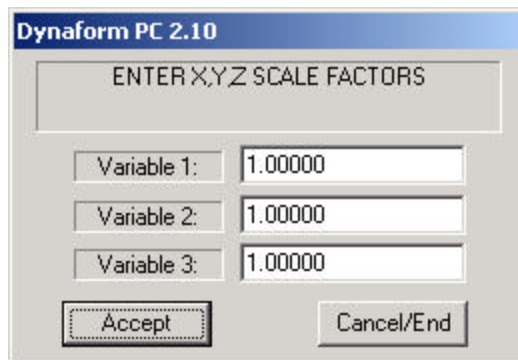
Figure 7.6 Enter XYZ Scale Factors

- All real numbers are valid. ZEROS or BLANKS will default to (+1).

    EXAMPLES:

- 1, -1, 1 will reflect selected nodes across the X-Z plane (Y-direction). All Y components will be multiplied by (-1).

- -2, 1, 0.5 will magnify all the X components by two times and reflect the nodes across the Y-Z plane (X-direction), not affecting the Y-components and will shrink the Z-components to half size and leave them on the original side of the X-Y plane.

5. Select ACCEPT to finish the operation.

7.6   TRANSFORM NODES

Translates or rotates the selected nodes to a new location. Transform Nodes Methods is displayed as Figure 7.7.
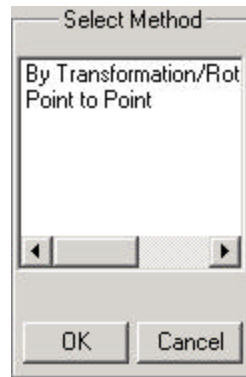
Figure 7.7 Transform Nodes Methods

7.6.1   BY TRANSFORMATION/ROTATION

Translates or rotates the selected nodes to a new location.
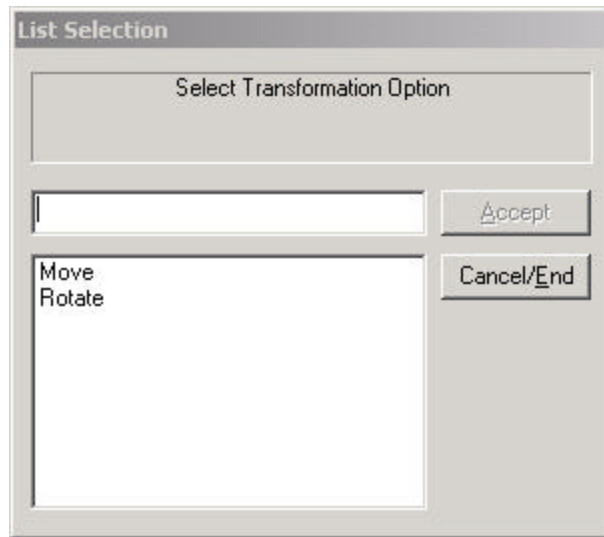
1. DYNAFORM-PC prompts:

    DEFINE A LOCAL COORDINATE SYSTEM
    PICK POINT FOR THE ORIGIN

    •   DYNAFORM-PC prompts the user to create a local system. A thorough
        explanation of how to create a local coordinate system is covered in Chapter
        1 *General Discussion* section under "Local Coordinate System."

2. Once the desired coordinate system is acquired, it will be displayed on the screen
    and the prompt will read via Pop-Up window:

    DO YOU ACCEPT THE DISPLAYED COORDINATE SYSTEM?

    ? YES   Continue to Step 3.
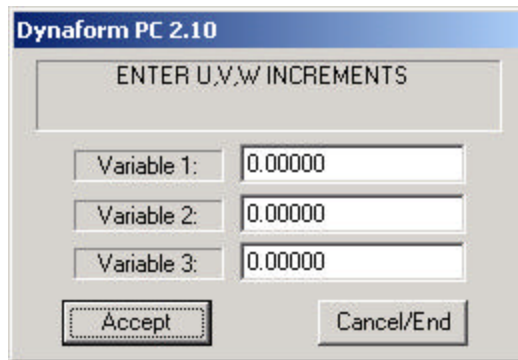   NO   Returns to Step 1.

3. DYNAFORM-PC prompts the following via Pop-Up window as Figure 7.8.

Figure 7.8 Transform Nodes Options

? MOVE

DYNAFORM-PC prompts a window as Figure 7.9 allowed user to put the
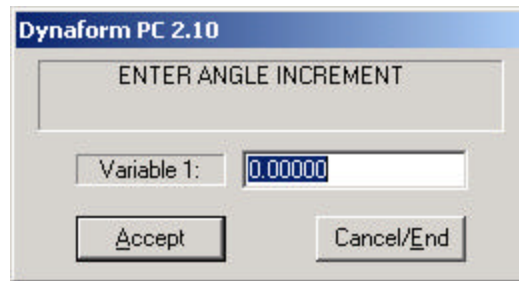position of moved element (s).

Figure 7.9 Enter UVW Increments

- i.e., 0,0,1000. This will copy the selected node(s) 1000 units along the W
  axis.

? ROTATE

DYNAFORM-PC prompts a window as Figure 7.10 allowed user to put the
position of rotated element (s).

Figure 7.10 Enter Angle Increment

- Enter the angle increment (right hand rule) about the local W or global Z.

3. DYNAFORM-PC prompts:

   SELECT A NODE TO TRANSFORM

4. Once the desired node is selected, DYNAFORM-PC automatically transforms the node. END SELECT will complete the function and return to Step 1.

5. CANCEL/END in Figure 7.8 will exit the function.

7.6.2 BY POINT TO POINT

Translates the selected nodes to new location by reference point.

1. DYNAFORM-PC prompts:

   DEFINE A DIRECTION VECTOR
   SELECT A NODE FOR THE ORIGIN

2. Once the desired point is selected, the direction vector displays and DYNAFORM-PC prompts:

   SELECT A NODE FOR THE TARGET LOCATION

3. Once the second point is selected, DYNAFORM-PC prompts:

   SELECT A NODE TO TRANSFORM

4. END SELECT will complete the function and return to Step 1.

7.7 DELETE NODES

Deletes all unreferenced nodes or the selected unreferenced nodes. Please see Figure 7.11.
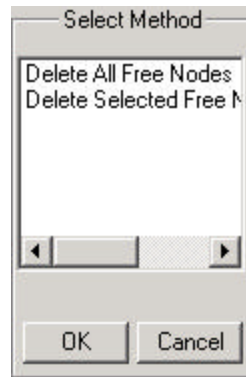
Figure 7.11 Delete Nodes Methods

7.7.1   DELETE ALL FREE NODES

Deletes all unreferenced nodes shown by asterisks (*) in the database.

    1. DYNAFORM-PC will prompt one of the following messages:

       xx UNREFERENCED NODES DELETED
       or
       NO UNREFERENCED NODES FOUND

7.7.2   DELETE SELECTED FREE NODES

This function allows the user to delete selected unreferenced nodes shown by asterisks (*) in the database.

    1. DYNAFORM-PC prompts:

       SELECT NODES

       • END SELECT will continue to the next step.

    2. If any of the selected nodes are referenced nodes the following message will echo:

       CAN NOT DELETE NODE xxxx, IT IS NOT UNREF.
       NO UNREFERENCED NODES FOUND

7.8   PROJECT NODE(S)

Project node(s) onto the element mesh, local U-V plane and surface. Please see Figure 7.12 to get project methods.

Figure 7.12 Project Node (s) Method

7.8.1   ONTO ELEMENT MESH

Projects a node or group of nodes onto any element mesh. Nodes may be referenced or unreferenced.

1. DYNAFORM-PC prompts:

    SELECT PLATE ELEMENTS FOR TARGET SURFACE
    SELECT ELEMENTS

    •   END SELECT will continue to the next step.

2. DYNAFORM-PC prompts:

    DEFINE PROJECTION VECTOR
    DEFINE A LOCAL COORDINATE SYSTEM
    PICK POINT FOR THE ORIGIN

    •   DYNAFORM-PC prompts the user to create a local system. A thorough explanation of how to create a local coordinate system is covered in Chapter 1 *General Discussion* section under "Local Coordinate System."

    Note: The nodes are projected only along the local W-axis. The user must define the local coordinate system accordingly.

3. Once the coordinate system is defined DYNAFORM-PC prompts:

    SELECT NODE(S) TO PROJECT ONTO MESH

4. END SELECT will complete the command.

7.8.2   ONTO LOCAL U-V PLANE

Projects a node or group of nodes onto any U-V plane. Nodes may be referenced or unreferenced.

1. DYNAFORM-PC prompts:

    DEFINE PROJECTION (UV) PLANE
    DEFINE A LOCAL COORDINATE SYSTEM
    PICK POINT FOR THE ORIGIN

    •   DYNAFORM-PC prompts the user to create a local system. A thorough explanation of how to create a local coordinate system is covered in Chapter 1 *General Discussion* section under "Local Coordinate System."

    •   This coordinate system will define the U-V plane.

2. DYNAFORM-PC prompts:

    DO YOU ACCEPT THE DISPLAYED COORDINATE SYSTEM?

    ?   YES   Continue to Step 3.
        NO   Returns to Step 1.

3. Once the U-V plane onto which the points will be projected is defined, DYNAFORM-PC prompts via Pop-Up window:

    PROJECT NODES ALONG W-AXIS?

    Note: Nodes are projected only along a local W-axis. The user can choose to use the W-axis of the coordinate system defining the U-V plane or to create a new coordinate system.

    ?   YES   The nodes will be projected along the W-axis of the local coordinate system defining the U-V plane.
        NO   Repeats Step 1 to define a new W-axis for the projection vector.

4. Once the projection plane and vector are defined, DYNAFORM-PC prompts:

    SELECT NODE(S) TO PROJECT ONTO U-V PLANE

5. END SELECT will project the nodes and complete the command.

7.8.3   ONTO SURFACE

Projects a node or group of nodes onto any surface. Nodes may be referenced or unreferenced.

1. DYNAFORM-PC prompts:

SELECT SURFACES

- END SELECT will continue to the next step.

2. DYNAFORM-PC prompts:

DEFINE PROJECTION VECTOR
DEFINE A LOCAL COORDINATE SYSTEM
PICK POINT FOR THE ORIGIN

- DYNAFORM-PC prompts the user to create a local system. A thorough explanation of how to create a local coordinate system is covered in Chapter 1 *General Discussion* section under "Local Coordinate System."

    Note: **The nodes are projected only along the local W-axis. The user must define the local coordinate system accordingly.**

3. DYNAFORM-PC prompts:

DO YOU ACCEPT THE DISPLAYED COORDINATE SYSTEM?

?   YES   Continues to Step 4.
     NO   Returns to Step 2.

4. Once the local coordinate system is defined for the projection vector, DYNAFORM-PC prompts:

SELECT NODE(S) TO PROJECT ONTO SURFACE(S)

5. END SELECT will project the nodes and complete the command.

7.9   CHECK DUPLICATE NODES

Renumbers any duplicate node numbers found in the DYNAFORM-PC database. DYNAFORM-PC will not allow the user to create nodes with duplicate numbers. However, duplicate nodes may exist on imported files. The second and subsequent duplicate node numbers will begin renumbering at the "maximum node number +1" and will continue to increment.

1. Once this function is selected and duplicate nodes are found, the following messages echo to the screen:

CHECKING xxx NODES FOR DUPLICATE NODES
NODE xxx ALREADY EXISTS RENUMBERED TO yyy
xxx DUPLICATE NODES FOUND

2. If no duplicate nodes are found, the following messages echo to the screen:

CHECKING xxx NODES FOR DUPLICATE NODES
NO DUPLICATE NODES FOUND
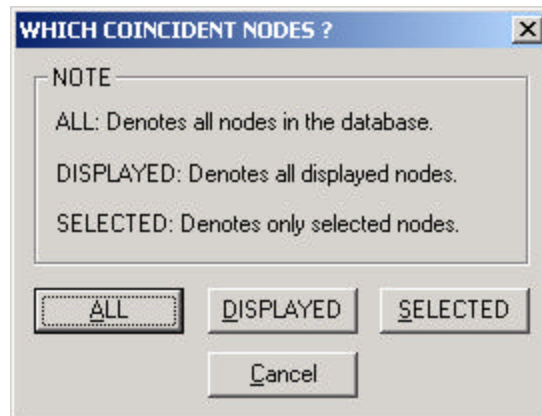
7.10   CHECK COINCIDENT NODES

Checks the displayed part(s) or the entire model for coincident nodes. Coincident nodes are defined as those nodes with a distance between each other that is less than or equal to a specified tolerance. The default tolerance, which is the minimal tolerance that DYNAFORM-PC will allow, is 0.01 units. The user may define a greater tolerance. Coincident nodes will merge to the lowest node number in the coincident node group (i.e., two nodes sharing the same location will be combined and renumbered to the lower grid number leaving the higher number node as an unreferenced node).

1. DYNAFORM-PC prompts via Pop-Up window as Figure 7.13.
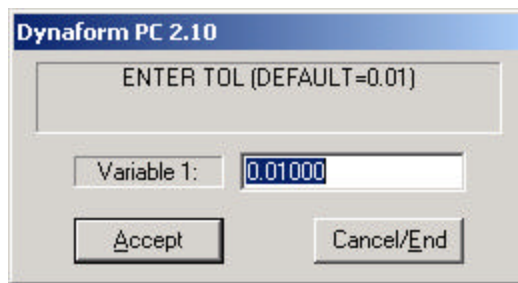
Figure 7.13 Enter Coincident Tolerance

- Any positive real number is a valid entry.

2. After entering a tolerance, DYNAFORM-PC prompts via Pop-Up window as Figure 7.14.

Figure 7.14 Which Coincident Nodes

- The user can select ALL, DISPLAYED or SELECTED.

ALL
Refers to each of the nodes in the database.

DISPLAYED
Nodes present in the parts that are currently turned on.

SELECTED
Nodes the user chooses.

- If coincident nodes are found, the following message will appear:

  XXXX COINCIDENT NODES FOUND
  MERGE COINCIDENT NODES (Y/N)

- Coincident nodes become unreferenced nodes and are represented by asterisks (*).

- Some or all of the unreferenced nodes may be deleted using the delete unreferenced node option.

7.11   SHOW PART CONNECTION

Displays information about the part connection.

1. DYNAFORM-PC prompts:

   SELECT NODE TO SHOW PART CONNECTION

2. Once the node is selected, the following message will echo to the command line:

   NODE xxxx IS CONNECTED TO: yyyy / PID= zzzz

   - If the node number entered is not used in the DYNAFORM-PC database file, the following message will echo to the command line:

   NODE IS NOT IN DATABASE

   - The prompt in Step 1 will repeat until END SELECT is selected.

7.12   CHANGE NODE NUMBER

Changes node numbers of existing nodes. Parts do not need to be on or current to have their node numbers changed.

1. DYNAFORM-PC prompts:

   SELECT NODE FOR NEW NODE NUMBER

2. Once the node is selected, DYNAFORM-PC prompts via Pop-Up window:

    ENTER NEW NODE NO. FOR XXX or [CR] FOR UNCHANGE

    •   Any unused positive integer is a valid entry. If the entered number is already
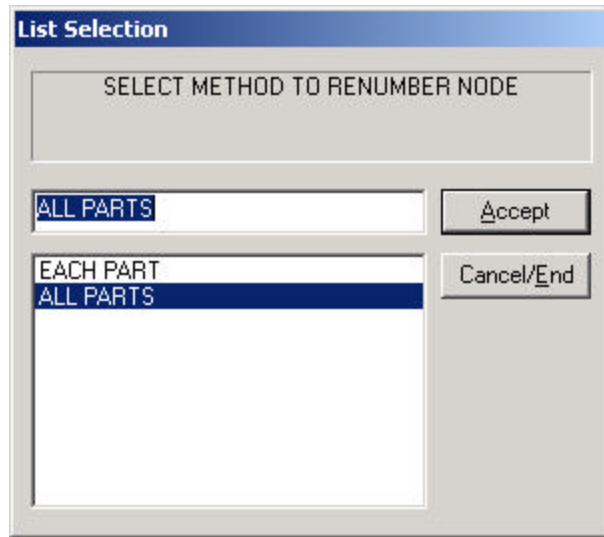   being used, DYNAFORM-PC prompts:

        NODE ID XXX ALREADY EXIST, REQUEST DENIED

    •   Select OK and DYNAFORM-PC will return the user to step 2.
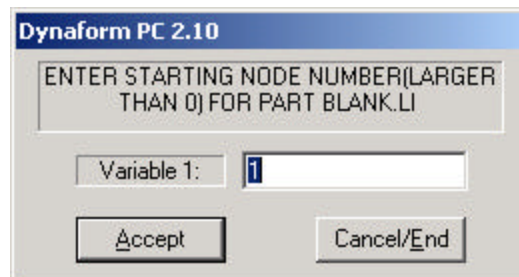
3. END SELECT will exit the function.

7.13   RENUMBER NODES

Renumbers all the nodes in the existing database. The user may renumber nodes by sequence
or parts (each part or all parts). Please see Figure 7.15.

Figure 7.15 Renumber Node Methods

1. If select EACH PART, DYNAFORM-PC prompts via Pop-Up window as Figure
7.16.

Figure 7.16 Enter Starting Node Number

- DYNAFORM-PC prompts:

  ENTER STARTING NODE NUMBER FOR PART (LARGER THAN 0)
  "xxxx"

- Once a number has been entered, DYNAFORM-PC prompts:

  NODE x TO xx ASSIGNED, NEXT NODE NO.: xxxx
  ENTER STARTING NODE NUMBER FOR PART (LARGER THAN 0)
  "xxxx"

- These prompts will continue for every part in the database.

- If a starting number is entered that is already used, the user is informed that
  the location is used and is sent back to the beginning of this step.

  NODE NUMBER HAS BEEN ASSIGNED

- Zero (0) will assign (MAX NODE NO. + 1).

3. If select ALL PARTS, DYNAFORM-PC prompts:

    ENTER STARTING NODE NUMBER (LARGER THAN 0)

- Any positive integer or zero will default to one (1).

    Once starting number is entered, DYNAFORM-PC will prompt a window for the
increment between node numbers as Figure 7.17.
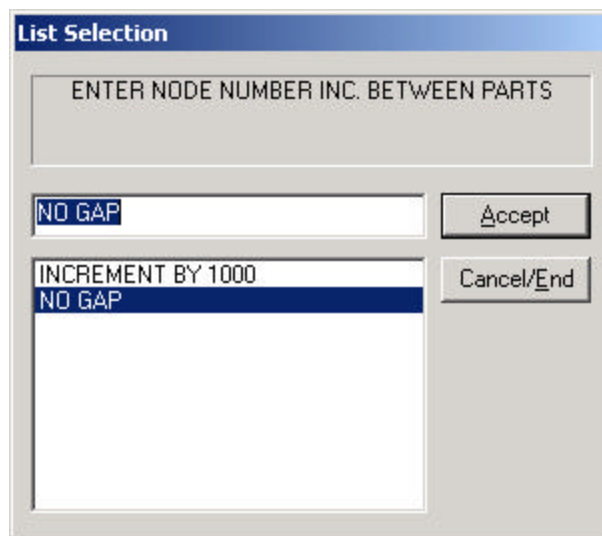
Figure 7.17 Enter Node Number Inc. Between Parts

INCREMENT BY 1000

NO GAP

- Select ACCEPT after an increment is selected and DYNAFORM-PC completes the function.
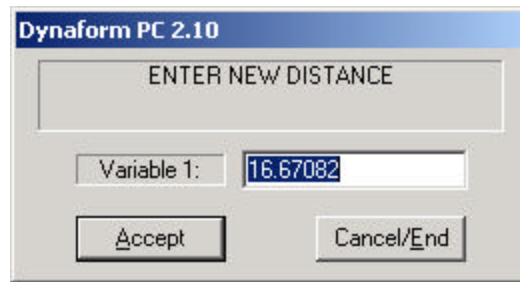
7.14   DISTANCE BETWEEN POINTS/NODES

Calculates the distance between two points, nodes or a node and point. After selecting two points/nodes, DYNAFORM-PC displays the distance:

D=XXXXX   DZ=XXXXX   DY=XXXXX   DZ=XXXXX

Adjust Magnitude allows user to adjust the distance between two selected nodes/points by adjusting the position of thefirst node/point along the reverse direction.
DYNAFORM-PC prompts:

Figure 7.18 Enter New Distance

After inputting the new distance, the first node/point will be moved to the new position.

Change Directional Dist, XYZ allows the user to adjust the distance between two selected nodes/points by changing the directional distance (DX, DY and DZ) of the second node/point.
DYNAFORM-PC prompts:
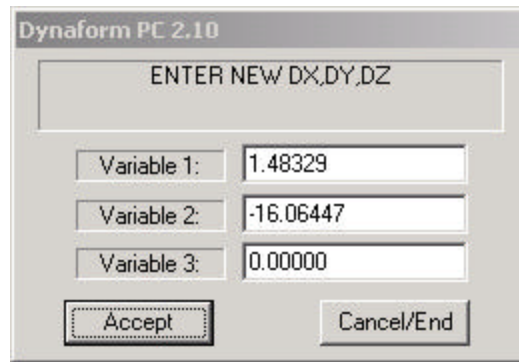
Figure 7.19 Enter New DX, DY, DZ

For details, see Section 12.2 UTILITY/DISTANCE BETWEEN 2 POINTS/NODES.

7.15   IDENTIFY POINTS/NODES

Identifies any point or node and its corresponding global location in XYZ coordinates.

1. DYNAFORM-PC prompts:

   SELECT A NODE FOR IDENTIFICATION

2. If node is selected, the number of the node nearest the cursor location is displayed. The following message will echo on the command line:

   NODE xxxx X = x.xxx Y = y.yyy Z = z.zzz

3. If point is selected, the point number nearest to the cursor location will display. The following message will echo on the command line:

   POINT xxxx X = x.xxx Y = y.yyy Z = z.zzz

4. END SELECT will exit the function.

After identifying a node, user can select Adjust Value to adjust the position of the selected node by defining the new coordinate location. Please see Figure 7.20.

Figure 7.20 Enter New Coordinate Location

7.16   FIND NODE

Locates and determines the coordinates of a specified node number by the input ID number.

1.   DYNAFORM-PC prompts via Pop-Up window:

Figure 7.21 Enter Node ID

- Any positive integer is a valid entry.

2.   If the node number entered is on the screen, the following message will echo the command line:

NODE nnnn X = x.xxxx Y = y.yyyy Z = z.zzzz

3.   If the node number entered is not used in the DYNAFORM-PC database file, the following message will echo to the command line:

NODE IS NOT IN DATABASE

- The prompt in Step 3 will repeat until CANCEL/END is selected.

CHAPTER 8
CHECK

The CHECK menu is used to validate DYNAFORM-PC models. The criteria values for model validation are set to defaults in DYNAFORM-PC, but may be adjusted to suit the user's needs. Element orientation, size, skew, connectivity and interior angles are compared to the specified criteria. The functions available are listed as Figure 8.1.
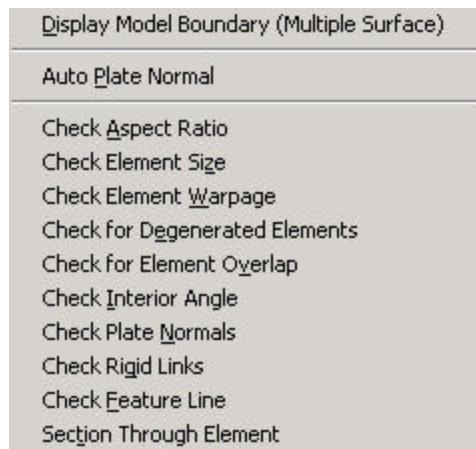
Figure 8.1 Model Check

A detailed description is provided in the following sections.

8.1   DISPLAY MODEL BOUNDARY

This function checks the boundary of the displayed part(s).

MULTIPLE SURFACE

This function checks the continuity of displayed elements. DYNAFORM-PC will highlight the non-intersecting free edges of multiple surfaces unless the intersections of the multiple surfaces are connected properly. In contrast to a Single Surface check, the common edge shared by three or more plate elements is not treated as a boundary. An example is given in Figure 8.2.
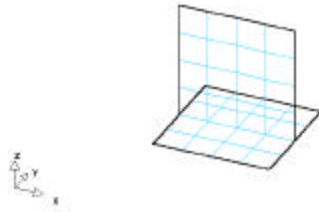
Figure 8.2 Multiple Surface

- The boundary will remain highlighted until the user selects the CLEAR function.

8.2   AUTO PLATE NORMAL

This function makes the element normal direction of all the plate elements currently displayed consistent with a selected reference plate element.

1. DYNAFORM-PC prompts:

    SELECT A PART FOR AUTO NORMAL

2. Select a part by clicking on one of its elements. The normal of the element will appear and the program will display a dialog window as Figure 8.3.
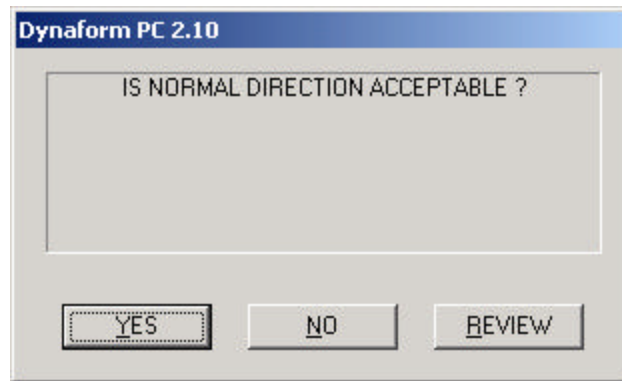
Figure 8.3 DYNAFORM-PC Question Window

?	YES	All displayed elements' normals will be consistent with the selected element's normal.
	NO	All displayed elements' normals will be the reverse of the normal of the selected element.
	REVIEW	Before deciding YES or NO, the user can review the elements by rotating the model. Exit review by pressing the left mouse button and select YES or NO in the OPTION LIST ICONS .

3. If the user selects YES and all the elements' normals are originally consistent, the program will prompt:

    ALL ELEMENT NORMALS ARE CONSISTENT

    If the user selects NO, the program will prompt:

    NORMAL REVERSED FOR XXXX ELEMENTS

    • All the elements whose normals are reversed are highlighted.

4. END SELECT will exit the function.

8.3   CHECK ASPECT RATIO

Checks 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. The default is 8:1.

1. DYNAFORM-PC prompts the user to enter the aspect ratio. Any real number is valid. Please see Figure 8.4.
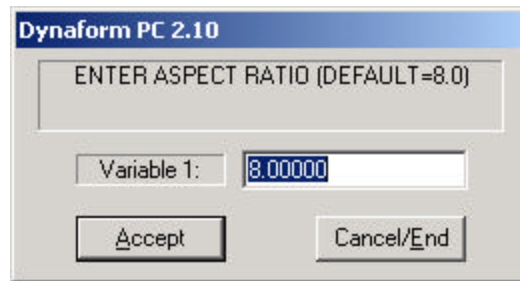
Figure 8.4 Enter Aspect Ratio

2. The aspect ratios are checked and the elements with aspect ratios greater than the desired limit are highlighted. The message echoed in the PROMPT WINDOW is the minimum element size and the number of elements that failed the check.

3. If elements fail the check, the following message will be displayed for the failed elements via Pop-Up window:

    INCLUDE FAILED ELEMENTS IN A NEW PART?

 | ? | YES | Prompts for a new part name. The PART ATTRIBUTES dialog box appears as Figure 8.5. The user can edit accordingly. |
    | | NO | Exits this command. |
    | | REVIEW | Rotate and review the model before selecting YES or NO. |
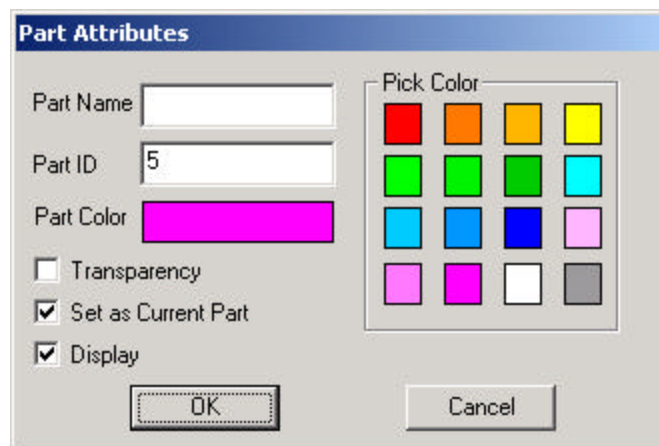
**Figure 8.5 Part Attribute**

8.4   CHECK ELEMENT SIZE

This function checks elements for minimum edge length. All elements smaller than the user defined length criterion will be highlighted.

1. DYNAFORM-PC prompts the user to enter length criteria via Pop-Up window. A valid response is any real number greater than zero.
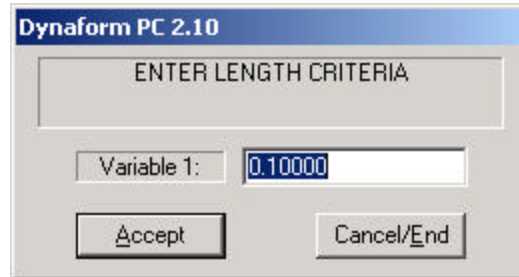
Figure 8.6 Enter Length Criteria

2. If all the edge lengths of the displayed elements are greater than the criterion, then the program will echo the message that all elements pass the check.

3. If any edge lengths of the displayed elements are less than the criterion then the program will echo which and how many elements failed the check.

4. If there are failed elements, DYNAFORM-PC prompts the user via Pop-Up window as Figure 8.7 to handle these elements:
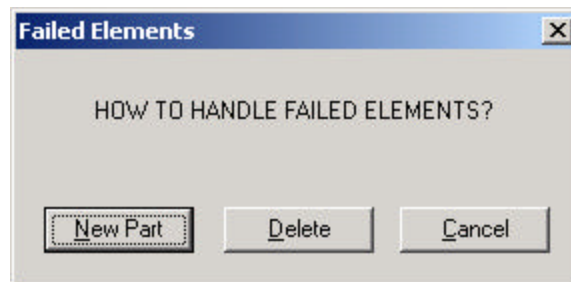

Figure 8.7 Handle Failed Element Question Window

?	New Part	Prompts for a new part name. Please refer to Figure 8.5.
	Delete	Delete all failed elements.
	CANCEL	Cancel the operation and exit the function.

8.5   CHECK ELEMENT WARPAGE

Checks the warpage (skew) of displayed plate and solid elements. DYNAFORM-PC highlights all warped elements that fail the warpage criterion. The user has the option to split the failed plate elements into triangular elements. All failed elements will be added to a new part unless otherwise specified.

1. DYNAFORM-PC prompts the user to enter criteria. Any real number is a valid entry.
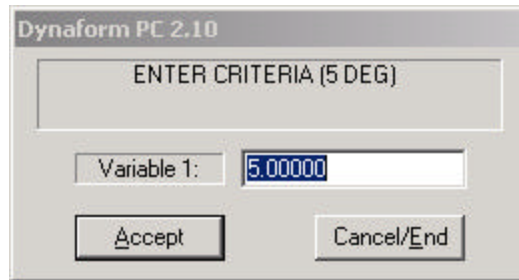
Figure 8.8 Enter Criteria for Checking Warpage

2. DYNAFORM-PC prompts the outcome of the check for the entered criteria. Step 3 is not applicable for solid elements, skip to Step 4.

3. The user is prompted to replace warped plate elements with triangular elements. If the user selects NO, continue to Step 4.

4. DYNAFORM-PC prompts:

    INCLUDE FAILED ELEMENTS IN A NEW PART?

    ?   YES   Continue to Step 5.
        NO   The warped elements will remain in their original parts and exits the function.

5. DYNAFORM-PC asks for a part name via Pop-Up window. The user can enter a name for the new part. Plates and/or solids will be added to this part and become current. Refer to Figure 8.5.

8.6   CHECK FOR DEGENERATED ELEMENTS

A degenerated element is an element that has two or more nodes that are identical in the node ID. DYNAFORM-PC lists all the elements that meet the criteria. If there are de-generated elements, DYNAFORM-PC prompts:

     xxxx ELEMENTS HAVE BEEN HIGHLIGHTED AS DE-GENERATED
    INCULDE FAILED ELEMENTS IN A NEW PART?

    ? YES   The user will be prompted for a new part name. Refer to Figure 8.5.
      NO   Exits the function.
        REVIEW   User can rotate the display window and decide whether input the failed element in a new part or exit the function.

8.7   CHECK FOR ELEMENT OVERLAP

An element that shares more than three nodes with another plate element is an overlapping element. DYNAFORM-PC lists all the elements that overlap.

1. If no overlapped elements are found in the database, a message displays in the PROMPT WINDOW of the number of elements that passed check.

2. If overlapped elements are found in the database, a message displays in the PROMPT AREA of the number elements that overlap. The overlapped elements are also highlighted and the user has the option of placing them in a new part:

    PLACE OVERLAPPED ELEMENTS IN A NEW PART?

    ?   YES   The user will be prompted for a new part name. Refer to Figure 8.5.
        NO   Exits the function.

8.8   CHECK INTERIOR ANGLE

This command checks the interior angles of all displayed plate and solid elements. DYNAFORM-PC checks and highlights all elements with interior angles less than the default user defined value.

1. DYNAFORM-PC prompts via Pop-Up window:

    ENTER INTERIOR ANGLE (DEFAULT = 30 DEGREES)

2. If all the elements pass the model check, the following message will be echoed to the user:

    ALL ELEMENTS PASS CHECK!

3. If all the elements do not pass the model check, the program will display which element(s) failed the check and the following message will appear:

    XXXX ELEMENTS FAILED CHECK

4. If there are failed elements, the program prompts the user via Pop-Up window to place these elements into a new part:

    INCLUDE FAILED ELEMENTS IN A NEW PART?

    ?   YES   Proceeds to the PART ATTRIBUTE dialog box (Figure 8.5)
        NO   Exits the function

8.9   CHECK PLATE NORMALS

Draws a white boundary line between displayed elements with opposite normal directions.

1. If all the elements have the same direction normals, the following message will be echoed:

   NORMAL CHECK COMPLETED, NORMAL IS CONSISTENT

2. If there are any elements with opposing normals, the program will highlight the boundary between these elements and others. It will echo the following message:

   PLATE NORMAL CHECK COMPLETED

8.10   CHECK RIGID LINKS

Checks the connectivity and length of all rigid elements. Rigid bodies that are loose, zero length, double dependent, cyclic, redundant and exceed the user's length criteria will be highlighted.

1. DYNAFORM-PC prompts via Pop-Up window:

   LIST RIGID LINKS THAT ARE LONGER THAN A CERTAIN LENGTH?

   ?   YES   Continue to Step 2.
       NO   Continue to Step 3.

2. DYNAFORM-PC prompts:

   ENTER LENGTH CRITERIA

   • Any real number zero or greater is acceptable.

3. DYNAFORM-PC prompts:

   ENTER NO. OF RIGID LINK STACKS TO BE CHECKED (<10)

4. The user has the option to place failed elements into new parts. If the user selects the option to place the failed elements into a new part, the following part names will automatically be created:

r.xlong	RIGID links that fail length check.
r.duplic	Duplicated dependent node in a RIGID link.
r.cuclic	Two RIGID links sharing the same two nodes in an independent / dependent node loop.
r.double	RIGID links with double dependent nodes.
r.loose	Loose connection RIGID links.

8.11   CHECK FEATURE LINE

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

1. DYNAFORM-PC prompts via Pop-Up window:

    ENTER FEATURE LINE ANGLE CRITERION

    *   The default angle is 20 degrees.

2. Once the angle is entered, any feature line where the angle is larger than or equal to the angle criteria will be highlighted and DYNAFORM-PC prompts:

    GENERATE PLOTEL ELEMENTS FOR FEATURE LINES

    ?   YES   Continue to Step 3.
        NO   Exits the function.

3. DYNAFORM-PC prompts:

    INCLUDE PLOTEL ELEMENTS IN A NEW PART?

    ?   YES   The Part Attributes window displays (Figure 8.5).
        NO   Exits the function.

8.12   SECTION THROUGH ELEMENT

This command computes and creates a set of section lines composed of display elements (PLOTEL) at a user defined U-V plane.

1. DYNAFORM-PC prompts:

    DEFINE A LOCAL COORDINATE SYSTEM
    PICK POINT FOR THE ORIGIN

    *   DYNAFORM-PC requires a local coordinate system UVW in which to create section display elements (PLOTEL). A thorough explanation of how to create a local coordinate system is covered in Chapter 1 *General Discussion* section under "Local Coordinate System."
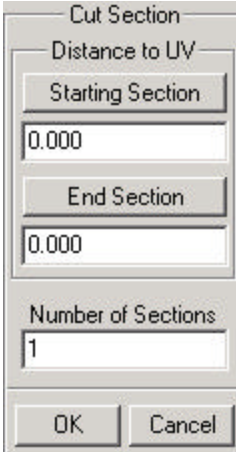
2. Once the desired coordinate system is defined. DYNAFORM-PC prompts via Pop-Up window:

    DO YOU ACCEPT THE DISPLAYED COORDINATE SYSTEM?

    ?   YES   Continue to Step 3.
        NO   Returns to Step 1.

3. DYNAFORM-PC displays the following menu to define the section line:

Figure 8.9 Cut Section

4. The user can define the start and end section plane by either clicking the STARTING SECTION and END SECTION button to select the section plane by node or input the location in local W-coordinates in the field below each button. The user may also specify the number of sections by inputting a number in the field provided.

5. Once the desired information is input, select OK to perform and exit the function.

TOOLS

The TOOLS menu contains commands used to create and manage tools. The user can create tool pieces via CAD data, define material, define property, generate load curve, define interface contact parameters, position tool pieces, create drawbeads, modify the positions of defined tools and modify deformed blank shape. This menu may be accessed through the TOOLS selection card located in the CONTROL WINDOW. The functions are organized as Figure 9.1.

Figure 9.1 Tools Options

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

9.1   DEFINE TOOLS

In DYNAFORM-PC, there are four standard tools: die, punch, upper ring and lower ring. The user can also create user-defined tools. Please see Figure 9.2.
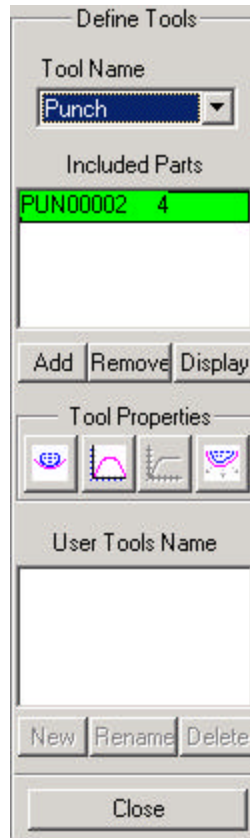
Figure 9.2 Define Tool

1. Select a tool type from the TOOL NAME pull-down menu as in Figure 9.3:
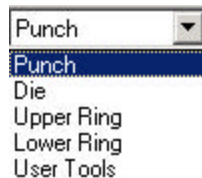
Figure 9.3 Tool Name List

- If USER TOOLS is selected, the window is activated as Figure 9.4.
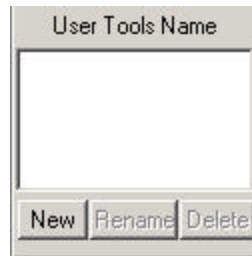
Figure 9.4 User Tool

- The user can create, rename and delete the user-defined tools.

  NEW
  In general, a tool is a set of parts containing finite elements. This function will let the user put the parts into the current tool.

- Once NEW is selected, DYNAFORM-PC prompts:

  SELECT PART FOR DEFINE XXX
  PICK AN ELEMENT TO IDENTIFY THE PART

- Once the part is selected, the name is added into the part list box automatically. Please see Include Parts List in Figure 9.2.

  REMOVE
  Select the part name from the part list box and click REMOVE.

  DISPLAY
  Select the part name from the part list box and click DISPLAY to display the tool.

2. TOOL PROPERTIES
   In DYNAFORM-PC, there are four interfaces defined as follows:

   - DEFINE CONTACT PARAMETERS
   - DEFINE MOTION CURVE
   - DEFINE FORCE CURVE
   - TOOL OFFSET

   The interface parameters will be described in the following pages.

3. Select CLOSE to exit the function.

9.1.1   DEFINE CONTACT PARAMETERS

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

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

1. A dialog window displays as Figure 9.5.
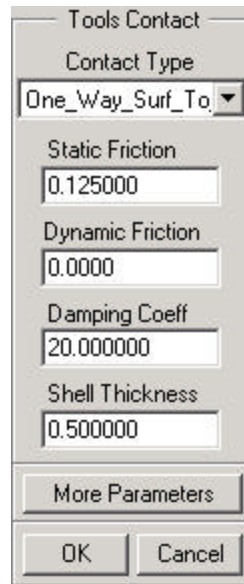

Figure 9.5 Tools Contact

2. Define the CONTACT TYPE
    There are nine types of contact DYNAFORM-PC supports:

          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

    •   The user can change the default values in the fields provided in the CARD
        DEFINITION dialog window.

3. MORE PARAMETERS
    This allows the user to define additional parameters related to the contact
    definition. Please see Figure 9.6.
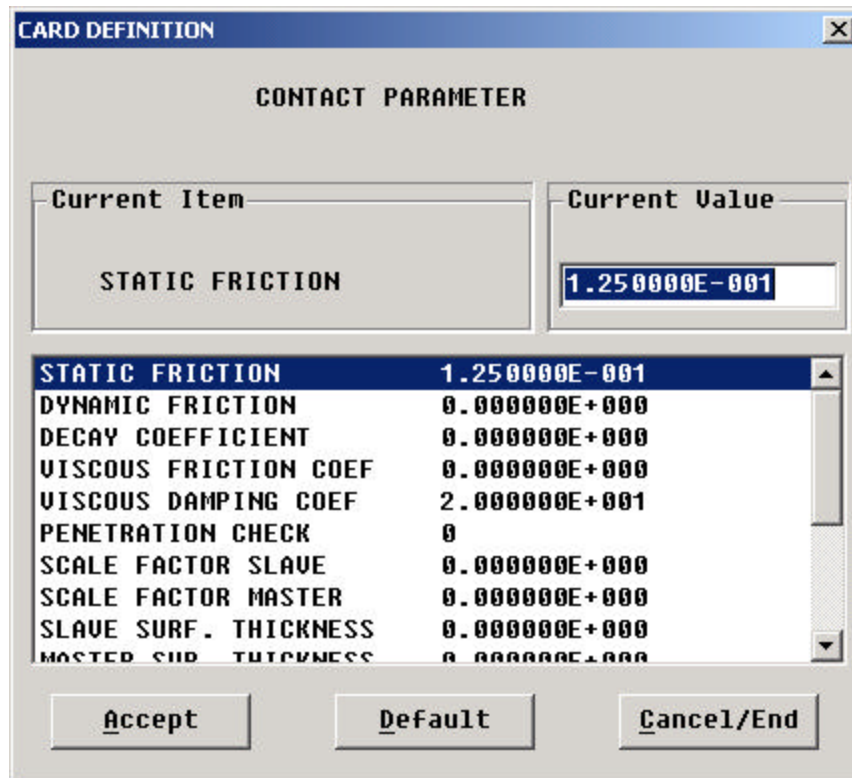
Figure 9.6 Contact Parameter

    4.   Select OK to finish the operation.

9.1.2   DEFINE MOTION CURVE

This function allows the user to generate a complete motion curve for the tool by a few parameters.

    1.   A dialog window displays the following to define the parameters:
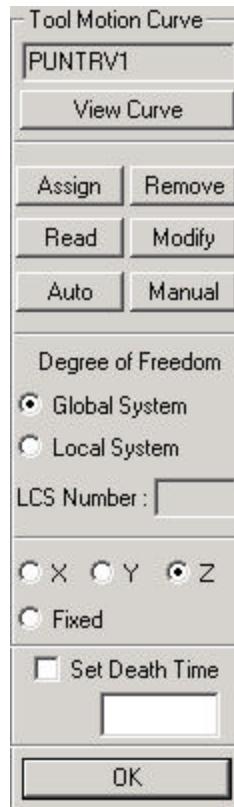
Figure 9.7 Tool Motion Curve

2. VIEW CURVE
    DYNAFORM-PC prompts:
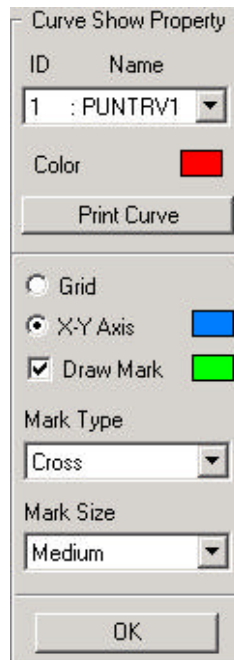
Figure 9.8 View Curve

The user is able to select from a list of available curves from the pull-down menu provided and the curve is displayed in the display window. The default is the curve list in the static edit window on Figure 9.8.

In this function the user is allowed to print the displayed curve, change the display appearance, select mark type and mark size.

3. ASSIGN
   DYNAFORM-PC prompts a window as Figure 9.9.
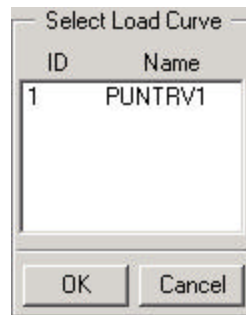
Figure 9.9 Select Load Curve

User can select a curve from the list window as Figure 9.9. DYNAFORM-PC will assign the selected curve to the selected tool as the motion curve.

4. REMOVE
   DYNAFORM-PC prompts:
   REMOVE THE CURVE FROM TOOL DEFINITION?

   ? YES   Removes the curve
     NO   Cancels the operation.

5. READ
   DYNAFORM-PC allows the user to import selected *.cur file. Once a curve is read, it will be saved into the current database.

6. MODIFY
   DYNAFORM-PC allows the user to modify the name, number and application of the curve using the following interface:
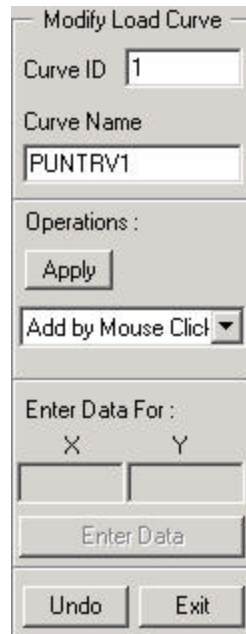
Figure 9.12 Modify Load Curve

The user can also modify the curve using the following functions, please see
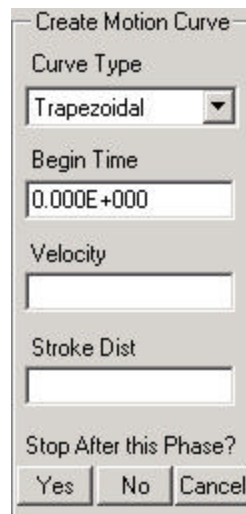Figure 9.13.

Figure 9.13 Operation Type

- Add by Mouse Click
  The point that the user clicks with the left mouse will be added into the curve

- Add by Key In X-Y
  The Enter Data Function will be activated. Please see Figure 9.14. After
  entering X value and Y value, user selects Enter Data, the point will be add
 into the current curve.

Figure 9.14 Enter Data

- Delete by Mouse Click
  The point nearest the mouse clicking will be deleted from the current curve.

- Negate
  The point nearest the mouse clicking will be replaced with the point that is symmetrical along X-axis.

- Scale
  The selected point will be scaled X and Y values along X-axis and Y-axis respectively.

- Query Value
  Values of selected point will be displayed in the message window.

7. Auto
   DYNAFORM-PC will prompt a window as Figure 9.15.

Figure 9.15 Create Motion Curve

- Select a curve type from the CURVE TYPE pull down menu.
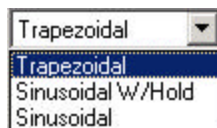
Figure 9.16 Curve Type

There are 3 types of load curve that can be generated automatically through this function:

SINUSOIDAL W/ HOLD	Same as sinusoidal except the maximum velocity will be held constant. This is the default curve.
SINUSOIDAL	Used to simulate the cam action for the tool.
TRAPEZOIDAL	Simplified analytical load curve.

- Input the values of BEGIN TIME, VELOCITY and STROKE DISTANCE fields.
  BEGIN TIME is the start time when the curve works.
  VELOCITY is the tool motion velocity.
  STROKE DISTANCE is the tool motion distance.
- Stop after This Phase?
  - ? YES   Figure 9.8 is displayed allowed the user to view the curve.
  -    NO   Figure 9.15 appears again and the Begin Time is the end time of the last created curve. Allowed user to define different velocity.
  - CANCEL   Cancels the operation and exits the function.

8. Manual
    The function allows user to define the curve by manual operation. DNAFORM-PC prompts a window as Figure 9.17.
    If select Velocity/Time Curve, the Y data is the velocity in the created curve and if select Displacement/Time Curve, the Y data is the displacement in the created curve.
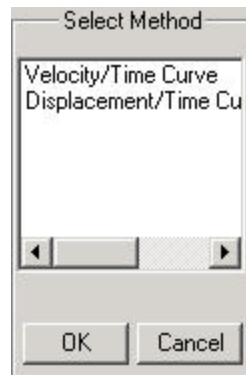
Figure 9.17 Curve Type

Once a curve type is selected, DYNAFORM-PC will prompt a window as Figure 9.18.

Figure 9.18 Create Load Curve

User can define the curve ID and name and the default curve ID is the largest ID plus 1 in the current database. Point List lists the created points. X data and Y data allow user to input the value to define the point.

- Add Point

  The function adds the point defined by X data and Y data

- Modify Point

  User selects a point in the Point List and the point values will appear in X data and Y data. After changing them and selecting Modify Point, the selected point will be modified.

- Delete Point

  The function allows user to delete the selected point in the Point List.

- Create

  Create a curve from points listed in the Point List. And DYNAFORM-PC prompts the window as Figure 9.18.

- Exit

  Exit the function without creating any curve.

9. Degree of Freedom

   If check on Global System, the degree of freedom can be one of X, Y and Z. If check on Local System, the degree of freedom can be one of U, V and W. If select Fixed, the tool doesn't move.

10. Set Death Time
   The default death time is the end of the curve time. User can also set death time
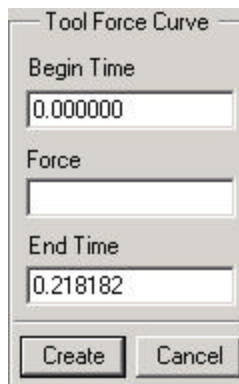   to decide the tool motion time.

11. Select OK to close the TOOL MOTION CURVE window.

9.1.3   DEFINE FORCE CURVE

This function is effective only when defining the upper ring, lower ring or a user-defined tool.
Please see Figure 9.19.
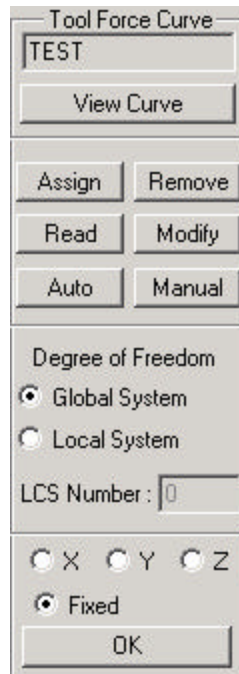
Figure 9.19 Tool Force Curve

Figure 9.20 Auto Tool Force Curve

If select Auto to define the tool force curve, DYNAFORM-PC prompts a window as Figure 9.20.

- Input parameter values of BEGIN TIME, FORCE and END TIME.

- Select CREATE and a curve window will display.

- Select Cancel to return to the Tool Definition window.

Other options of this function are same with those in DEFINE MOTION..

9.1.4   TOOLS OFFSET

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. A dialog window is displayed as in Figure 9.21:
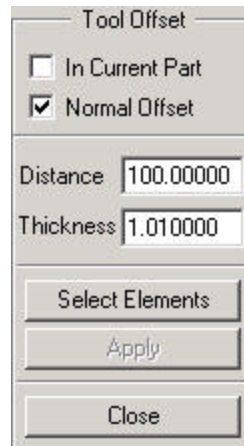
Figure 9.21 Tool Offset

IN CURRENT PART (toggle)
If it is toggled on, the newly 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.

NORMAL OFFSET (toggle)
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.

DISTANCE
The DISTANCE value is used for moving the new elements in the stroke direction.

THICKNESS
The THICKNESS value is used for offsetting the elements in the normal direction. A negative value will cause the elements to be offset in a negative normal direction.
SELECT ELEMENTS
The function is used for selecting the elements.

2. Click SELECT ELEMENTS, DYNAFORM-PC prompts:

    SELECT ELEMENTS
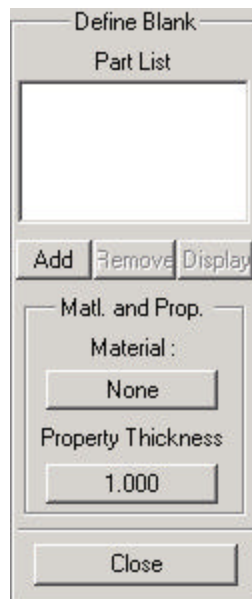
    • END SELECT will continue to the next step.

3. Select APPLY to offset the tool.

4. Select CLOSE to return to the tool definition.

9.2   DEFINE BLANK

This function is used to define a blank.

1. A dialog window is displayed as in Figure 9.22:

Figure 9.22 Define Blank

2. Select ADD, DYNAFORM-PC prompts:

   SELECT PART FOR DEFINE BLANK
   PICK AN ELEMENT TO IDENTIFY THE PART

   - Once the part is selected, its name is added into the part list box automatically. Please see Figure 9.24.
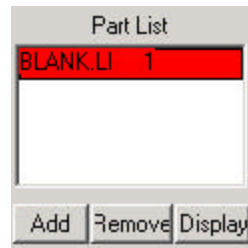
Figure 9.23 Part List

   - Once the blank is added into the list box, REMOVE and DISPLAY are activated. Please see Figure 9.23.

     MATERIAL
     The user can define a new material or select an existing material for the current blank. See DEFINE MATERIAL in Section 9.4.

     PROPERTY
     The user can define a new property or select an existing property for the current blank. See DEFINE PROPERTY in Section 9.5.

3. Select CLOSE to exit the function.

9.3   DRAWBEAD

The function is used to create, modify and delete drawbeads. Please see Figure 9.24.
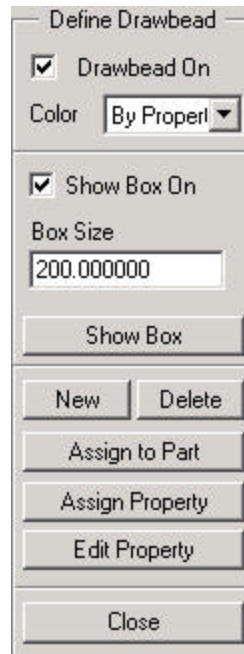
Figure 9.24 Define Drawbead

1. DRAWBEAD ON
   Toggle this function ON to display the bead lines.

2. DRAWBEAD COLOR

   - Select BY PART to display the drawbead in the color of its assigned part.

   - Select BY PROPERTY to display each drawbead in the color of its assigned property.

3. SHOW BOX
   The SHOW BOX function is used to define a box around the drawbead to limit the size of the master surface considered for the drawbead. It is also used to display graphically, the drawbead boxes that have been defined.

   - Select SHOW BOX ON to activate the BOX SIZE field and the SHOW BOX button.

   - Type the desired size in the box size.

   - Select the SHOW BOX button to display the drawbead box.

4. NEW
   This allows the user to create a drawbead.

- Select NEW and the property definition dialog window is displayed as Figure 9.25. Please see EDIT PROPERTY to get information of defining property.

- Define the desired property, select ADD and then OK.
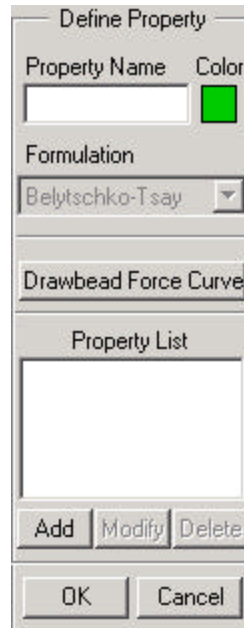
Figure 9.25 Define Property

- DYNAFORM-PC prompts via Pop-Up window as Figure 9.26.
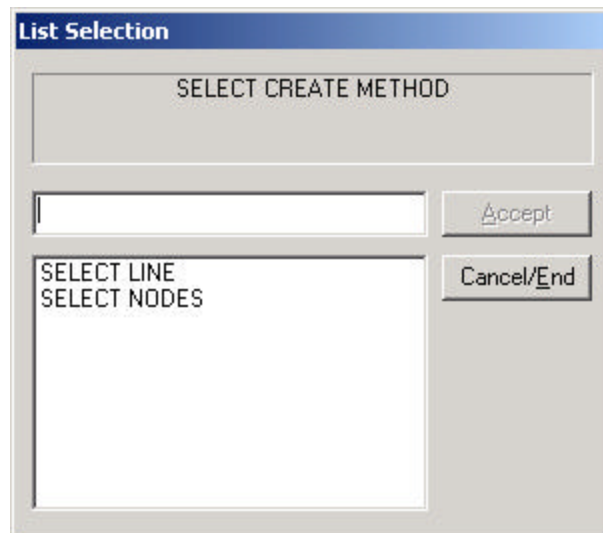
Figure 9.26 Select Create Method

This window allows the user to choose a way to create beads.

- SELECT LINE
  Users can create a drawbead through the selection of a displayed geometric line. Once the desired line(s) are selected, END SELECT will complete the function.

- SELECT NODES
  Users can create drawbead through node selection. Once the desired nodes are selected, END SELECT will complete the function.

5. DELETE DRAWBEAD

   - DYNAFORM-PC prompts:

     SELECT DRAWBEAD
     PICK A DRAWBEAD

     SELECT DRAWBEAD OPTIONS

      Select By Cursor (default)

      Select by Property
     User can select a bead with Cursor and all drawbeads that have the same property as its will be selected. User can also select a property from property list and all drawbeads with the selected property will be selected.

   - Once the desired drawbeads are selected, END SELECT will complete the function.

6. ASSIGN TO PART
   This function assigns a drawbead to a rigid body part.

   - DYNAFORM-PC prompts:

     SELECT TARGET PART (DIE OR BINDER)
     PICK AN ELEMENT TO IDENTIFY THE PART
   - Once the part is selected, DYNAFORM-PC prompts:

     SELECT DRAWBEAD

   - Once the drawbeads are selected, END SELECT will complete the function.

7. ASSIGN PROPERTY
   This function allows the user to change the property for a defined drawbead.

   - Select the property from the property list box.

   - DYNAFORM-PC prompts:

SELECT DRAWBEAD PROPERTY

- Once the drawbeads are selected, END SELECT will complete the function.

8. EDIT PROPERTY

- A dialog window is displayed as Figure 9.25.

- Enter the property name in the PROPERTY NAME field and COLOR will change the display color of the property.

- Select ADD and the user can define the relative parameters via Pop-Up window as Figure 9.27
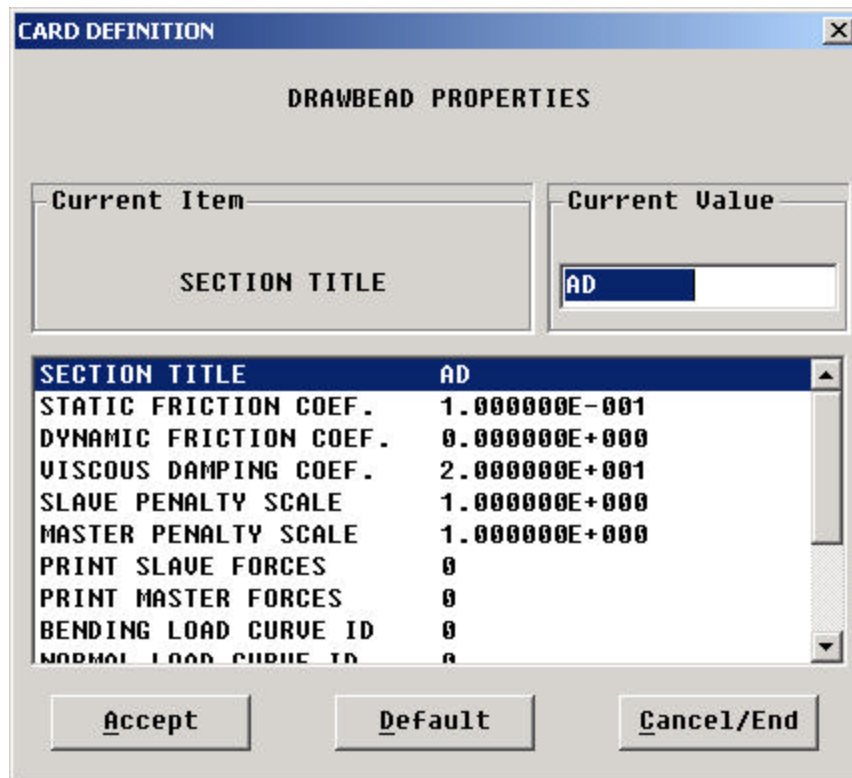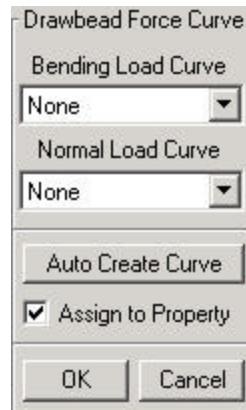

Figure 9.27 Drawbead Property Card

- Click on any variable name to edit the data.

- Select ACCEPT to close the dialog window.

- Select DRAWBEAD FORCE CURVE and a dialog window displays as Figure 9.28.

Figure 9.28 Drawbead Force Curve

- AUTO CREATE CURVE
  The function is used to create a BENDING load curve and a NORMAL load curve automatically, according the geometry of the draw bead. Please see Figure 9.29.
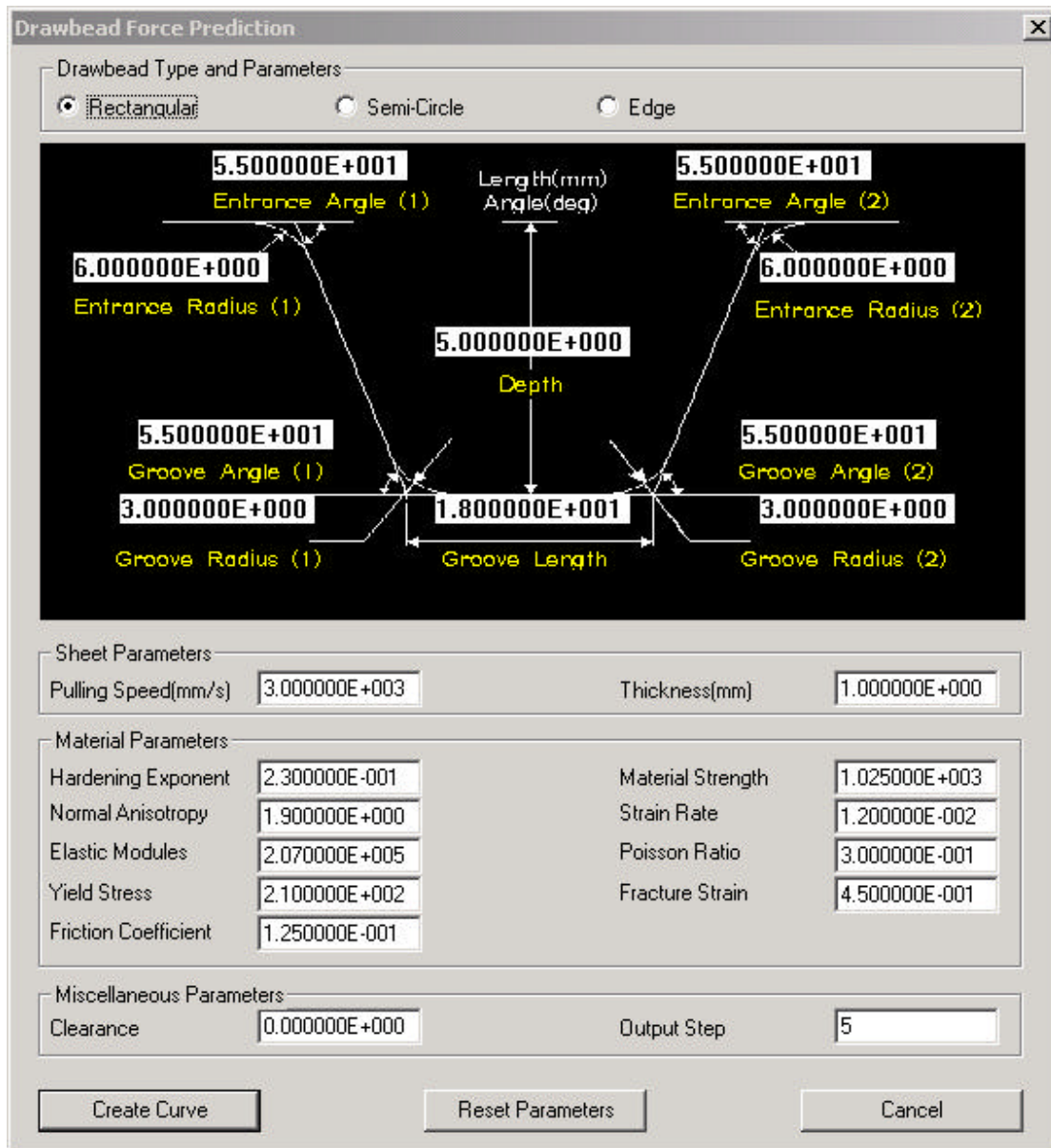
Figure 9.29 Drawbead Force Prediction

- A dialog box is displayed (Figure 9.29) to define the geometry and material property of a drawbead. Click on any data field to define or modify the value.

- Select CREATE CURVE and close the Pop-Up curve window. The defined curves are added into the current curve list box.

- ASSIGN TO PROPERTY (toggle)
  If toggled ON, the current property is assigned to the curve.

- Select OK to return to the property definition dialog window.

- Once the property is defined, select OK to exit.

9.4   DEFINE MATERIAL

These functions are used to create, modify and delete material, as well as save and read to the material library.
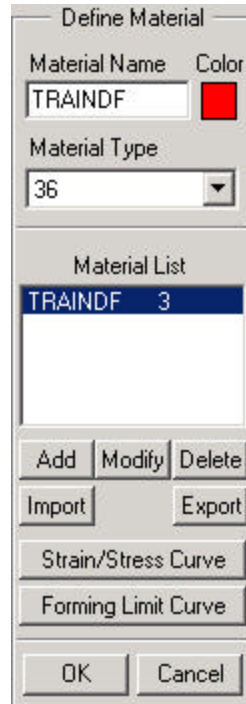
Figure 9.30 Define Material

1. CREATE MATERIAL

   - Enter the material name in the MATERIAL NAME edit field and select a color from the box provided right of the edit field.

   - Select the desired material type from the MATERIAL TYPE pull-down menu. There are five types of blank materials in DYNAFORM-PC. For detailed descriptions of these materials, refer to the *LS-DYNA User's Manual.* The default material type is type 36.

   - Select ADD and a material dialog window displays as Figure 9.31.

   - Select ACCEPT to close the dialog window, a new material is added into the MATERIAL LIST.

Figure 9.31 Material Card

2. MODIFY MATERIAL

    • Select a name in the MATERIAL LIST window.

    • Select MODIFY and change the values in the Pop-Up window.
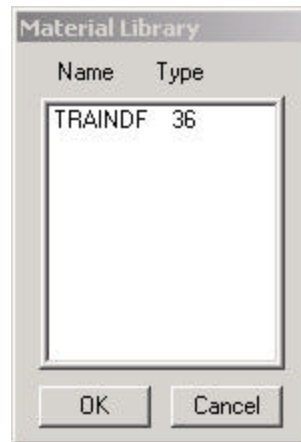
3. DELETE MATERIAL

    • Select a name in the MATERIAL list window.

    • Select DELETE.

4. IMPORT
    The function is used to import the material from the material library.

    • A Pop-Up window displays the available materials in the library. Please see
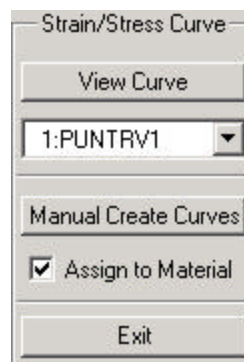   Figure 9.32.

Figure 9.32 Material Library

- Select the material from the window and OK to read the material into the database. The material properties will be displayed in a material dialog box for the user to edit.

5. EXPORT
The material library consists of two files: **material.ind** and **material.lib**. **Material.ind** contains a list of names of all materials saved in the library. **Material.lib** contains all of the material information saved in the library. The first time the user exports material data, the two files are automatically created into the current directory of the user's computer.

- Select a name in the MATERIAL list window.

- Select EXPORT. The information about the selected material will be appended to those files.

6. STRAIN/STRESS CURVE
The function is used to create the stress/strain curves of materials.

- Select STRESS/STRAIN CURVE and the following displays:
-

Figure 9.33 Strain/Stress Curve

- VIEW CURVE
  Once the curve is defined and assigned to the current material, the curve name is shown in the VIEW CURVE window. Select VIEW CURVE to retrieve the curve. If no curve is assigned to the current material, no curve names will be displayed.

- MANUAL CREATE CURVES
  This function is used to define a curve through a set of point. See Section 12.6 UTILITY/LOAD CURVE for additional information.

- ASSIGN TO MATERIAL (toggle)
  If toggled ON, the defined curve is assigned to the current material automatically.

- Select EXIT to exit the curve definition.

7. FORMING LIMIT CURVE
   This function is used to create, read and display the FLD curve of the materials. These options are similar to those in Section 12.6 UTILITY/LOAD CURVE.

Figure 9.34 Forming Limit Curve

9.5   DEFINE PROPERTY

These functions are used to create, modify and delete property. Please see Figure 9.35.
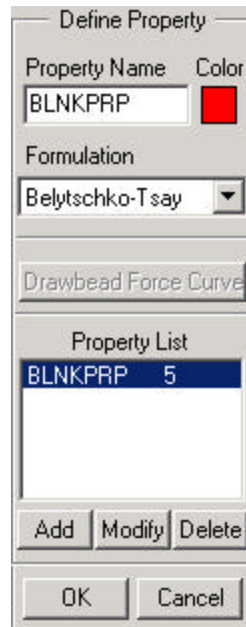
Figure 9.35 Define Property

1. CREATE PROPERTY

    - Enter the material name in the PROPERTY NAME field.

    - Select the desired property color from the COLOR box next to the PROPERTY NAME field.

    - Select the desired property type from the FORMULATION pull-down menu.

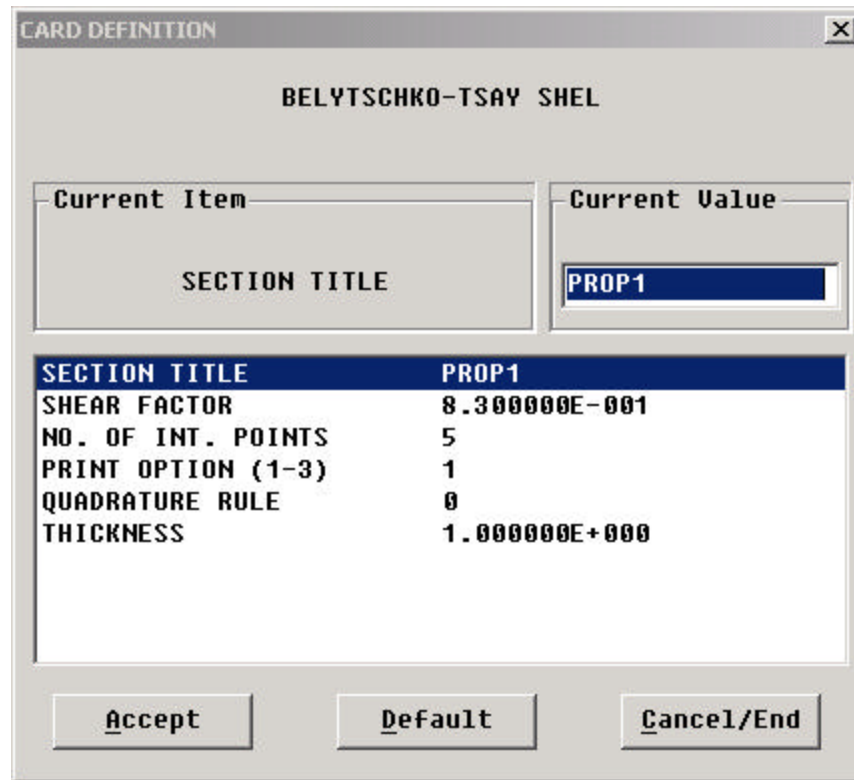   - Select ADD and the CARD DEFINITION window displays.

Figure 9.36 Property Card

- Click on any variable name to edit the data.

- Select ACCEPT to close the dialog window, the property is added into the PROPERTY LIST.

  Note: **There are eleven types of blank materials in DYNAFORM-PC, for detailed descriptions of these properties see the** *LS-DYNA User's Manual*.

2. MODIFY PROPERTY

   - Select a name in the PROPERTY LIST window.

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

3. DELETE PROPERTY

   - Select a name in the PROPERTY list window.

   - Select DELETE.

4. Select OK to exit the function.

9.6   BLANK SIZE ESTIMATE

This function provides a blanks size estimate based on the original part and the blank material and thickness. Before beginning the blank estimation function the desired part should be meshed with a tool mesh. The mesh should contain no overlapping elements, gaps or cracks, especially on the part edges. The blank size estimation interface follows:
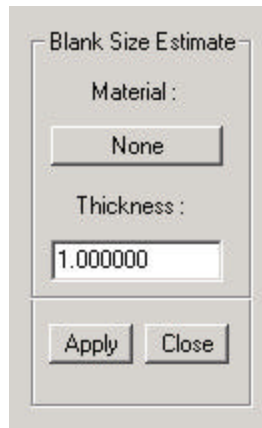
Figure 9.37 Blank Size Estimate

Select material button to define the blank material, please refer to section 9.4 DEFINE MATERIAL to get detailed information about defining material. After defining the material and inputting the blank thickness, select APPLY;DYNAFORM-PC will estimate the blank size and output a blank outline. This line represents the trimline of the part. This line can be output as an iges file.

9.7   BLANK MAPPING

A deformed coarse blank mesh for binder wrap simulation needs to be refined for draw die simulation. This function allows the user to map a flat fine blank mesh onto the deformed binder wrap shape.

1. DYNAFORM-PC prompts:

    SELECT COARSE MESH FOR THE FLAT BLANK
    PICK AN ELEMENT TO IDENTIFY THE PART

2. After selecting the flat blank with the coarse mesh, DYNAFORM-PC will prompt:

    SELECT COARSE MESH FOR THE DEFORMED BLANK

3. Once the user selects the deformed blank with the coarse mesh, DYNAFORM-PC prompts:

    SELECT FINE MESH FOR THE FLAT BLANK

4. Once the user has selected the blank with the fine mesh, DYNAFORM-PC will automatically map the fine blank mesh to the deformed blank shape.

9.8   BLANK TRIMMING

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. DYNAFORM-PC prompts:

    SELECT TRIM LINES
    SELECT LINE(S)

    *   Once the desired lines are selected, END SELECT will continue to the next step.

2. Once the desired lines have been selected, DYNAFORM-PC prompts:

    DEFINE A LOCAL COORDINATE SYSTEM
    PICK POINT FOR THE ORIGIN

    *   DYNAFORM-PC prompts the user to create a local system. A thorough explanation of how to create a local coordinate system is covered in Chapter 1 *General Discussion* section under "Local Coordinate System."

3. Once the desired coordinate system is acquired, DYNAFORM-PC prompts via Pop-Up window:
    DO YOU ACCEPT THE DISPLAYED COORDINATE SYSTEM?

    ? YES   Continues to Step 4.
       NO   Returns to Step 2.

4. DYNAFORM-PC prompts a window as Figure 9.38.

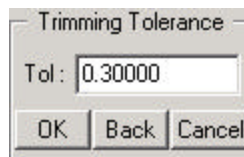   ENTER TOLERANCE (0.0 - 1.0) DEFAULT 0.3

Figure 9.38 Trimming Tolerance

5. Select OK, DYNAFORM-PC prompts:

   TRIM OUT ELEMENTS OUTSIDE THE TRIM LINE

6. Select YES or NO to complete and exit the function. YES will cause elements out
   the trim line to be trimmed.   These elements will be included in the part
   TRIMOUT. NO will cause the elements within the trim lines to be trimmed.
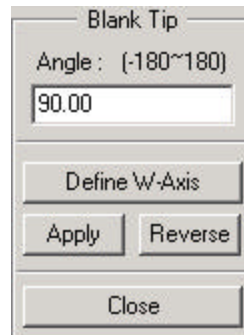

9.9   BLANK TIPPING

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

1. DYNAFORM-PC prompts:

   PLEASE INPUT ANGLE OR DEFINE W-AXIS

   • Input the angle and define W-axis in the window provided. Please see Figure
     9.39.

Figure 9.39 Blank Tip

2. Enter the value in the ANGLE field.

3. Select DEFINE W-AXIS, DYNAFORM-PC prompts:

   DEFINE A LOCAL COORDINATE SYSTEM
   PICK NODE FOR THE ORIGIN

   • DYNAFORM-PC prompts the user to create a local system. A thorough
     explanation of how to create a local coordinate system is covered in Chapter
     1 *General Discussion* section under "Local Coordinate System."

4. DYNAFORM-PC prompts via Pop-Up window:

   DO YOU ACCEPT THE DISPLAYED COORDINATE SYSTEM?

   ?   YES   Continue to Step 5.

NO   Returns to Step 3.

5. Select APPLY. The blank is changed and rotated about the W-axis by the input angle increment. The REVERSE function rotates the blank in the opposite direction. The user may select APPLY and REVERSE as many times as desired to fine-tune the tipping.

6. Select CLOSE to exit the function.
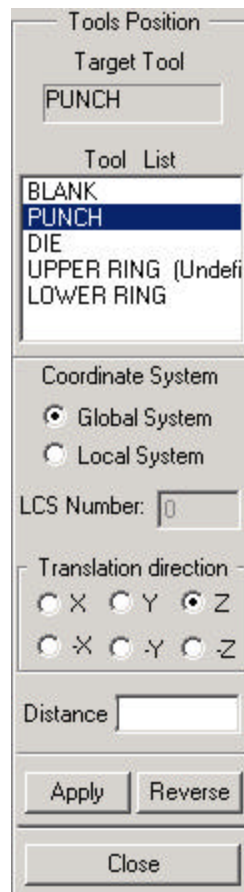
9.10   MOVE TOOLS

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

1. DYNAFORM-PC prompts:

    PLEASE SELECT TARGET TOOLS FIRST

    •   Select a tool from the list provided.

Figure 9.40 Tools Position

2. DYNAFORM-PC prompts:

   NOW PLEASE SELECT DIRECTION OR INPUT DISTANCE

   • TRANSLATION DIRECTION (toggle)
     If the Coordinate System is the Global System, the Translation Direction
     includes X, Y, Z, -X, -Y and -Z. If the Coordinate System is a Local System,
     the user should input the LCS number; the Translation Direction includes U,
     V, W, -U, -V and -W.
     Select the desired direction from the translation direction group.

   • INPUT DISTANCE
     Enter the value of the DISTANCE in the field provided.

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

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

5. Select CLOSE to exit the function.
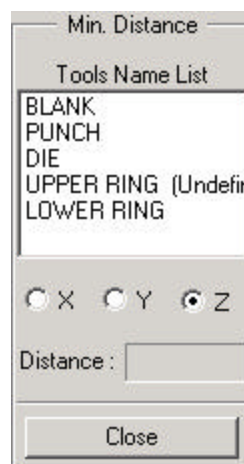
9.11   MINIMUM DISTANCE

This function measures the minimum distance between two selected tools.

1. DYNAFORM-PC prompts:

   SELECT THE REFERENCE TOOL

   • Select a reference tool from the displayed dialog window as in Figure 9.41.
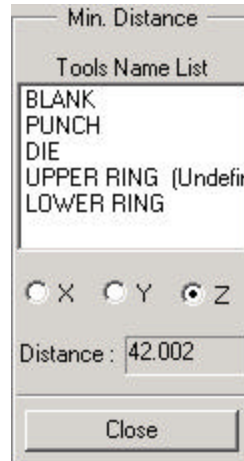

Figure 9.41 Min. Distance

2. Once the reference tool is selected, it will be highlighted. DYNAFORM-PC prompts:

SELECT ANOTHER TOOL FOR DISTANCE

3. Once the second tool is selected, a distance will be displayed in the DISTANCE field. Please see Figure 9.42.
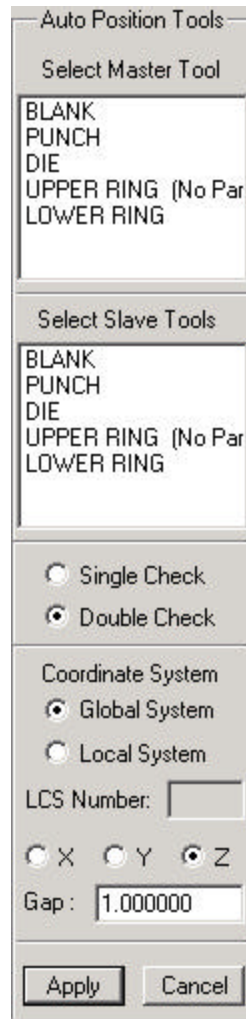

Figure 9.42 Calculated Distance

4. Select CLOSE to exit the function.

9.12   AUTO POSITION

This function automatically repositions the tools according to the user-defined draw type tool clearance and the stroke direction. The master tool is the stationary tool. Each slave tool moves based on the master tool. Please see Figure 9.43.

Figure 9.43 Auto Position Tools

1. Select the master tool.
2. Select slave tools.
   If the tool is not defined, DYNAFORM-PC prompts:
   TOOL XXX IS NOT DEFINED IN DATABASE
3. Check on Single Check or Double Check.
4. Select the reposition direction. The default direction is from the section 13.1 ANALYSIS SETUP.
5. Define the gap between the master tool and slave tools. The default value is from the blank Thickness in the section 13.1 ANALYSIS SETUP.
6. APPLY to complete the function and CANCEL to exit the function.

9.13   TAILOR WELDED

This function gives the user the ability to spot-weld two parts. The user needs to define a set of nodes on a part as the weld locations and another corresponding part to be welded.

---

1. DYNAFORM-PC prompts:

    SELECT TARGET PART FOR TAILOR WELD
    PICK AN ELEMENT TO IDENTIFY THE PART

2. Once the target part is selected, DYNAFORM-PC prompts:

   SELECT NODES TO WELD THE TARGET PART
    SELECT NODES

    •   A set of nodes from the source part needs to be selected to define the weld
        location.

3. After the desired node is selected, END SELECT will complete the operation. A
    set of rigid elements is generated to connect the two parts at the selected node
    locations.

9.14   BLANK MESH CALCULATOR

This function allows the user to mesh the blank with the element size based on the part's
tightest radius
.

1. DYNAFORM-PC prompts for the type of blank input:
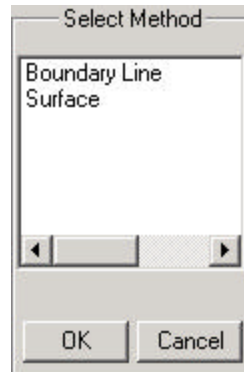
Figure 9.44 Select Method

•   BOUNDARY LINE
    The user selects a blank outline to be meshed.
•   SURFACE
The user selects a surface to be meshed.
    .
2. Once the boundary line or the surface is selected, DYNAFORM-PC prompts for
    the concerned radius:

Figure 9.45 Concerned Tool Radius

CONCERNED TOOL RADIUS
This radius should represent the smallest or tightest radius in the part.
DYNAFORM-PC calculates element size by the radius and refinement level
displayed in the lower window; ACCURATE and FAST. The ACCURATE
blank mesh is finer than the FAST blank mesh but requires more time to solve.

3. Select OK, DYNAFORM-PC prompts:

    DO YOU ACCEPT THE GENERATED MESH?

    ? YES   Accepts the generated mesh.
      NO   Comes back to Figure 9.45.
      REVIEW   Allows the user to review and remesh the blank.

9.15   TOOLS SUMMARY

This function displays statistical information about each tool.

1. DYNAFORM-PC prompts:

    SELECT A TOOL FOR SUMMARY

    •   Select a tool from the TOOLS SUMMARY window as in Figure 9.46.

Figure 9.46 Tools Summary

2. Once the desired tool is selected, a summary table for the selected tool will be displayed in the GRAPHICS WINDOW. An example is shown in Figure 9.47.

Figure 9.47

3. Select CLOSE to exit the summary table. Exit the function by selecting CLOSE from the TOOL SUMMARY window.

9.16   TOOL ANIMATE

This function is used to show movement of tools according to the defined motion. The tool motion is defined by the DEFINE MOTION CURVE function within DEFINE TOOL in Section 9.1.

1. DYNAFORM-PC prompts a window as Figure 9.48.

Figure 9.48 Enter Number of Frames

• Enter any frame number from 1 to 20.

2. Select ACCEPT to construct the desired number of frames and continue to the next step. The OPTION LIST ICONS are activated as Figure 9.49.

Figure 9.49 Select Frame Options

 ALL FRAMES (default)

 FIRST FRAME

 PREVIOUS FRAME

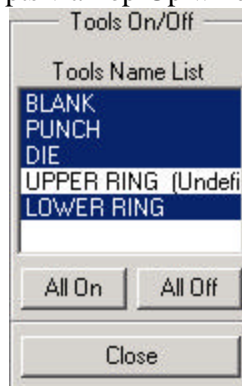 NEXT FRAME

 LAST FRAME

 END SELECT

3. END SELECT will exit the function.

9.17   TOOLS ON/OFF

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

1. DYNAFORM-PC prompts via Pop-Up window:

Figure 9.50 Tools On/Off

- The highlighted tool is shown in the current graphic area.

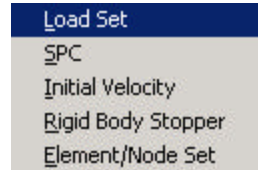ALL ON
This function turns all tools on.

ALL OFF
This function turns all tools off.

CHAPTER 10
BOUNDARY CONDITIONS

The functions of this menu create and verify constraints and loads and define initial velocities on finite element models. The functions of the BOUNDARY CONDITIONS menu are shown in Figure 10.1.
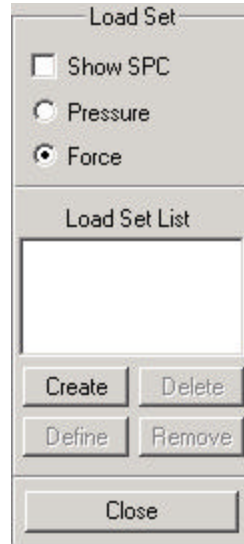

Figure 10.1 Boundary Conditions Menu

A detailed description is provided in the following sections.

10.1   LOAD SET

This function displays the LOAD SET window in the CONTROL WINDOW. Please see Figure 10.2.
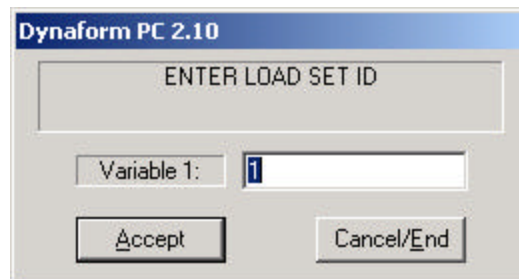

Figure 10.2 Load Set

10.1.1   SHOW SPC (toggle)

Toggles SPC (Single Point Constraint) on/off. Please refer to Section 10.2 SPC for additional information.
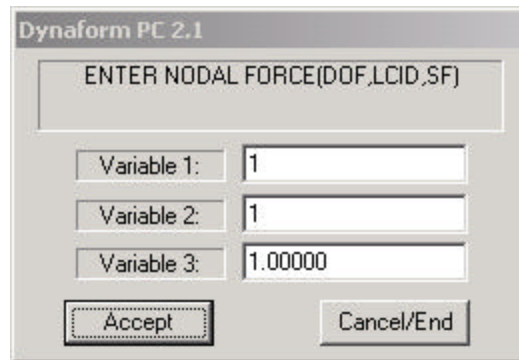
10.1.2   DEFINE FORCE ON SELECTED NODES (default)

1. Toggle FORCE on and select CREATE to create a load set.

2. Enter a load set number in the Pop-Up dialog window as Figure 10.3 and select ACCEPT.

Figure 10.3 Enter Load Set ID

3. Define nodal force in the following Pop-Up dialog window as Figure 10.4 and select ACCEPT.

Figure 10.4 Enter Nodal Force

DOF (Variable 1)
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 (Variable 2)
Load curve describes motion value versus the time. Please see Section 12.6.1 UTILITY/LOAD CURVE/CREATE CURVE.

SCALE FACTOR (Variable 3)
Load curve scale factor.

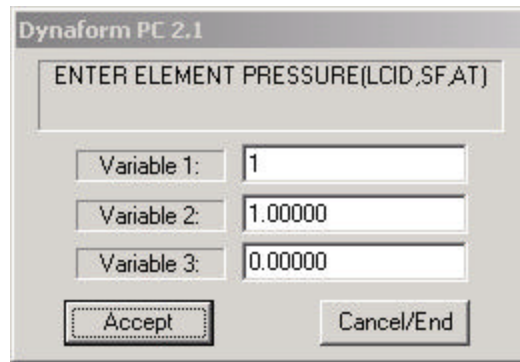4. Once nodal force is defined, DYNAFORM-PC prompts:

   SELECT NODES

   - END SELECT will return the user to Step 3.

5. Select CANCEL/END to complete and exit the function.

10.1.3 DEFINE PRESSURE ON SELECTED ELEMENTS

1. Toggle PRESSURE on and select CREATE to create a load set.

2. DYNAFORM-PC prompts the user to enter a load set number in the Pop-Up dialog window as Figure 10.3.

3. DYNAFORM-PC prompts a window as Figure 10.5.

Figure 10.5 Enter Element Pressure

LOAD CURVE ID (Variable 1)
Load curve describes motion value versus time. Please see Section 12.6.1 UTILITY/LOAD CURVE/CREATE CURVE.

SCALE FACTOR (Variable 2)
Load curve scale factor.

ACTIVE TIME (Variable 3)
Time imposed motion/loading is active.

4. Once element pressure is defined, select ACCEPT and DYNAFORM-PC prompts:

   SELECT ELEMENTS

   • END SELECT will return the user to Step 3.

5. Select CANCEL/END to complete and exit and the function.

10.1.4 DELETE

This function deletes a selected load set. It is used for both loading node force and element pressure.

10.1.5 DEFINE

This function adds more nodes or elements to a selected load set. It is used for both loading node force and element pressure.

10.1.6 REMOVE

This function removes forces/pressures from selected nodes/elements in a load set. It is used for both loading node force and element pressure.

10.2   SPC

This function allows the user to create and/or modify the SPC options. Please see Figure10.6.
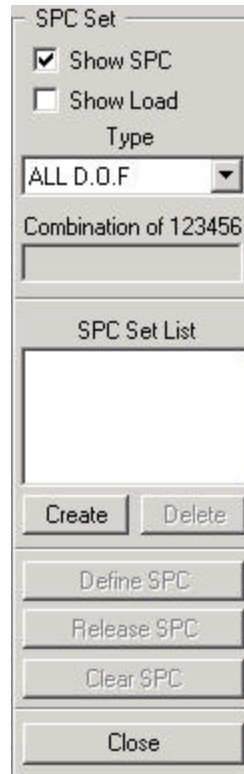
Figure 10.6 SPC Set

10.2.1 SHOW SPC (toggle)

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

10.2.2 SHOW FORCE (toggle)

Toggles defined force ON/OFF to show the current load force.

10.2.3 CREATE SPC SET

1. Select an SPC type. Please see Figure 10.7. There are seven types of degree of freedom in LS-DYNA:

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

6 - Z-Rotation

7 - Any Combination

- The user can create 63 combinations. The six basic types and seven of the most often used types are listed besides ANY COMBINATION.

2. If ANY COMBINATION is selected, a combination should be entered in COMBINATION OF 123456 field. Any combination including the thirteen types listed can be entered.
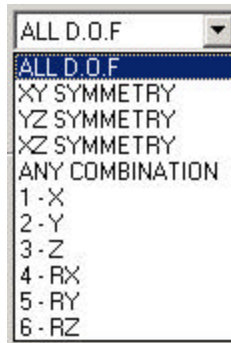
Figure 10.7 SPC Type

3. Select CREATE.

4. Enter the SPC SET ID Number in the next window.

5. Select the nodes to constrain and END SELECT will complete the function.

Once create a SPC, Delete, Define SPC, Release SPC and Clear SPC will be activated.

10.2.4 DELETE SPC SET

1. Select a SPC set from the list.

2. Select DELETE.

10.2.5 DEFINE SPC

1. Select a set from the list.

2. Select a type.

3. Select DEFINE SPC.

4. Select nodes. More nodes with the selected degree of freedom will be added to the highlighted SPC set.

10.2.6 RELEASE SPC

Removes SPC definition from the selected nodes in a SPC set.

10.2.7 CLEAR SPC

Removes all nodes from a selected SPC set.

10.3   INITIAL VELOCITY

This function allows the user to define and assign linear or angular velocities to selected nodes. It has several sub-functions. Please see Figure 10.8.

Figure 10.8 Initial Velocity

10.3.1 DEFINE INITIAL VELOCITY

Creates velocity set.

>   1. Enter the initial velocity components in the Pop-Up dialog window and select ACCEPT. Please see Figure 10.9.
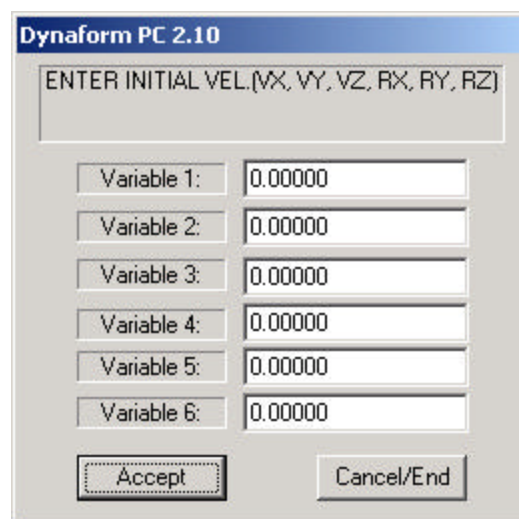

Figure 10.9 Enter Initial Velocity

2. Select nodes for the initial velocity location and END SELECT will return the user to Step 1.

3. Select CANCEL/END to complete and exit the function.

10.3.2 DELETE ALL

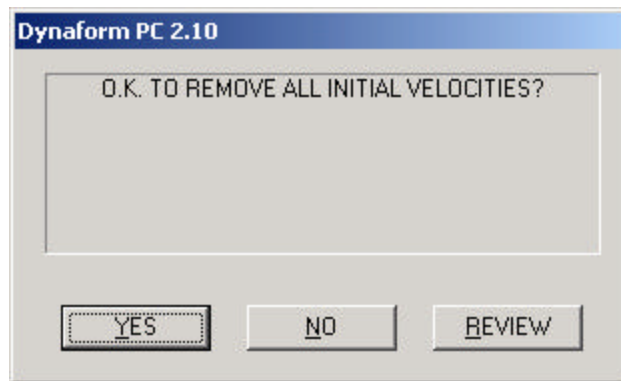Removes all nodes from the selected velocity set. A DYNAFORM-PC question window appears as Figure 10.10.

Figure 10.10 DYNAFORM-PC Question Window

10.3.3 REMOVE

Removes initial velocity from the selected nodes in the velocity set.

10.3.4 IDENTIFY

The initial velocity of selected node is listed.

1. DYNAFORM-PC prompts:

    SELECT NODE FOR INITIAL VELOCITY

2. If there is initial velocity specified at the node, it will be displayed in the PROMPT WINDOW. If not, a message will display in the PROMPT WINDOW.

3. END SELECT will exit the function.

10.3.5 SHOW

All nodes with an initial velocity are highlighted with an arrow. The CLEAR icon in the tool bar will remove the highlight.

10.4   RIGID BODY STOPPER

This function allows the user to define a rigid body stopper. A rigid body stopper provides a convenient way of controlling the movement of rigid tooling in metal forming. The function options are shown in Figure 10.11. By entering the minimal data in the Rigid Body Stopper Table, DYNAFORM-PC generates all the necessary data cards of a rigid body stopper for LS-DYNA analysis. For a detailed description, refer to the *LS-DYNA User's Manual.*
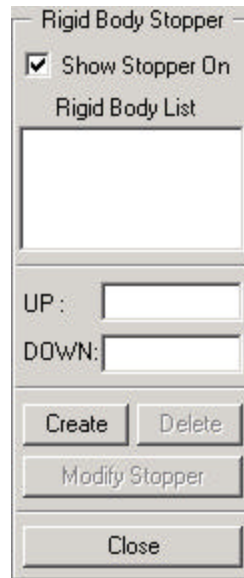

Figure 10.11 Rigid Body Stopper

10.4.1 SHOW STOPPER ON (toggle)

A stopper listed in the RIGID BODY STOPPER LIST will be shown in the CONTROL WINDOW.

10.4.2 CREATE

1. Enter a value in the UP and DOWN fields to define the distance from the upper stopper (or lower stopper) to the master part.

2. Select CREATE. The SELECT PART dialog window is displayed. Select a part as the master part.

3. Two square planes are displayed to show the upper and lower limit of the rigid body stopper. The edges of the planes are in white and the color of the grid is the same as the master part.

10.4.3 DELETE

1. Select a stopper in the list and select DELETE. A Pop-Up question window will display:

OK TO DELETE STOPPER FOR PART?

?   YES   Removes the selected stopper.
    NO   Cancels the function.
    REVIEW   Continue to Step 2.

2. If REVIEW is selected, pick YES or NO from the OPTION LIST ICONS.

10.4.4 MODIFY STOPPER

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

10.5 ELEMENT/NODE SET

This function defines element and node sets to facilitate the organization of output control data. Please see Figure 10.12.
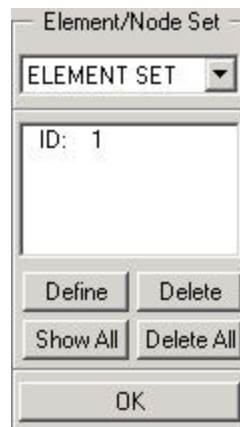

Figure 10.12 Element/Node Set

10.5.1 DEFINE

Creates new element/node sets from element/node selection.

1. Select ELEMENT SET or NODE SET from the pull down menu and then click DEFINE SET.

2. DYNAFORM-PC prompts:

   ENTER ELEMENT (NODE) SET ID

3. Once the element (node) set ID number is entered, the user is prompted to select elements (nodes). END SELECT will continue to the next step.

4. DYNAFORM-PC prompts:

   XXX ELEMENTS (NODES) INCLUDED IN SET XX

10.5.2 DELETE SET

Removes all or selected elements/nodes.

1. Select ELEMENT SET or NODE SET from the pull-down menu and select a ELEMENT SET or NODE SET, then DELETE.

2. DYNAFORM-PC prompts:

   OK TO DELETE ELEMENT (NODE) SET   XX?

3. Select YES and the selected element (node) set is deleted. NO comes back the function as Figure 10.12. REVIEW allows user to rotate the display window to decide whether delete the selected set.

10.5.3 SHOW ALL

Displays and highlights all or selected element/node sets and their ID numbers.

1. Select ELEMENT SET or NODE SET from the pull-down menu and then click SHOW SET.

2. DYNAFORM-PC prompts:

   SHOW ALL ELEMENT (NODE) SETS

3. Select YES to display all the element (node) sets and their ID numbers. NO allows the user to select which sets to display.

10.5.4 DELETE ALL

DYNAFORM-PC prompts:

OK TO DELETE ALL ELEMENT (NODE) SET   XX

Select YES and all element (node) sets are deleted. NO comes back the function as Figure 10.12. REVIEW allows user to rotate the display window to decide whether delete the all sets.

CHAPTER 11
SETUP

This menu contains various pre-process utilities that are toggle switch activated. Please see Figure 11.1.
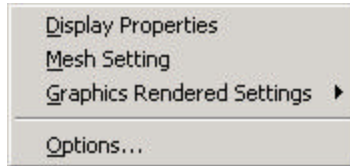
Figure 11.1 Set up Menu

These functions are described in the following sections.

11.1   DISPLAY PROPERTIES

This function is also accessible through the TOOL BAR. It has toggle switches that allow the user to select various ways to display information contained in the DYNAFORM-PC database. ACCEPT will execute the chosen options and CANCEL will exit the function without any changes occurring to the display screen. Please see Figure 11.2.
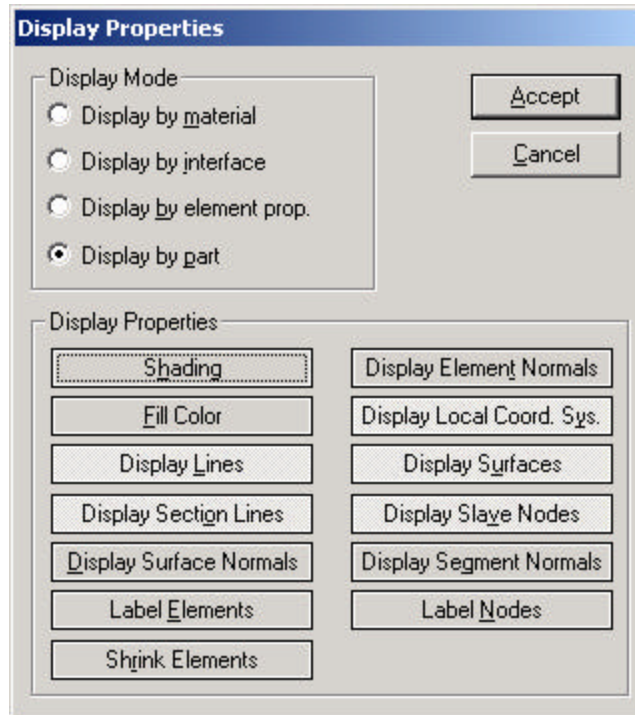

Figure 11.2 Display Property

11.1.1 DISPLAY MODE

The options in this menu can be toggled on/off.

> **DISPLAY BY MATERIAL**
> Displays the part in their respective material colors. If a part does not have a material property assigned to it, the part will become highlighted.
>
> **DISPLAY BY INTERFACE**
> Displays the master and slave segments for the parts that are turned on.
>
> **DISPLAY BY ELEMENT PROPERTY**
> Displays the parts in their respective element property colors. If a part does not have an element property assigned to it, the part will become highlighted.
>
> **DISPLAY BY PART (default)**
> Turns on all displayed parts in their respective colors in the database.

11.1.2 DISPLAY PROPERTIES

The user can select more than one at the same time to change the display properties.

SHADING
Displays realistic rendering of objects using light sources. DYNAFORM-PC represents elements as if a physical light source were present. Elements that are not directly exposed to the light source are appropriately "shaded" to emulate real life shading. This function is also accessible in the DISPLAY OPTION WINDOW.

FILL COLOR
Fills the displayed elements with their part color.

DISPLAY LINES (default)
Turns on all line data whose part has been turned on. This function is also accessible in the DISPLAY OPTION WINDOW.

DISPLAY SECTION LINES (default)
Turns on the section lines. The section lines are present within a surface (the dashed inner lines).

DISPLAY SURFACE NORMALS
Displays surface orientation as an arrow perpendicular to the surface's centroid.

LABEL ELEMENTS
Turns element labels on. All the displayed elements in the database will show their ID numbers.

SHRINK ELEMENTS
Plots the elements with their size reduced by 20 percent. This causes the model to appear as if there is a space between each element. This function is also accessible in the DISPLAY OPTION WINDOW.

DISPLAY ELEMENT NORMALS
Displays element orientations as an arrow perpendicular to the element's surface at the centroid.

DISPLAY LOCAL COORDINATE SYSTEM
Displays local coordinate systems that exist in the database.

DISPLAY SURFACES
Turns on surfaces whose part has been turned on. This function is also accessible in the DISPLAY OPTION WINDOW.

DISPLAY SLAVE NODES (default)
Turns on slave nodes in a database.

DISPLAY SEGMENT NORMALS

Displays the normal vectors for master and slave segments in a database.

LABEL NODES
Label all the nodes. All the displayed nodes in the database will show their node ID numbers.

11.2   MESH SETTING

Below is the mesh control setting window. This determines what type of mesh and how that mesh is created.
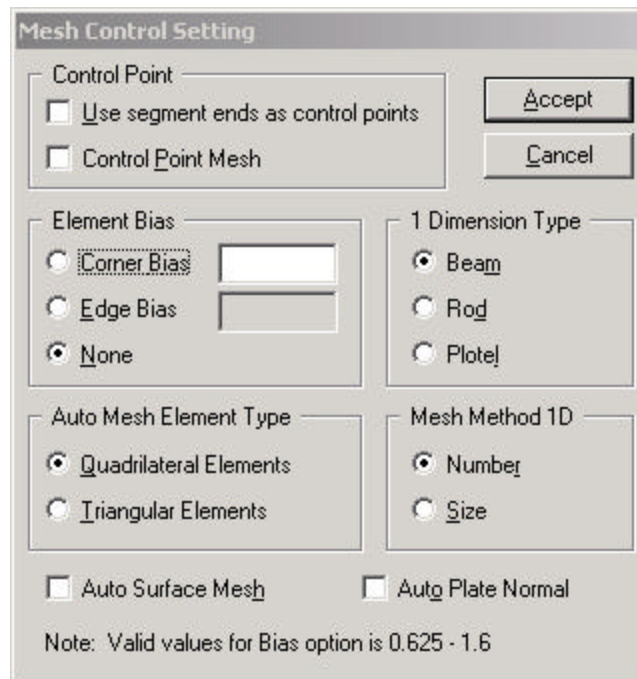

Figure 11.3 Mesh Control Point

11.2.1   CONTROL POINT

The options in this menu can be toggled on/off.

USE SEGMENT ENDS AS CONTROL POINTS (toggle)
Check it to automatically use the endpoints of line segments as control points. This function is used in conjunction with CONTROL POINT MESH. The program prompts the user to select control points on lines that were joined using the line segment option.

CONTROL POINT MESH (toggle)
Check it to pre-select points on a line that will be used for the desired node locations when executing the 2, 3, 4, 6, 8, 9 or 12 Line Mesh commands.

1. Once the user has selected the desired lines for a Line Mesh command, DYNAFORM-PC prompts via Pop-Up window for the user to enter the number of elements along each line.

2. After the user entering the desired mesh density, all selected lines will highlight and DYNAFORM-PC prompts:

   SELECT CONTROL POINTS ON LINE 1

3. The user can select control points on this line for control point locations.

   • Select NEXT LINE to proceed to the next line.

   • This prompt will repeat depending on the mesh style selection.

   • The beginning and end points of a line are the default control points.

4. END SELECT will end the control point selection. Once selected, continue the Line Mesh command.

11.2.2 ELEMENT BIAS

The options in the menu can be toggle on/off.

CORNER BIAS (toggle)
Sets the element bias factor (0.625 to 1.6) relative to a selected corner of a line to be modeled. This function multiplies each adjacent element created from the selected corner by the bias factor. A bias factor greater than 1.0 will generate proportionately larger elements from the selected corner and a bias factor less than 1.0 will generate proportionately smaller elements from the selected corner. This function may only be used with the 2, 3, 4, 6, 8, 9 or 12 Line Mesh commands.

1. Enter the bias factor (0.625 to 1.6).

2. Once a bias factor and a mesh density has been entered, DYNAFORM-PC prompts:

   CORNER BIAS OPTION IS ON, PICK BIASED CORNER

EDGE BIAS (toggle)
Sets the element bias factor (0.625 to 1.6) relative to a selected edge of a line to be modeled. This function multiplies each adjacent element created from the selected edge by the bias factor. A bias factor greater than 1.0 will generate proportionately larger elements from the selected edge and a bias factor less than 1.0 will generate proportionately smaller elements from the selected edge. This function may only be used with the 2, 3, 4, 6, 8, 9 or 12 Line Mesh commands.

1. Enter the bias factor (0.625 to 1.6).

2. Once a bias factor and a mesh density has been entered, DYNAFORM-PC prompts:

    EDGE BIAS OPTION IS ON, PICK BIASED EDGE

NONE (toggle)
Turns off the above two buttons.

11.2.3   1 DIMENSION TYPE

The options decides the element type that user does line mesh. The default type is BEAM.

BEAM
The 1-dimension type is the beam element when the user does line mesh.

ROD
The 1-dimension type is the rod element when the user does line mesh.

PLOTEL
The 1-dimension type is the plotel element when the user does line mesh.

11.2.4 AUTO MESH ELEMENT TYPE

Toggle the 2D element types prior to utilizing the 2, 3, 4 Line Mesh or Surface Mesh. Quadrilateral Elements is the default type.

11.2.5 MESH METHOD 1D

The options in this dialog window are used to control the number of 1D mesh.

NUMBER
Preset the element number that will be followed when executing the 1D mesh. The mesh will be automatically generated with the input number.

SIZE
Preset a maximum element size that will be followed when executing the 1D mesh. The mesh will be automatically generated with the desired element size.

11.2.6 AUTO SURFACE MESH (toggle)

If toggle on the switch, when select SURFACE MESH, DYNAFORM-PC will mesh all displayed surface automatically.

11.2.7   AUTO PLATE NORMAL (toggle)

If toggle on the switch, all plate elements normal will be consistent when mesh.

11.3   GRAPHICS RENDERED SETTINGS

The functions in this submenu are used to adjust the display of the items on the screen. Please see Figure 11.4.
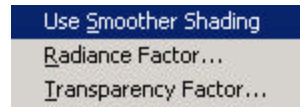
Figure 11.4 Graphics Rendered Setting

### 11.3.1 USE SMOOTHER SHADING (toggle)

This function can be toggled on/off. DYNAFORM-PC uses two methods for object shading: flat and smoothed. Flat shading method shades each polygon based on the intensity of the light source that reaches the polygon. Smoothed shading method averages the intensity of the light over a series of the polygons making the image appear less flat or angled. An example is given in Figure 11.5.
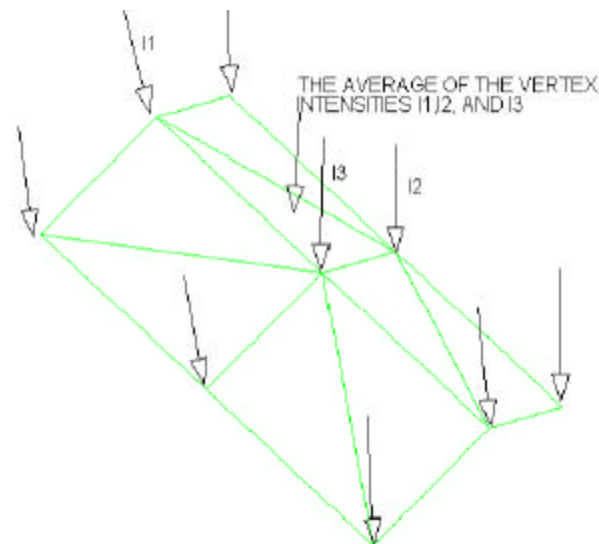
Figure 11.5 Smoother Shading

11.3.2 RADIANCE FACTOR

The function prompts the user via Pop-Up window as Figure 11.6 to enter default radiance intensity factor.
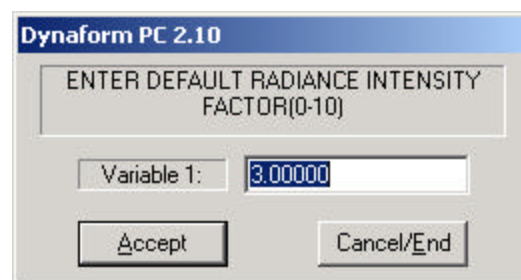
Figure 11.6 Radiance Factor

Radiance Factor controls the radiance intensity of the light source.

11.3.3 TRANSPARENCY FACTOR

A part is considered transparent when another part that would normally be obscured by the transparent part is still visible. The transparency option can be activated through the MODIFY and NEW functions in the DISPLAY OPTION WINDOW (refer to Section 1.5 of Chapter 1 General Discussion).

The program provides transparency through screening. Screening is a pattern-based polygon filling method that simulates transparency. This function adjusts the degree of transparency by prompting the user to enter default transparency factor via Pop-Up window.
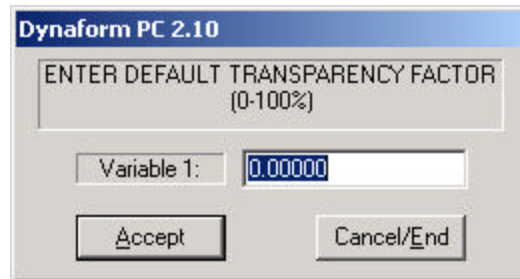
Figure 11.7 Transparency Factor

11.4   OPTIONS

This function displays additional options the user may change to the display, auto protect and the startup of DYNAFORM-PC. Please see Figure 11.8.
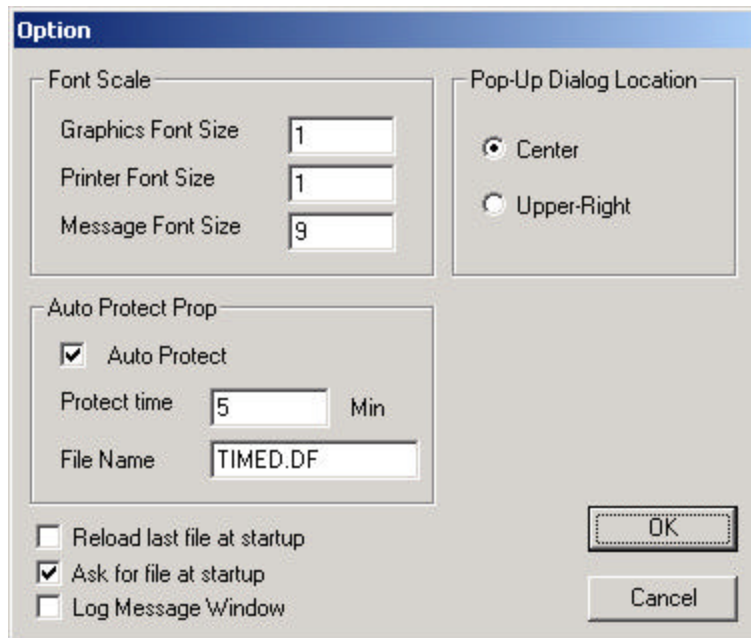
Figure 11.8 Option

11.4.1 FONT SCALE

Independently adjusts the size of the font on-screen and the size of the font sent to the printer.

- GRAPHICS FONT SIZE adjusts the font size in the GRAPHICS DISPLAY WINDOW.

- PRINTER FONT SIZE adjusts the size of the font sent to the printer.

- MESSAGE FONT SIZE adjusts the size of the font in the PROMPT WINDOW.

11.4.2 AUTO PROTECT PROPERTY

DYNAFORM-PC can automatically save a database file at regular intervals to avoid losing data.

- AUTO PROTECT (toggle) enables/disables the function described above.

- PROTECT TIME specifies the interval between this time and last time to save.

- FILE NAME gives the backup filename.

11.4.3 POP-UP DIALOG LOCATION

This function allows the user to select the location of the Pop-Up dialog window when it appears on the screen. There are two options for the location:

- CENTER of the screen.

- UPPER-RIGHT corner of the screen.

11.4.4 RELOAD LAST FILE AT STARTUP

If this box is checked, DYNAFORM-PC will reload the current modified database file at the next startup of the program.

11.4.5 ASK FOR FILE AT STARTUP

When it is checked, DYNAFORM-PC will run the OPEN function in the FILE menu automatically at the next startup of the program.

Note: If RELOAD LAST FILE AT STARTUP is also checked in addition to this function, DYNAFORM-PC will not load or ask for a file at the next startup.

11.4.6 LOG MESSAGE WINDOW

Enables/disables the output of a log containing all messages posted to the PROMPT WINDOW during a file session. If it is checked, the program maintains a log of all messages in a file named SESSION.LOG. The file is created in the directory that contains the database file currently being modified.

UTILITY

The functions in the UTILITY menu make up DYNAFORM-PC's "tool kit." Many of these functions are also found in other chapters. The options are shown in Figure 12.1.
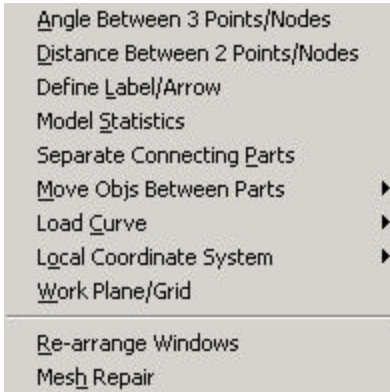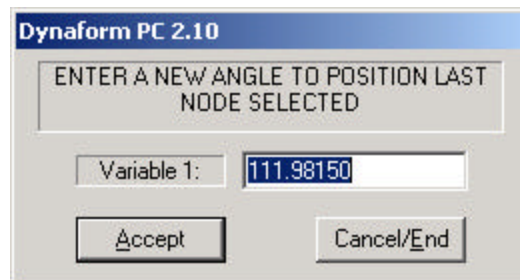

Figure 12.1 Utility Menu

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

12.1   ANGLE BETWEEN 3 POINTS/NODES

This function measures the angle between two selected vectors.

1. DYNAFORM-PC prompts:

    SELECT NODE FOR VERTEX

2. Once the node is selected, DYNAFORM-PC prompts:

    SELECT 2 NODES

3. After the two nodes are selected, the angle between the two selected vectors is displayed. The measurements of the angle are also displayed in the PROMPT WINDOW. DYNAFORM-PC returns the user to Step 1.

4. If the user wants to adjust the angle, select ADJUST VALUE in the OPTION LIST ICONS prior to selecting a next point for the vertex. DYNAFORM-PC prompts via Pop-Up window as Figure 12.2.

Dynaform PC 2.10

ENTER A NEW ANGLE TO POSITION LAST
NODE SELECTED

Variable 1:   111.98150

Accept   Cancel/End

Figure 12.2 Enter A New Angle

5. Enter a value in the dialog window and select ACCEPT. DYNAFORM-PC moves the location of the third node/point and changes the shape of the elements around it.

6. END SELECT will exit the function.

12.2   DISTANCE BETWEEN 2 POINTS/NODES

This function calculates the distance between two points, two nodes or a node and a point.

1. DYNAFORM-PC prompts:

    SELECT POINT

2. Select two points. DYNAFORM-PC displays the distance between the two nodes/points and its three components in the PROMPT WINDOW.

3.   The user may change the distance (the location of the second node/point along the line from the first to the second). Select ADJUST MAGNITUDE and DYNAFORM-PC prompts the user to enter a value via Pop-Up window as Figure 12.3.
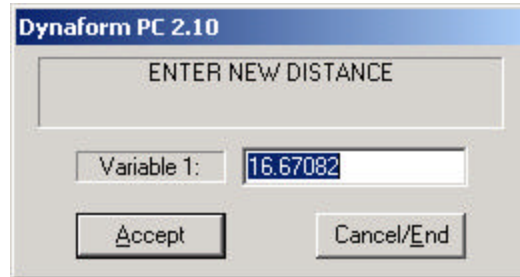
Figure 12.3 Enter New Distance

•   Enter the new value and select ACCEPT.

4.   The user may change the location of the second node/point. Select CHANGE DIRECTIONAL DIST. XYZ. DYNAFORM-PC prompts the user to enter new values via Pop-Up window as Figure 12.4.
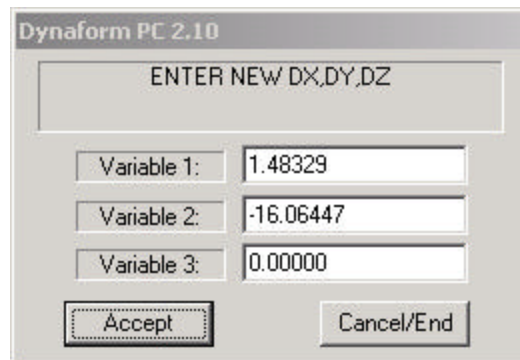
Figure 12.4 Enter New DX, DY, DZ

•   Enter the new values and select ACCEPT.

5.   END SELECT will exit the function.

12.3   DEFINE LABEL/ARROW

This function allows the user to draw arrows about a specific image and create a label at any location in the GRAPHICS DISPLAY WINDOW. There are two label types as Figure 12.5.
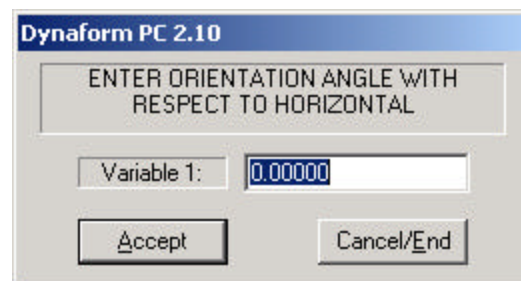
Figure 12.5 Select Label Type

12.3.1 TEXT LABEL

1. Once TEXT LABEL is selected, DYNAFORM-PC prompts via Pop-Up window as Figure 12.6.
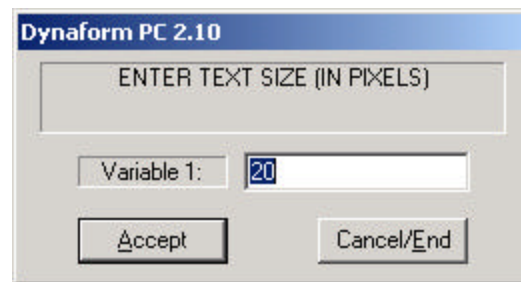
Figure 12.6 Enter Text Size

2. After the TEXT SIZE is entered, DYNAFORM-PC prompts via Pop-Up window as Figure 12.7.

Figure 12.7 Enter Orientation Angle with Respect to Horizontal

3. Enter the orientation angle and DYNAFORM-PC prompts a color selection window for a text color.

4. Select the label color and the user is prompted to enter the label name.

5. After the label name is entered, DYNAFORM-PC prompts:

SELECT X, Y LOCATION FOR LABEL

- Once a location is selected, DYNAFORM-PC returns to Step 4.

6. Select END SELECT and then CANCEL/END from Figure 12.5 when it redisplays to exit the function.

12.3.2 ARROW

This function allows the user to draw an arrow in the GRAPHICS WINDOW. It is particularly useful with the TEXT LABEL to point the description to a particular part of the model.

1. Once arrow is selected, DYNAFORM-PC prompts via Pop-Up window as Figure 12.8.
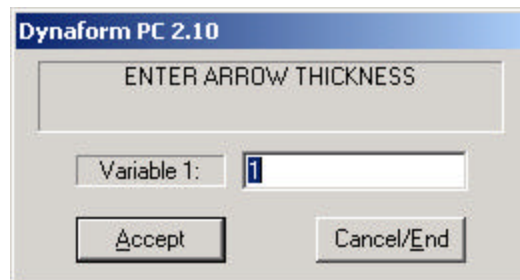
Figure 12.8 Enter Arrow Thickness

2. Enter the arrow thickness and DYNAFORM-PC prompts a color selection window for an arrow color.

3. After a color is selected, DYNAFORM-PC prompts:

SELECT FIRST X, Y LOCATION FOR TAIL OF POINTER

Select a location and DYNAFORM-PC prompts:

SELECT SECOND X, Y LOCATION FOR HEAD OF POINTER

- Once a location is selected, DYNAFORM-PC repeats Step 3.

4. Select END SELECT and then CANCEL/END from Figure 12.5 when it redisplays to exit the function.

12.4   MODEL STATISTICS

This function displays statistics about the DYNAFORM-PC database regarding geometry, models, materials and interfaces.

    1.   Once selected, DYNAFORM-PC displays the Model Statistics window as Figure 12.9.

Figure 12.9 Model Statistics

    2.   Select OK.

12.5 SEPARATE CONNECTING PARTS

This function allows the user to quickly separate parts having common nodes. Once the parts are separated, each common node becomes several nodes, one for each part, at the same location. If the function is completed, DYNAFORM-PC prompts:

    THE SELECTED PARTS HAVE BEEN SEPARATED.

12.6   MOVE OBJECT BETWEEN PARTS

This function allows the user to move entities from one part in order to organize the model. It has three sub-functions shown in Figure 12.10.

Move Elements
Move Lines
Move Surfaces
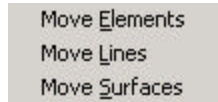
Figure 12.10 Move Object between Parts Menu

12.6.1 MOVE ELEMENTS

Move existing elements from one part to another. The target part does not have to be current or on.

1. DYNAFORM-PC prompts:

    SELECT TARGET PART
    PICK AN ELEMENT TO IDENTIFY THE PART

2. Once the target part is selected, the program prompts:

    PART XXXX IS SELECTED
    ELEMENTS WILL BE ADDED TO PART XXXX
    (SELECT ELEMENTS)

3. DYNAFORM-PC will move the elements into the target part when END SELECT is selected.

12.6.2 MOVE LINES

Move existing lines from one part to another. The target part does not have to be current or on.

1. DYNAFORM-PC prompts:

    SELECT TARGET PART
    PICK AN ELEMENT TO IDENTIFY THE PART

2. Once the target part is selected, DYNAFORM-PC prompts:

    PART XXXX IS SELECTED
    LINES WILL BE ADDED TO PART XXXX
    SELECT LINE(S)

3. END SELECT will move the line(s) into the target part.

12.6.3 MOVE SURFACES

Move existing surfaces from one part to another. The target part does not have to be current or on.

1. DYNAFORM-PC prompts:

   SELECT TARGET PART
   PICK AN ELEMENT TO IDENTIFY THE PART

2. Once the target part is selected, the program prompts:

   PART XXXX IS SELECTED
   SURFACES WILL BE ADDED TO PART XXXX
   SELECT SURFACES

3. Select surface(s) with these options and END SELECT will move the surface(s) into the target part.

12.7   LOAD CURVE

The sub-functions in this menu are used to define and modify load curves. Please see Figure 12.11.

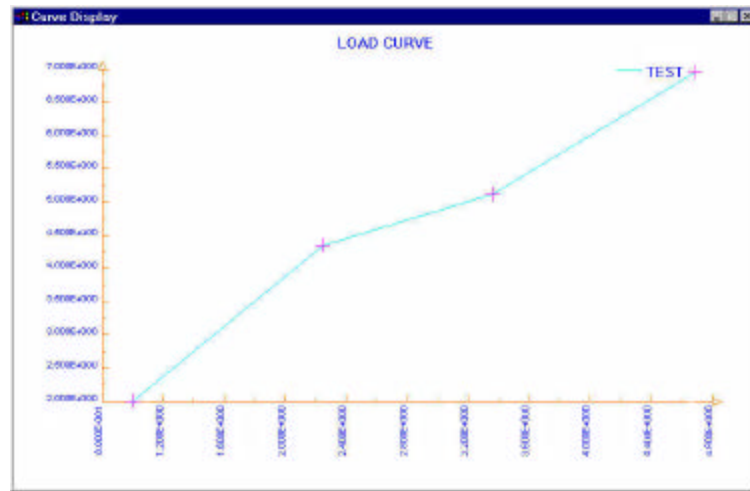**Figure 12.11 Load Curve Menu**

12.7.1 CREATE CURVE

This function allows the user to create load curves. Please see Figure 12.12.

Figure 12.12 Create Load Curve

- The curve ID is automatically assigned by DYNAFORM-PC. Enter the name of the curve in the NAME field.

- Enter a pair of numbers to define a point on the curve. They will appear in the POINT LIST. Select ADD to define another point.

- Select a pair of numbers from POINT LIST**.** Change the values and select MODIFY to modify a point.

- Select a pair of numbers from the POINT LIST. Select DELETE to remove the point from the curve.

- Select CREATE to generate the curve. A curve window displays and covers the DISPLAY AREA as Figure 12.13.
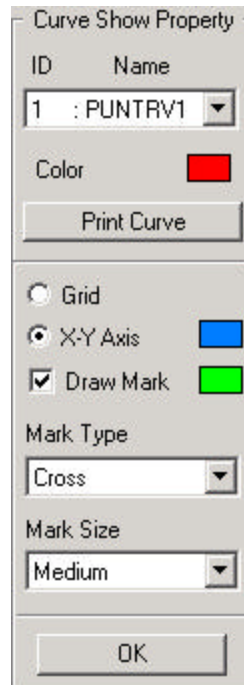
Figure 12.13 Curve Window

12.7.2 SHOW CURVES

This command lists all available load curves in the SELECT LOAD CURVE window as Figure 12.14.
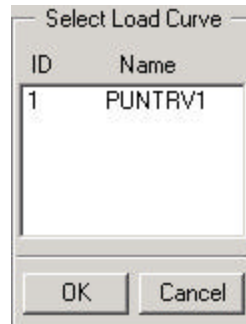
Figure 12.14 Select Load Curve

- Select a curve from the list and the CURVE SHOW PROPERTY window displays as Figure 12.15.

Figure 12.15 Curve Show Property

- Select COLOR to change color of the curve.

- Select PRINT CURVE to print the Curve Display window.

- Select GRID to display grid instead of X and Y-axes.

- Select X-Y AXIS to display X and Y axes instead of a grid. Click the color block on the right to change the axes' color.

- Select DRAW MARK to display point marks on the curve. Click the color block on the right to change the marks' color.

- Select a MARK TYPE from four options: Cross, Circle, Up-Triangle and Down Triangle.

- Select a MARK SIZE from five options: Very Small, Small, Medium, Large and Very Large.

12.7.3 MODIFY CURVE

Allows the user to modify points on a selected curve.

1. Once the user selects this function, the SELECT LOAD CURVE window displays (Figure 12.14).

2. After the curve is selected, the MODIFY LOAD CURVE window displays in the
    CONTROL WINDOW as Figure 12.16.

Figure 12.16 Modify Load Curve

- The CURVE ID field displays the curve ID number.

- The CURVE NAME field displays the name of the curve.

- There are three key OPERATIONS. Select the pull-down to select the type of
 operation and click APPLY. Please see Figure 12.17.

Figure 12.17 Modify Load Curve Operations

- ADD point(s): Select any desired location(s) in the Curve Display window by
  mouse pick or use KEY IN X-Y to enter the X, Y values in the fields of
  ENTER DATA FOR and then click the ENTER DATA button.

- DELETE point(s): Select the desired point(s) on the curve by mouse pick.
  The point(s) will be deleted automatically.

- MODIFY point(s): Use NEGATE to select point(s) to flip about the X axis or use SCALE to enter the scale factors in the fields of ENTER DATA FOR and click on ENTER DATA. Select point(s) that will be moved to the scaled coordinates.

- QUERY VALUE: The selected point values are displayed in the prompt window.
- UNDO will reject the last change made by the user.

- EXIT will end the function and close the MODIFY LOAD CURVE window. If needed, a Pop-Up question window will appear to ask the user to save the changed curve or not.

12.7.4 DELETE CURVE

Allows the user to delete any load curve. A DELETE LOAD CURVE window will appear in the CONTROL WINDOW as Figure 12.18.
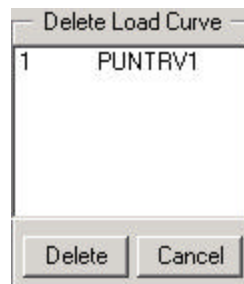

Figure 12.18 Delete Load Curve

- Select any curve(s) in the window.

- Select DELETE and a Pop-Up question window will display. Select YES to delete the selected curve(s) or NO to return to the Delete Load Curve window without deleting any curve(s).

- Select CANCEL to delete nothing and exit the function.

12.7.5 READ CURVE FILE

Imports existing curve data into the DYNAFORM-PC database file. The OPEN LOAD CURVE FILE window is displayed in Figure 12.19. DYNAFORM-PC lists all the curve data files with the suffix .cur. The user can select any curve data file. An example of a curve file format can be found in Appendix C.

12.7.6 SAVE CURVE FILE

Saves selected curve data as a curve data file.
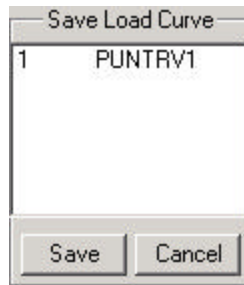
Figure 12.20 Save Load Curve

- Select a curve from the SAVE LOAD CURVE window as in Figure 12.20 and click SAVE.

- The SAVE LOAD CURVE FILE window is displayed. Input a name in the FILE NAME field and select a directory to save the curve in.

12.8   LOCAL COORDINATE SYSTEM

The functions in this menu are used to create and modify local coordinate systems. The functions are given as Figure 12.22.
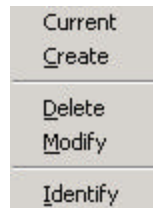
Figure 12.22 Local Coordinate System Menu

12.8.1 CURRENT

Set a local coordinate system as the current system.

1. DYNAFORM-PC prompts:

   SELECT LOCAL COORDINATE SYSTEM

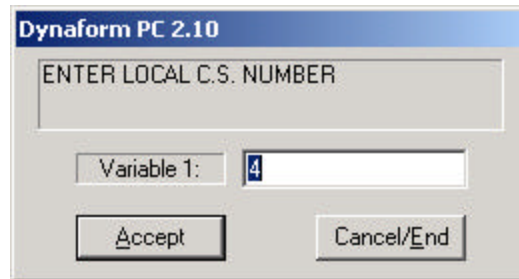2. After the local coordinate system is selected, DYNAFORM-PC prompts:

   LOCAL COORDINATE SYSTEM XXXX IS USED

   The selected local coordinate system is the current system and the function is completed.

12.8.2 CREATE

This function creates a coordinate system.

1. DYNAFORM-PC prompts via Pop-Up window as Figure 12.23.

Figure 12.23 Enter L.C.S. Number

- Input a positive integer. If the number is already used, DYNAFORM-PC will prompt the user to select again. The default number is next to the maximum of the already created system numbers.

2. After the number is entered, DYNAFORM-PC prompts the user to select a coordinate system type via Pop-Up window as Figure 12.24.
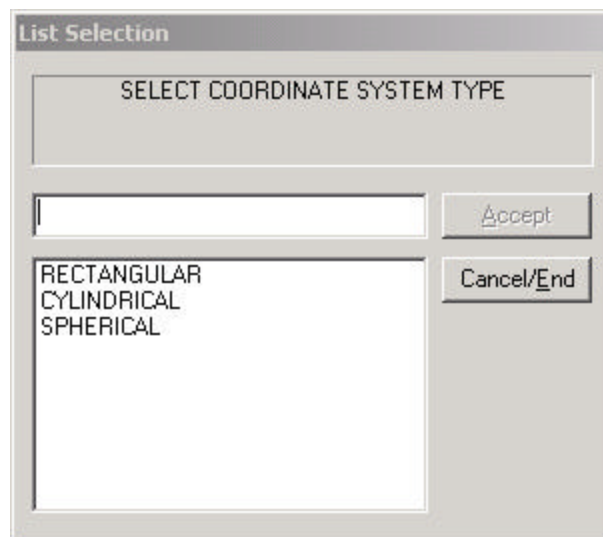
Figure 12.24 Select Coordinate Systme Type

3. Once a coordinate system type is selected, DYNAFORM-PC prompts:

DEFINE A LOCAL COORDINATE SYSTEM
PICK A POINT (NODE) FOR THE ORIGIN

At this time, the user can also select GLOBAL and Last Coordinate System to define a new local coordinate system.

4. After the origin is defined, DYNAFORM-PC prompts:

    PICK THE NEXT POINT (NODE)

    At this time, the user can also select W along X (Y or Z) -axis to create the coordinate system. The created system axes is parallel with the global axes according to the right hand rule.

    •   Refer to Section 1.10 for detail about defining the local coordinate system.

5. After the coordinate system is defined, the program will prompt the user to accept/reject the displayed coordinate system. Once the coordinate system is accepted, the coordinate system is labelled at the origin of the local coordinate system.

6. Step 1 will repeat, select CANCEL/END to exit the function.
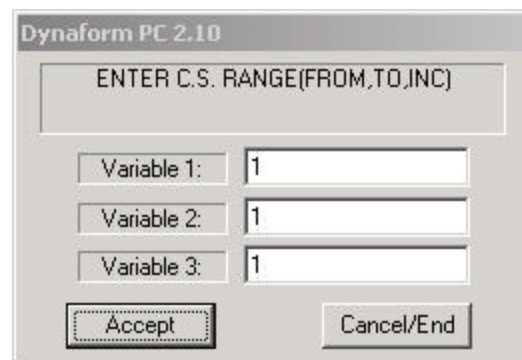
12.8.3 DELETE

Deletes a local coordinate system from the database.

1. DYNAFORM-PC prompts:

    SELECT COORDINATE SYSTEM

    The user can select system(s) by Cursor, Key In Coordinate System and Drag Window. If by Key In Coordinate System, DYNAFORM-PC will prompt:

Figure 12.25 Enter C.S. Range

    Variable 1 is the start number of the range, Variable 2 is the the end number of the range, and Variable 3 is the gap between the two C.S..

2. Once the coordinate system(s) are selected, END SELECT will delete the local coordinate system(s) and exit the function.

12.8.4 MODIFY

Modifies the definition of an existing coordinate system. The number or ID may not be modified in this function.

1. DYNAFORM-PC prompts:

   SELECT LOCAL COORDINATE SYSTEM

2. After the local coordinate system is selected, DYNAFORM-PC will prompt via Pop-Up window:

   SELECT COORDINATE SYSTEM TYPE

3. DYNAFORM-PC prompts:

   DEFINE A LOCAL COORDINATE SYSTEM
   PICK A POINT FOR THE ORIGIN

4. Define a local coordinate system as described in Section 12.7.2 CREATE.

   - DYNAFORM-PC prompts the user to create a local coordinate system. A thorough explanation of how to create a local coordinate system is covered in Chapter 1 *General Discussion* section under "Local Coordinate System."

5. Once the desired coordinate system is created, it will be displayed on the screen and the prompt will read via Pop-Up window:

   DO YOU ACCEPT THE DISPLAYED COORDINATE SYSTEM?

   ? YES   Returns to Step 1.
       NO   Returns to Step 3.

6. END SELECT will exit the function.

12.8.5 IDENTIFY

Identifies type and origin of a coordinate system.

1. DYNAFORM-PC prompts:

   SELECT LOCAL COORDINATE SYSTEM

2. Once selected, the coordinate system number is automatically displayed, along with the type (RECT, SPHR or CYL) and global location of the system origin.

3. END SELECT will exit the function.

12.9   WORK PLANE/GRID

This function allows the user to create and modify a work plane. A work plane is a reference plane displayed as white dots. The functions are given as Figure 12.26.

Figure 12.26 Work Plane Edit

1. Input data in the fields of NAME, GRID DIVISION (number of points along the length of a work plane), and LENGTH (size of a work plane).

2. Select COORDINATE SYSTEM to define a coordinate system at the desired position.

3. Select a plane location for the coordinate system from the pull down menu as Figure 12.27.

Figure 12.27 Plane Location Options

4. Once Steps 1 – 3 are completed, select CREATE to create the work plane.

5. After selecting a plane from the DEFINED WORK PLANE list, the user has three options to modify the work plane: DELETE, CURRENT and MODIFY.

   - Select DELETE to delete the selected work plane.

   - CURRENT turns the selected work plane on.

   - Change the numbers in the field of GRID DIVISION and/or LENGTH of the selected plane. After this is completed, select MODIFY to modify GRID DIVISION and/or LENGTH.

6. The DISPLAY WORK PLANE toggle switch will toggle the work plane on/off.

12.10   RE-ARRANGE WINDOWS

This function moves the CONTROL WINDOW to the opposite side of the DISPLAY AREA.

12.11   MESH REPAIR

This window provides a convenient way of repairing mesh. It collects many functions from NODE, ELEMENT and MODEL CHECK etc. to avoid switching between the menus.
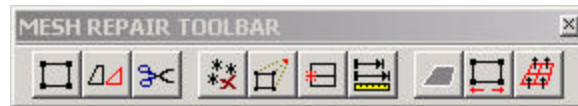

Figure 12.28 Mesh Repair Toolbar

CREATE ELEMENTS
Please refer to the section 6.8 CREATE ELEMENTS to get detailed information.

MODIFY ELEMENTS
Please refer to the section 6.14 MODIFY ELEMENTS to get detailed information.

DELETE ELEMETS
Please refer to the section 6.15 DELETE LEMENTS to get detailed information.

DELETE ALL FREE NODES
Please refer to the section 1.4 TOOL BAR to get detailed information.

MOVE NODES
Please refer to the section 7.4 MOVE NODES to get detailed information.

CHECK COINCIDENT NODES
Please refer to the section 7.10 CHECK COINCIDENT NODES to get detailed information.

ADJUST MAGTITUDE
Please refer to the section 12.2 DISTANCE BETWEEN 2 POINTS/NODES to get detailed information.

DISPLAY MODEL BOUNDARY<off>
The default of the switch is off. Please refer to the section 8.1 to get detailed information.

CHECK ELEMENT SIZE
Please refer to the section 8.4 to get detailed information.

AUTO PLATE NORMAL
Please refer to the section 8.2 to get detailed information.

CHAPTER 13
ANALYSIS

The functions in this menu allow the user to run a complete analysis or generate an input file. The following options are available as Figure 13.1.

Analysis Setup...
Run LS-DYNA...
Quick Setup
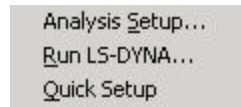
Figure 13.1 Analysis Menu

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

13.1   ANALYSIS SETUP

This function is used to define the forming parameters of the output. The options are shown in Figure 13.2.
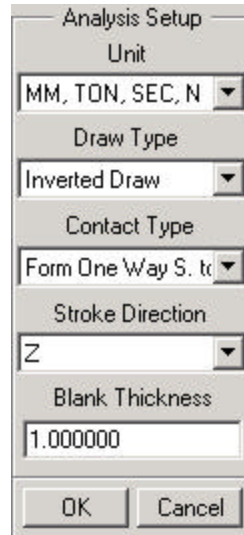

Figure 13.2 Analysis Setup
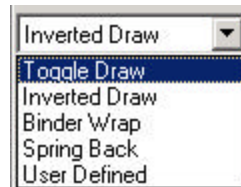
13.1.1 UNIT

Select the appropriate unit system from the list provided as Figure 13.3.

Figure 13.3 Unit List

13.1.2 DRAW TYPE

Select the pull down menu to display the menu as Figure 13.4.

Figure 13.4 Draw Type

- Select a type from the five options briefly described below.

TOGGLE DRAW	The punch is above the blank.
INVERTED DRAW	The punch is under the blank.
BINDER WRAP	Only the upper ring and lower ring. No punch or die.
SPRING BACK	Reads in DYNAIN file (for trimming) or restart file d3dump01. Run LS-DYNA in Implicit Mode using one of these files.
USER DEFINED	Draw type defined by the user.

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

13.1.3 CONTACT TYPE

There are several contact interface types. The user can select the interface type for all tool/blank interfaces. Select the pull down menu below the CONTACT TYPE field and select from one of the nine options as Figure 13.5.
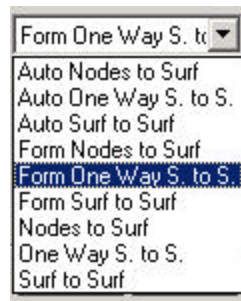
Figure 13.5 Contact Type List

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

13.1.5 STROKE DIRECTION

Select the pull down menu below the STROKE DIRECTION field that includes three options: X, Y and Z. Select a direction from these options.

13.1.6 BLANK THICKNESS

The user can set a clearance between the tool pieces and the blank for the AUTO POSITION command. The default is 1.0.

13.2   RUN LS-DYNA

The ANALYSIS PARAMETERS dialog window appears in the CONTROL WINDOW area:
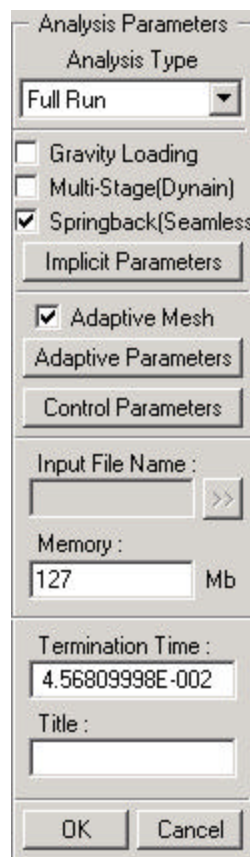
Figure 13.6 Analysis Parameters

13.2.1 ANALYSIS TYPE

There are two types of analysis options the user may choose after the arrow button is selected:

1. LS-DYNA INPUT FILE
   This function outputs an LS-DYNA input file for future analysis.

2. FULL RUN
   This function outputs an LS-DYNA input file and submits it for LS-DYNA analysis, which will run in the background. A DOS window is opened to display the status of analysis.

13.2.2 GRAVITY LOADING (toggle)

This function switches gravity loading on/off.

13.2.3 MULTI-STAGE (Dynain) (toggle)

This function automatically writes out the INTERFACE_SPRINGBACK_DYNA3D card to the input deck. Therefore, LS-DYNA creates a DYNAIN file when it completes the running job. This option is required for the Spring Back analysis with trimming operation.

13.2.4 SPRINGBACK (Seamless) (toggle)

This function allows the user to write out the *INTERFACE_SPRINGBACK_SEAMLESS card to the input deck. Therefore, LS-DYNA will automatically switch to implicit mode for Springback analysis after it completes the Draw-Die Simulation.

The IMPLICIT PARAMETERS button will be activated once the Springback option is checked. This button will display a list of CONTROL_IMPLICIT cards for the implicit solver. Please see Figure 13.7.
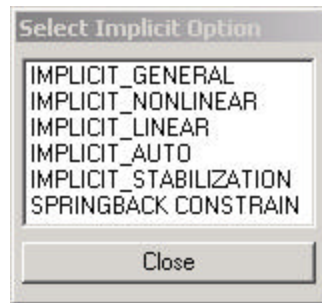
Figure 13.7 Implicit Option List

DYNAFORM-PC assigns all the default values for the necessary CONTROL_IMPLICIT cards for the Springback simulation. If the user wishes to customize any control card, select a card name in the Select Implicit Option window. A Pop-Up window will display to allow the user to modify any data in the card. For example, the IMPLICIT_GENERAL card is displayed in the following window as Figure 13.8.
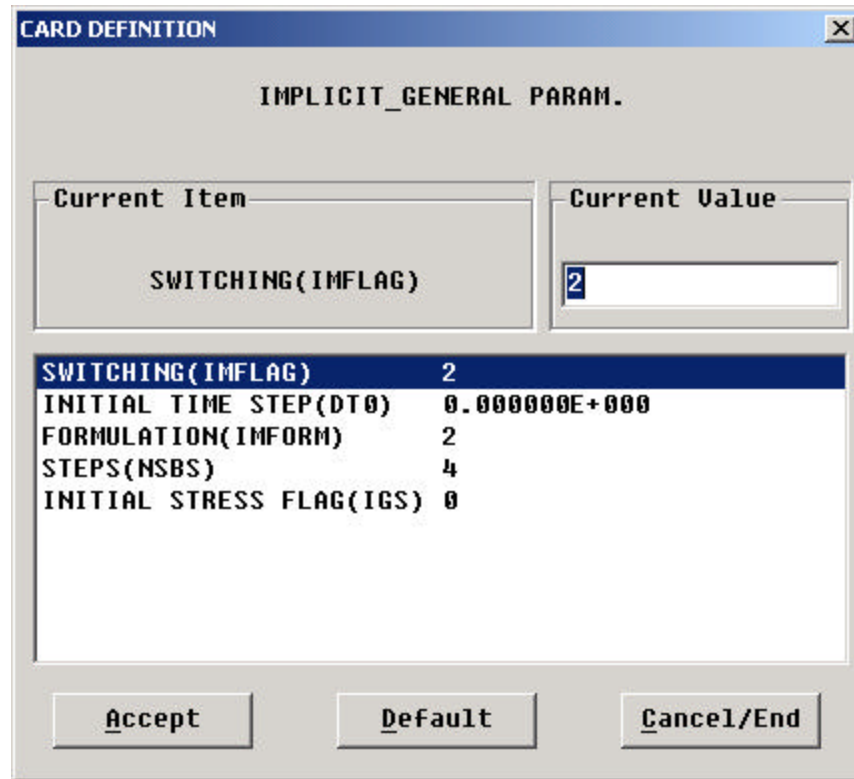
Figure 13.8 IMLICIT_GENERAL Parameters

Click on the variable name in the window to edit the data and ACCEPT will save the modifications. DEFAULT will reset all the data in the card to the default values and CANCEL/END will discard all current modifications made to the card. Refer to the LS-DYNA User's Manual for more information on control parameters for an implicit solver.

SPRING_CONTRAINT allows the user to define the constraints for springback simulation. It is recommended to avoid rigid body movement of the blank during the solution process. DYNAFORM-PC displays the blank in the display area and prompts the user to select three nodes for constraint. The program will automatically assign the constrain degree-of-freedom to the three nodes: all three transitional D.O.F. (X,Y, Z) will be fixed in the first node, horizontal D.O.F. (X,Y) will be fixed in the second node and vertical D.O.F. (Z) will be fixed in the third node. These three nodes should be in the area of the blank where minimal springback is anticipated and they must not be co-linear.

13.2.5 ADAPTIVE MESH (toggle)

If it is on, ADAPTIVE PARAMETERS button will be activated. You can then define the parameters by clicking the button and editing the dialog box. Please see Figure 13.9.

Note: Refer to *LS-DYNA User's Manual*, keyword **control-adaptive*, for detailed information.
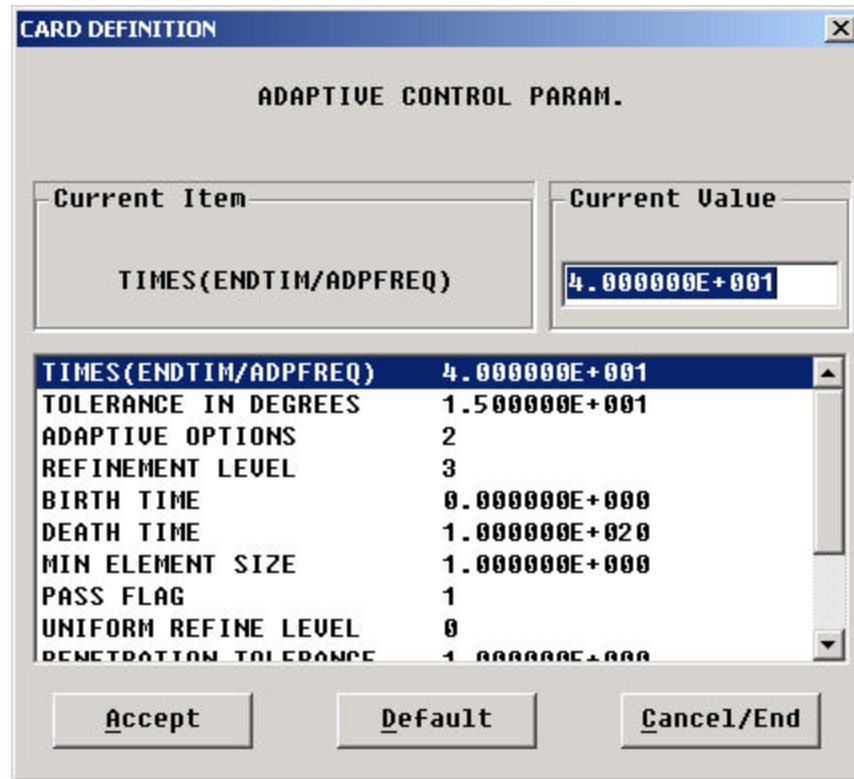
Figure 13.9 Adaptive Control Parameters

13.2.6 CONTROL PARAMETERS

This function is to define control parameters. A dialog box will display as Figure 13.10.

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

Figure 13.10 LS-DYNA Control Parameters

13.2.7 INPUT FILE NAME

If ANALYSIS is set to LS-DYNA INPUT FILE, a file name must be given. Select the arrow button to right of the field to display the EXPORT LS-DYNA INPUT FILE window. The user can enter a name for the LS-DYNA input file in a selected directory. Two files will be written *.dyn and *.mod. The .dyn file contains the analysis parameters and material property data and .mod files contains the node and element data. The .dyn file will be required for the LS-DYNA analysis while .mod file will be required as an include file.

13.2.8 MEMORY

The memory is allocated for the LS-DYNA run. The memory size depends on the number of nodes and elements. The default memory size is 127MB.

13.2.9 TITLE

Select the field and type a brief description for the analysis.

13.2.10 TERMINATE TIME

Termination Time is given automatically by the program according to the motion curve.

13.3   QUICK SETUP

This interface allows the user to set up a model and submit it for analysis quickly. The user defines the binder, die, blank and drawbead, and DYNAFORM-PC automatically offsets the mating parts. Drawbeads are defined by line data input and must be defined in the same part. Binder runout and die cavity must be separated for a binder to automatically be created. After setup, selecting preview generates the mating parts, travel curves and sets the default parameters for job submission. These default parameters are controlled by a text file in the DYNAFORM installation directory (profile.txt). This file can be edited according to user defined paramaters. These parameters are not compatible with standard setup parameters. Once the preview button has been selected, these default parameters are assigned to the current database. Do not attempt to use this model for standard setup. The Quick Setup interface is displayed below. *For more information on the Quick Setup procedure please consult the "Quick Setup viewlet tutorial". This viewlet is available on the 2.0 release CD and with standard training materials.*
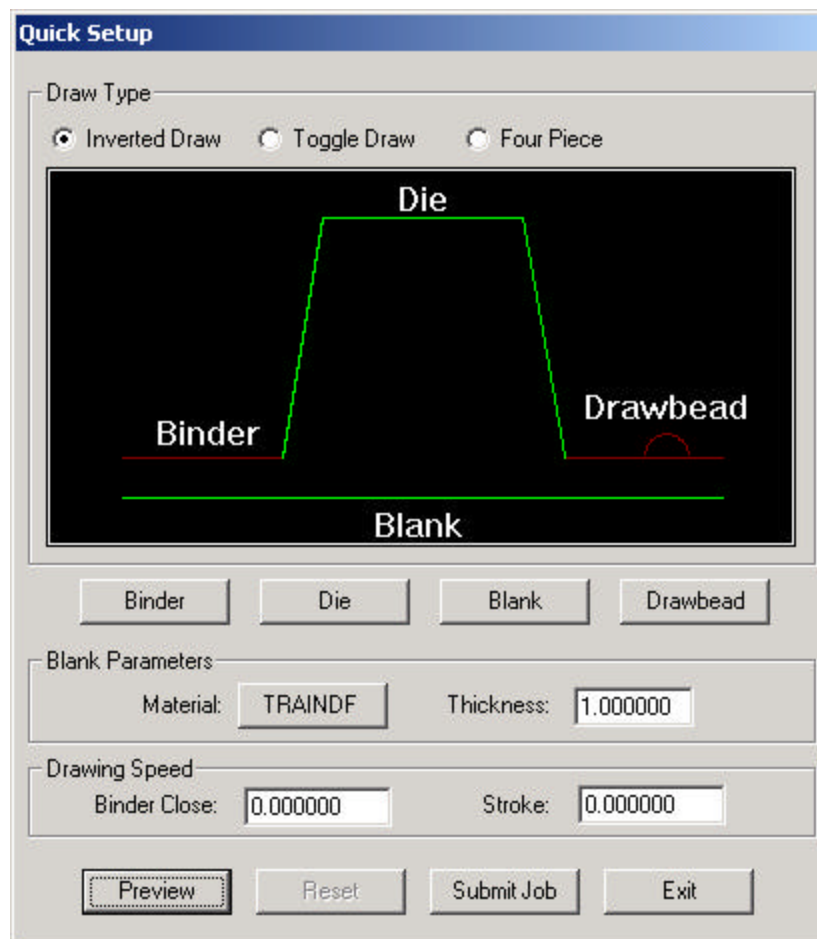
Figure 13.11 Quick Set up

13.3.1   DRAW TYPE

There are three draw types.

     1. INVERTED DRAW
        The female tool is above the blank and the punch and binder are offset from the upper. The tools and blank are configured as pictured:
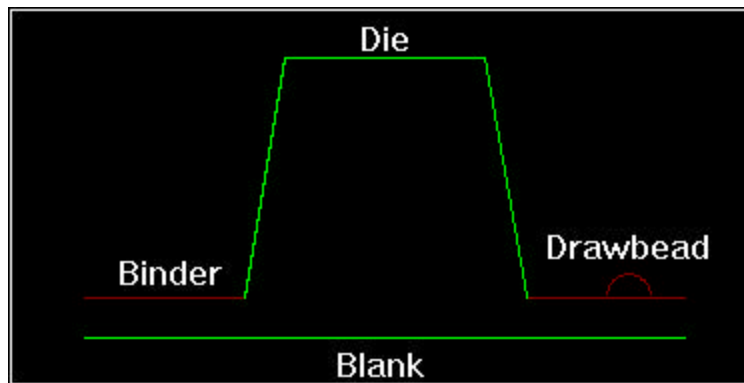
Figure 13.12 Inverted Draw

This configuration can be used for crash form scenarios by leaving the binder undefined.

     2. TOGGLE DRAW
        The female tool is under the blank and the offset punch will be above the blank. The tools are configured as displayed below.
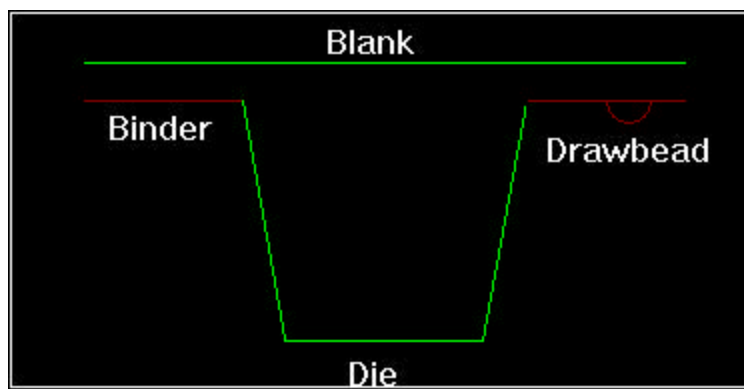
Figure 13.13 Toggle Draw

This configuration can be used for crash form scenarios by leaving the binder undefined.

3. FOUR PIECE
   The die is above the blank and the created punch will be under the blank. The upper and lower binder rings close first and then pull the blank over the post before the punch begins to move.
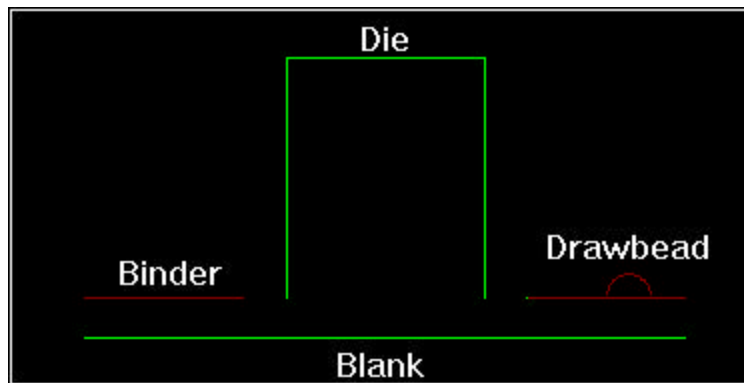

Figure 13.14 Four Piece

13.3.2   DEFINE TOOLS, BLANK AND DRAWBEAD

1. BINDER
   Once this button is selected, a list window is displayed.
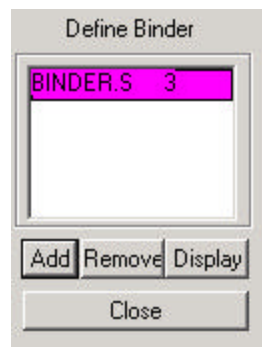
Figure 13.15 Define Binder

   ADD

Allows user to select part(s).

REMOVE
Allows user to remove parts.

DISPLAY
Displays only the defined binder in the graphic display window.

2. DIE
   This function behaves the same as the DEFINE BINDER function.

3. BLANK
   Please refer to the section 9.2 DEFINE BLANK to get detailed information about this function.

4. DRAWBEAD
   Once this function is selected the Define Drawbead window is displayed:

Figure 13.16 Define Drawbead

FULL LOCK FORCE
The full lock force is based on material type and thickness. If no material has been defined, this value will be zero.

DEFINE BEAD
Once the ADD button is selected the user can define the drawbead. Drawbeads

are defined as line data and can only be assigned to one part. However, multiple
lines can be assigned to the same part for multiple drawbeads.

> SELECT A PART TO DEFINE DRAWBEAD
> PICK AN ELEMENT TO IDENTIFY A PART

If there is no line in the selected part, DYNAFORM-PC will prompt:

> PART XXX IS SELECTED
> No line in the part

For each line in the selected part a drawbead will be displayed in the window
with a default lock force of 20 percent.

CHANGE PERCENT
DYNAFORM-PC allows the user to change the Lock Bead force percentage.
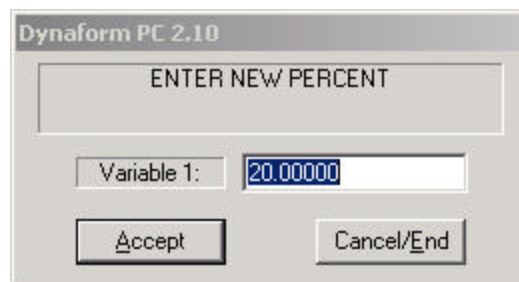
Figure 13.16 Enter New Percent

REMOVE
Removes all drawbeads from the current list.

13.3.3   BLANK PARAMETERS

This function allows the user to define the blank material and the blank thickness.

1. MATERIAL
    Please refer to section 9.4 DEFINE MATERIAL to get detailed information about
    defining material

2. THICKNESS
    Determines blank thickness.

13.3.4   DRAWING SPEED

This function allows the user to define tool velocity. Numbers displayed are in millimeters per

second.
> 1. BINDER CLOSE
>   Defines binder-closing velocity.
>
> 2. STROKE
>   Defines stroke velocity of the tool.

13.3.5 PREVIEW / RESET / SUBMIT /EXIT

After the model setup is finished, these functions determine what will be done with the model.

> 1. PREVIEW
>   This function generates and assigns all travel curves mating parts and default setup parameters.
>
> 2. RESET
>   This function deletes all generated travel curves, mating parts and drawbeads.
>
> 3. SUBMIT JOB
>   Takes the user to the Analysis Menu for job submission. Please refer to section 13.2 to get detailed information.
>
> 4. EXIT
>   Exits the Quick Setup interface without changing the model. This model can be saved for later analysis.

APPENDIX A

HARDWARE AND SOFTWARE REQUIREMENTS

DYNAFORM-PC 2.0 is compatible with LS-DYNA/PC 960.   It will run in a Windows 95, 98, 2000 or NT 4.X (or higher environment. It is not recommended for use with earlier versions of Windows.   The following are minimum requirements for proper operation of DYNAFORM-PC in a Windows 98 environment:

Minimum Graphics Requirement: 1024x768

> **NOTE:** **The current monitor resolution (e.g. 1024 x 768) is determined by the Windows display settings. The settings can be accessed by clicking the right mouse button on the Windows desktop and selecting PROPERTIES from the pull down menu and SETTINGS from the displayed pop-up window. These settings can also be accessed through the WINDOWS START MENU/SETTINGS/CONTROL PANEL/DISPLAY.**

Minimum Memory Requirement:

Small models (10,000-20,000 elements):	64 megabytes RAM
Medium models (20,000-50,000 elements):	128 megabytes RAM
Large models (50,000-100,000 elements):	256 megabytes RAM
Huge models (100,000-200,000 elements):	512 megabytes RAM

Minimum Loadspace requirement: 2.8 megabytes

APPENDIX B

SUPPORTED IGES ENTITY TYPES

NAME	TYPE
Null Entity	0
Circular Arc Entity	100
Composite Curve Entity	102
Conic Arc Entity	104
Copious Data Entity	106
Plane Entity	108
Line Entity	110
Parametric Spline Curve Entity	112
Parametric Spline Surface Entity	114
Point Entity	116
Ruled Surface Entity	118
Surface of Revolution Entity	120
Tabulated Cylinder Entity	122
Transformation Matrix Entity	124
Rational B-Spline Curve Entity	126
Rational B-Spline Surface Entity	128
Offset Surface Entity	140
Boundary Entity	141
Curve on a Parametric Surface Entity	142
Bounded Surface Entity	143
Trimmed (Parametric) Surface Entity	144
Subfigure Definition Entity	308
Associativity Instance Entity	402
Property Entity	406
Singular Subfigure Instance Entity	408

APPENDIX C

CURVE FILE FORMAT EXAMPLE

LIST DIRECTED FORMAT

```
RECORD 1        :N
RECORD 2        :N1 'TITLE'
RECORD 3        :XDATA1        YDATA1
          .
          .
          .
RECORD 2 + N1          :NDATAN1       YDATAN1
RECORD 3 + N1          :N2 'TITLE FOR CURVE2'
RECORD 4 + N1          :XDATA1        YDATA1
          .
          .
          .
etc.
```

```
   Where:

N       = Number of curves, integer
N1      = Number of points on the first curve, integer
N2      = Number of points on the second curve, integer
TITLE   = Title with a maximum of 20 characters, Starts and ends
          with a quote (').  Separate with a space between N1  or
          N2.

XDATA1 = X value of the first part on the curve, real number

YDATA1 = Y value of the first part on the curve, real number.
         Separate with a space between XDATA1.


etc.
```

FINAL NOTES

We at ETA would like to thank all those who helped in creating this manual. We have all tried to make this manual as accurate as possible. In an effort to keep future versions as error free as possible, we ask that you send us your suggestions and notify us of any errors that you come across. You can contact the ETA software support group at the Troy office via:

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The DYNAFORM Team
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