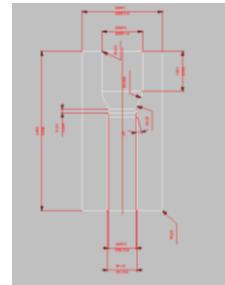
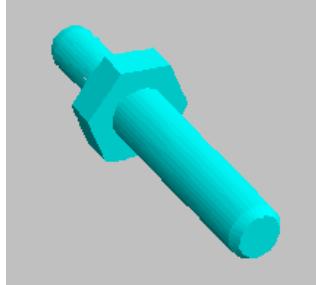
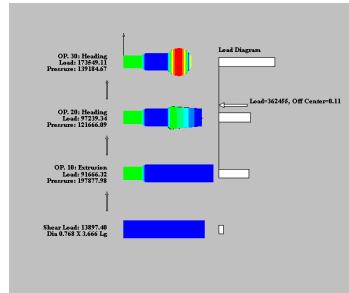
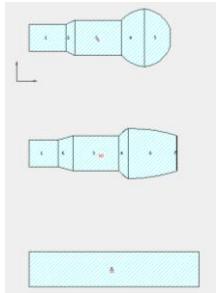


NAGFORM

"Way to Design"



USER'S MANUAL

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1. Introduction

1.1 What is NAGFORM Program?

NAGFORM (Numerical Analysis and Geometry of Forming processes) is an interactive computer-aided design software program for the design of metal forming processes. It uses design guide rules and simple analyses to determine the sequence of forming operations and shapes of the preforms required to form a part. Currently, axisymmetric and 'nearly' symmetric parts can be analyzed using NAGFORM.

The selection of a metal forming process to manufacture a component depends upon numerous factors such as design tolerances, strength, cost, available equipment, lot size, lead time etc. Process design involves selection of forming sequence and process parameters. Analysis of load/stresses is required for selection of equipment and tooling and to ensure that desired tool life is obtained.

1.2 Purposes for which NAGFORM can be used

It can be used to:

1. Create a historic database of parts and their forming sequence designs
2. Search for similar parts in the part database.
3. Create reusable forming-sequence templates for design of similar parts.
4. Obtain alternative forming sequences based on forming rules, for forming a part
5. Create generic tooling and analysis file for NAGSIM.2D FEA simulation program.
6. Assist in Cost Estimating through volume and weight calculation, machine selection, forming sequence selection etc.
7. Construct geometric model of axisymmetric and nearly symmetric components.
8. Obtain part volume and weight.
9. Assist in determining whether the part can be formed by cold or hot forming.
10. Assist in determining the optimal forming-process sequence.
11. Assist in reducing number of operations to form a part.
12. Estimate the strain hardening/strengthening of material from cold forming.
13. Estimate the load and pressures required to form the part.
14. Obtain DXF drawing of forming sequences within a few minutes.
15. Assist in selecting the equipment required for forming.
16. Learn the principles of metal forming and design of metal forming process.

1.3 Other Applications of NAGFORM

•Cost estimating / quoting

- Search for similar parts in your database before creating any designs
- Where possible, use templates to come up with sequence design
- Use 'Machine Selection Summary' to assist in selection of the machine.
- Create designs at cost estimation stage, then transfer the file to Engineering Department for completing the design if the job comes in.

• Product Redesign

- Help customer redesign his product to reduce additional forming or machining operations or improve tool life.
- In NAGFORM, create forming progressions for various redesigns and compare to come up with the improved design.

- **Retain Corporate Knowledge**
 - Built historic database of parts and their designs.
 - Improve the design templates (Auto design or Design by Command)
 - Search for similar parts to avoid reinventing the wheel.
- **Standardize Design Procedures**
 - Use design templates where possible.

1.4 Features:

- . Windows application
- . Written in a object oriented language
- . User friendly
- . Guide rules can be easily adapted
- . Does not require special analytical skills in metal forming.

1.5 Limitations:

1. Design is based upon the forming rules set up by the designer. The software does not perform any metal forming simulations to verify the validity or accuracy of the rules. Like a designer, it can not ensure that the part would be formed as expected.
2. NAGFORM can not predict forming defects such as cold shuts, laps, non-fill of tool cavity etc as it does not perform simulation of the metal flow.
3. The calculation of loads and strains is based upon simplified analyses. The results are only estimates.
4. The program requires a knowledgeable designer to make the judgement on the validity or applicability of designs obtained. The program is intended to assist the designer not substitute for his/her professional judgement.

1.6 Components/Modules of NAGFORM

NAGFORM is a collection of components that can be used independently or collectively to obtain desired results. The main components of NAGFORM are:

1. Geometric Model
2. Automatic Design based on rules
3. Manual Design Modules
4. Tooling Module

1.7 Geometric Model

This module is used to construct geometry of the part to be formed. The geometry that can be built is limited to axisymmetric and 'nearly' axisymmetric parts. Since the primary objective of NAGFORM is to reduce the time required to generate forming progression for any part, this module is written to allow easy and quick construction of the part geometry. The basic entity used is a 'primitive' element or object.

Complete discussion of primitive objects and how these are used in constructing a part-geometry is given in chapter 3 on 'Geometric Model'.

Using this module, the designer can create primitive elements of different dimensions and join them to form a part model. Capabilities to delete, modify, scale, move and mirror primitives and their unions are provided to facilitate construction.

Another approach to construct geometric model is to start from a part template. NAGFORM allows any part and its design to be stored as a template. To construct a part similar to a part previously stored as a

template, the designer can start from the template and modify it to the desired shape and dimensions.

1.8 Material Data

The material data information is needed to estimate the forming loads and the extent of strain hardening from deformation, when cold formed. Strain hardening is the property of a material to strengthen as it is deformed. A convenient method to represent strain hardening of a material is by a constitutive equation representing true stress - true strain curve of the material. A commonly used constitutive equation is:

$$s = k e^n$$

Where
 s = True stress (Load / Actual Area)
 e = True strain
 k = Strain hardening coefficient
 n = Strain hardening exponent

The material module of NAGFORM allows the user to select, create, modify, and display a material in a material database.

1.9 Design Rules

In practice, sequence design is based on design rules. These rules define the limits of forming with various forming operations. The rules provide a starting step for design which is modified to include past experiences and other factors such as equipment and facilities available, quantity to be formed etc. These design rules are different for different operations such as cold extrusion, hot upsetting, cold heading of wire etc.

NAGFORM uses a rule-based approach to determine possible sequence designs. The limits of the rules are set by the user. In addition to the rules, NAGFORM uses other constraints on the design process such as the tonnage available, maximum forming pressure, wire size etc specified by the user. Within the framework of these rules/limitations, NAGFORM determines the possible design solutions.

1.10 Sequence Design

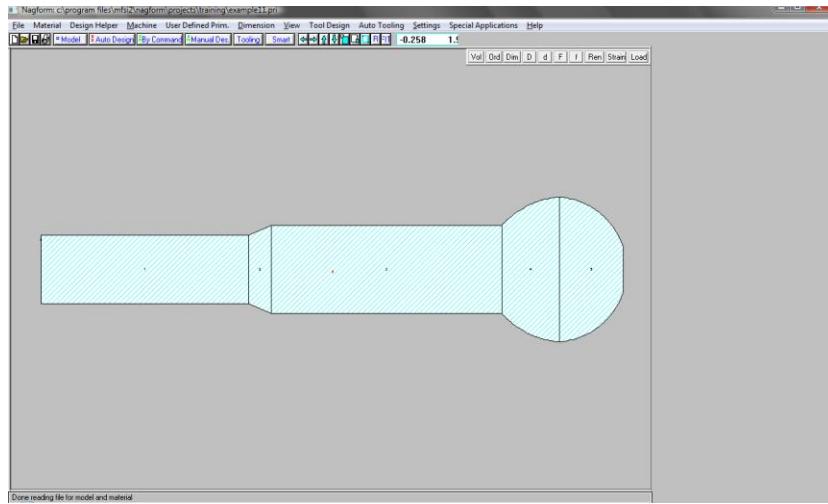
The primary purpose of NAGFORM software program is to determine forming sequence for axisymmetric and nearly symmetric components. The results of the design include determination of:

1. Blank shape and size
2. Number of forming operations required
3. Type of forming operations
4. Shape to which the part is formed in each step
5. Sequence of forming steps

A powerful feature of NAGFORM is that it determines a number of alternative design sequences within the limits specified by the rules and other constraints set by the designer. Another feature of NAGFORM is that it estimates the load required and strain distribution (extent of straining) in the preform at each forming step. A summary of results is presented which can be very useful in selecting the best preform design.

2. User Interface

2.1 Main Window Layout



NAGFORM is a Windows based program. If you have experience in running a Windows program, you will be able to understand NAGFORM format and use it with ease.

The above Figure shows the main application window. This is also the NAGFORM 'Desktop' window used for user interface and display of graphical information and messages. As in any standard 'Desktop' window, it has a title bar, a menu bar below the title bar and then a utility task bar (Tool Bar) that is used for frequently used tasks. There is another smaller tool bar than can be turned on or off. The main menu bar has main titles with subtitles.

When you start NAGFORM, no file is opened. As first step, you must open a 'New' or 'Existing' file. If you click on any tool bar button without opening a file, you will get a message to open a file first.

A part of the 'Desktop' is covered by the graphics window created automatically when you start NAGFORM. The graphics window has a fixed size and location. The main purpose of this window is to display graphical information such as shapes of the part and the preforms. Some textual information on results may also be displayed in this window. The graphics window is extensively used during construction of the geometric model and during sequence design phase. The 'View' menu and 'Setting' menu control the display of graphics. Objects can be zoomed in or out, moved and scaled to fit the screen using the view menu. Another window that forms part of the desktop is the Messages window. It is a small window located at the bottom of the screen. Messages useful to the user in running NAGFORM are displayed here.

2.2 Accessing NAGFORM Commands

The two methods of accessing NAGFORM commands are by

- 2.2.1 Menu bar
- 2.2.2 Utility task bars (Toolbars)

2.2.1 Menu

Under the title bar is the main menu. This menu bar allows user access to almost all of the commands. You can execute these commands by picking them with the mouse as in a standard window. All of the commands on the main menu have another 'drop-down' menu. Some of the menu items in the 'drop-down'

menu have a small arrow on the right side of the menu. Selecting one of these menu items will display another lower level menu. A menu item that has three dots at the end of the text opens a dialog box when chosen. Following is a list of some available menu options:

File menu:

<i>New</i>	Opens a special window dialog box to create a new file
<i>Open</i>	Opens the standard window dialog box to open an existing file
<i>Close</i>	Closes the current file
<i>Save</i>	Saves the current file and continues with the program
<i>Template</i>	Add or remove a template file from the template database
<i>Preview Part Geometry</i>	Preview part geometry in certain directories without opening the files
<i>Search Similar Parts</i>	Search similar parts in certain subdirectories
<i>Print</i>	Print the display in Graphics window
<i>Print Design Summary</i>	Print summary of all forming sequence designs
<i>Add Comments</i>	Add comments in the Comments (.com) file of the project
<i>Set Path to User DB</i>	Set path to the user database of parts for search purposes
<i>Exit</i>	End the program and exit

Material Menu Opens the *Material* Dialog Box when ‘Select’, ‘Create’, ‘Modify’, ‘Delete’ or ‘Show’ is chosen

Design Helper Menu For help in sequence design. Various options are:

- Not Getting Any Designs
- Getting Too Many Designs
- Reduce Num. Of Operations
- Blank Size, Weight

Machine Menu Opens the Machine Database Setup Dialog Box

User Defined Prim Opens the Dialog Box for creating/modifying/deleting/bringing in/saving etc of user defined primitives

Dimension Options to create / modify dimensions

View Menu

<i>Redraw</i>	Refreshes the graphics window display
<i>Zoom In</i>	Zooms in the display of graphics window enlarging the displayed objects but covering less area of the current display
<i>Zoom out</i>	Zooms out the display of graphics window reducing the displayed objects but covering more area of the display
<i>Center</i>	Brings the center of display in the center of the graphics window area
<i>Window</i>	Allows the user to select an area of the current display to be displayed on the full graphics window
<i>Fit</i>	Changes display scale to fit all objects in the graphics window area
<i>Scale</i>	Display current scale of magnification which can be changed
<i>Render Design</i>	Ray tracing method of rendering sequence design
<i>Wireframe/Render</i>	Draw wireframe and rendered view of the display in graphics window

Settings Menu

Design Opens dialog box to Change the Display, Design Parameters, and Output settings

Tool bar on/off Turns on or off the small tool bar at the top right corner of the Graphics Area.

Help Menu:

Version Opens the dialog box to show program version.

Help Topics Opens the dialog box for choosing a help topic to display

Calculate Opens a dialog box for calculations on volume, weight, reduction etc.

2.2.2 Utility task bar (Toolbar)

The utilities bar has a number of buttons to perform frequently used tasks:

New File: Open a New File.

Open: Open an Existing File

Save File: Saves the file

Print: Print Screen

Model: Opens the *Model* Dialog Box

Auto Design: Opens the *Preform design: Set up* Dialog Box

Manual Design Opens the ‘Manual Design’ dialog box. Includes ‘Design by Command’

Tooling: Opens the ‘Tooling Design’ dialog box

Left: Scroll graphics display to Left

Right: Scroll graphics display to Right

Up: Scroll graphics display Up

Down: Scroll graphics display Down

Zoom Out/In/Window: Zoom Functions

Redraw: Redraw the drawing with the updates

Fit: Fits the objects in graphics window

2.2.3 Small Toolbar

This removable tool bar is located at the upper right corner of the ‘Graphics area’. It can be shown or hidden using Main Menu ‘Setting’ – Tool Bar On/Off selection. It has a number of buttons to display items such as volume, dimensions, rendering, strain and load. Click these buttons once to display and click again to hide the item.

Vol: Turn on /off display of volumes of the primitives and the unions. Volume of unions is written in Red

Ord: Turn on/off display of ordinate type dimensioning

Dim: Turn on/off display of conventional ‘Automatic’ dimensioning. You may need to click on ‘R’ redraw button to display the full automatic dimensioning

F: Enlarge Font size of identification numbers of primitives and unions

f: Reduce Font size

Ren: Show rendered view. A dialog box appears at the bottom. You have to close this dialog box to turn off rendering

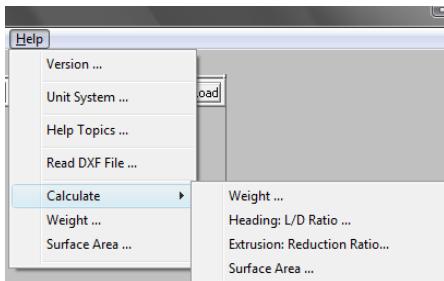
Strain: Turn on/off display of strains in parts in ‘Auto’ design

Load: Turn on/off display of load distribution in ‘Auto’ design

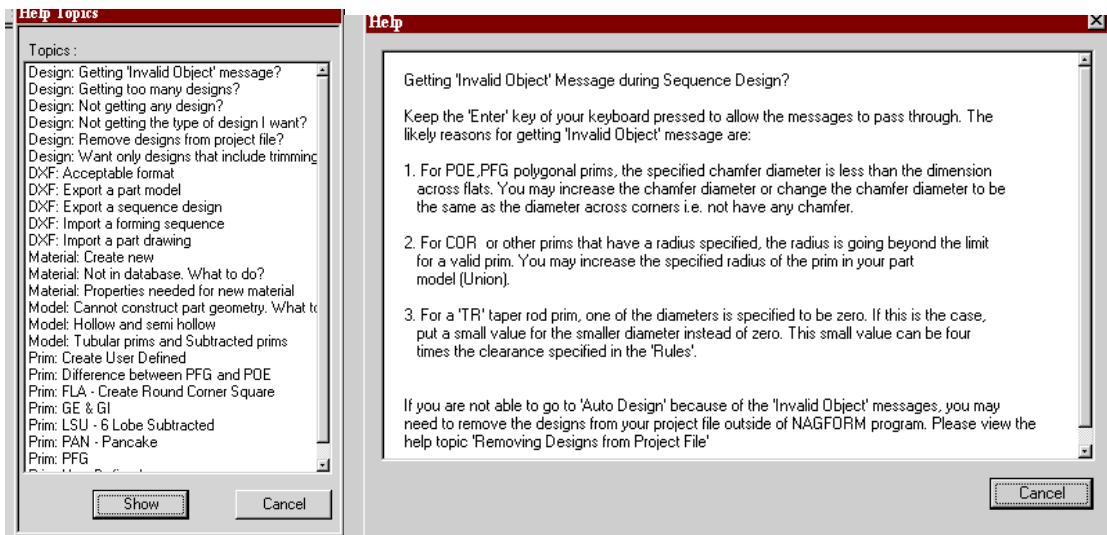
2.2.4 Help

Help menu has three selections:

- Version ...
- Unit System ...
- Help Topics ...
- Read DXF File ...
- Calculate ...

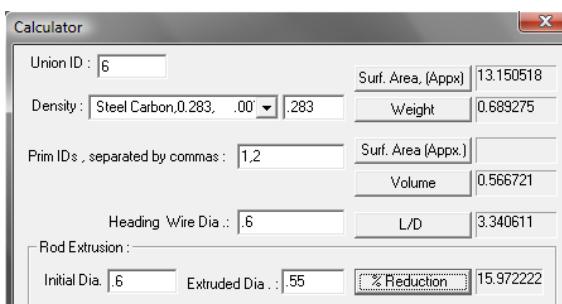


- Version: Selecting this opens a Message Box that displays the version of the NAGFORM program.
- Unit System: Selecting this displays the current file Unit System



- Help Topics: Selecting this opens the dialog box shown below. Select the topic you want to see and press 'Show' button. This would open a dialog box shown below. The relevant help document is displayed in this dialog box.
- Read DXF File: Allows the user to read and display a .DXF file.
- Calculate ... Selecting this open up the dialog box shown below.

Enter the union ID, enter density and press on 'Weight' button to give you the weight of the part. Note that the density values given in the drop down list are approximate values as these are averaged for a class of material. To be more accurate, enter the actual value of the density of the material you are using. Also you must insure that you are using the proper units for part dimensions and density to calculate the weight.

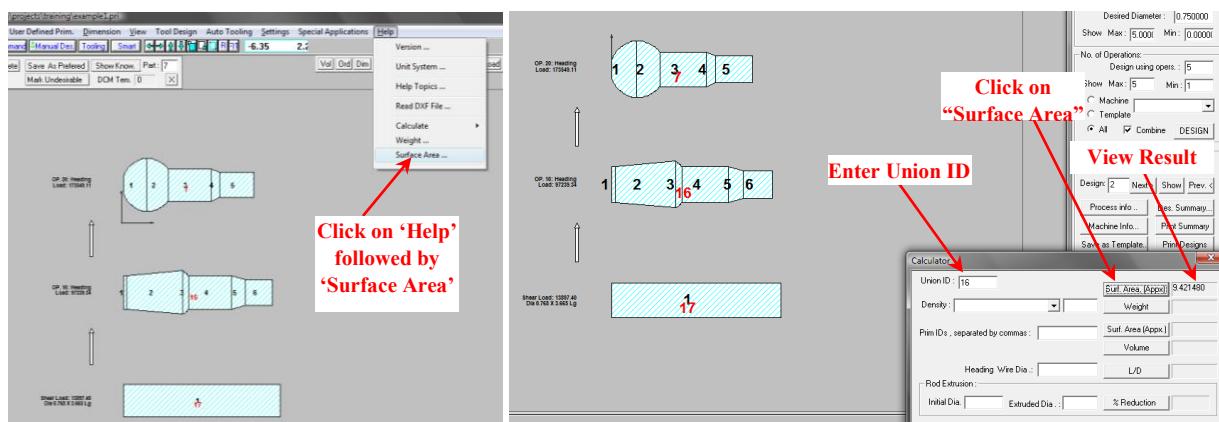


To calculate L/D ratio for upset/headed portion of the part, enter the prim Ids separated by comma, enter the diameter ‘D’ and press on L/D button to get the ratio.

To find percent reduction in rod extrusion, enter the initial diameter and the diameter after extrusion. Press % Reduction button to get the percentage reduction in area.

Surface Area Calculations: NAGFORM now has the capability to calculate the surface area of any part including the model and preforms. Following are the steps needed in order to calculate the Surface Area:

- Click on ‘Help’ followed by ‘Surface Area’
- Enter Union ID (in this case ‘16’) and click on ‘Surface Area (approx)’ Button. The program will display the Surface Area on the Right.



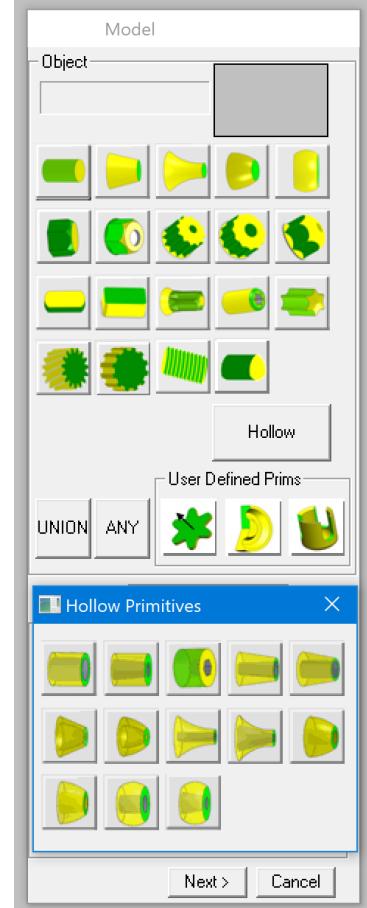
3. Geometric Model

3.1 Primitives

In NAGFORM, Geometric CAD model of a part or a preform is constructed by joining simple building blocks called **primitive** objects or **primitives** shown in Figure on next page. The various primitives available in NAGFORM are:

Description

Rod
Rod-Taper OD
Corner
Fillet
Pancake
Polygon
Polygon with Taper Matl.
12-Point (Double Hex)
12-Point (Triple Sq)
Polygon with Spherical End.
Rod with flats
Rectangle (with Radius)
Six Lobe – external
St. Tube with Six Lobe ID
Six Lobe for Subtraction
Gear – Helical and Spur
Standard Threads
Tri-Arc Feature
Hollow Primitives



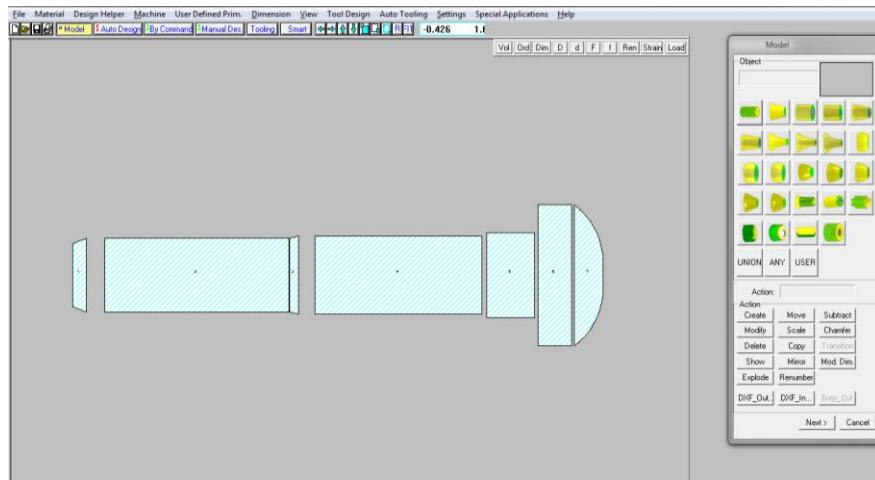
For Deleting Purposes
User Defined Primitives

ANY – for deleting any primitive, not for auto designs
USR

Primitives other than POE, PFG, POI, FLA, LOB, ILB, LSU, GEA, GE, GI, USR are axisymmetrical, i.e. all radial cross sections through the axis are the same. POE, PFG etc. have a central axis about which they are placed and are ‘nearly’ axisymmetric. All the primitives are defined by certain dimensions. For example, a ‘Rod’ primitive is defined by coordinates (xc,yc) of its starting point O, length (L), and outside diameter (D). A ‘Tube-Taper ID’ is defined by its starting point (xc,yc), length (L), outside diameter (D), and the inside diameters (do and df) (see figure above). Each primitive should have a unique ID number.

3.2 Union

A ‘Union’ consists of a number of primitives joined together. A part or a preform is a ‘Union’ with which a ‘Material’ has been associated. Figure below shows individual elements of a hex-bolt that are joined to create a union.



3.3 Construction of A Geometric Model of A Part

Construction of a geometric model of a part consists of the following three steps:

1. Divide the part geometry into primitives
2. Create these primitives in NAGFORM
3. Join these primitives into a ‘Union’ in NAGFORM
4. For hollow part created by ‘Subtraction’, create the primitives to subtract
5. Subtract these primitives from left and/or right end of the union as required

You can also bring in part geometry in DXF format from any CAD program. There are, however, certain requirements on how the geometry must be drawn in DXF format. Read the section on “Import and Export in DXF Format” for further instructions.

3.3.1 Divide Part Geometry into Primitives

This step is done outside of NAGFORM. The geometry of the part in a drawing is divided into primitives.

3.3.2 Create the Primitives

The second step is to create the primitives of required dimensions. This is accomplished using dialog boxes as outline below:

1. Click the ‘Model’ button in the main window desktop. This opens the ‘Object-Action’ dialog box as shown in Figure above. Click the primitive you wish to create, then click the ‘Create’ action button and press ‘Next’ to move to the next dialog box. When a primitive object is clicked, a diagram of the object is displayed in the ‘Model’ dialog box. This should help you in selecting the desired primitive.
2. Clicking ‘Next’ in previous step closes the Object-Action dialog box and opens another dialog box

called ‘Primitive-Create’. This dialog box provides three ways to create a primitive. One method is to create a default primitive first. When the ‘Default’ button is clicked, the user can define a window that serves as the boundary for the primitive to be created. The proportions of this primitive object are defined by the width and height of the drawn window. This default primitive would not have the dimensions you want. So an additional step to modify the default primitive to correct dimensions has to be performed. This option is useful where the designer wants to sketch a part and try a few variations before defining the actual size and shape of the part.

Another way to create a primitive is to enter its dimensions in a ‘short’ format. This format is shown in the dialog box to help the designer enter the dimensions in this format.

The third method to construct a primitive is to select the ‘full’ format button. This displays a number of data entry boxes for entering the dimensions of the primitive. The dimensions to be entered are shown in the diagram of the primitive drawn in the dialog box. After the dimensions are entered, click ‘Apply’ button. A primitive will be created and drawn on the screen.

3. Repeat above step (2) to create other primitives that make up the part. You can use the Cross-Lines on the screen and their center point location shown in the tool bar to place any primitive on the screen.

3.3.3 Join the Primitives into a Union

1. In the Object-Action dialog box, select ‘Union’ as the object and ‘Create’ as the action. Then click ‘Next’.

2. The Object-Action dialog box will close and Union-Create dialog box will appear. The primitives to be joined are added into a list in the order in which they are to be joined. Enter the IDs of the primitives in any of the acceptable formats, shown in the dialog box ,and press ‘Add’ button to enter it in the list box. Repeat this for other primitives to be added. To enter a number of primitives with IDs from A to B simultaneously, you can use the format 'A:B' in the edit box. You can also write the IDs separated by commas to enter a number of primitives in the list box. For example, entering 2,5,10 in ID edit box and pressing enter would add primitives with IDs 2, 5 and 10 to the list box. If any primitive is to be removed from the list, select that primitive in the list box and press ‘Remove’.

After all the primitives have been entered in the list in the order in which they have to be joined, press ‘Union’ button. All the primitive selected except the first primitive will move from their current locations in the graphics window and join in after the first primitive in the order listed in the list box. If for any reason, the ‘Union’ formed is not correct, you can undo the ‘Union’ process by pressing ‘Undo’ button.

3.4 Modification and other actions

In the ‘Object-Action’ dialog box, the ‘Action’ to be performed on any primitive or a Union can be any one of the following:

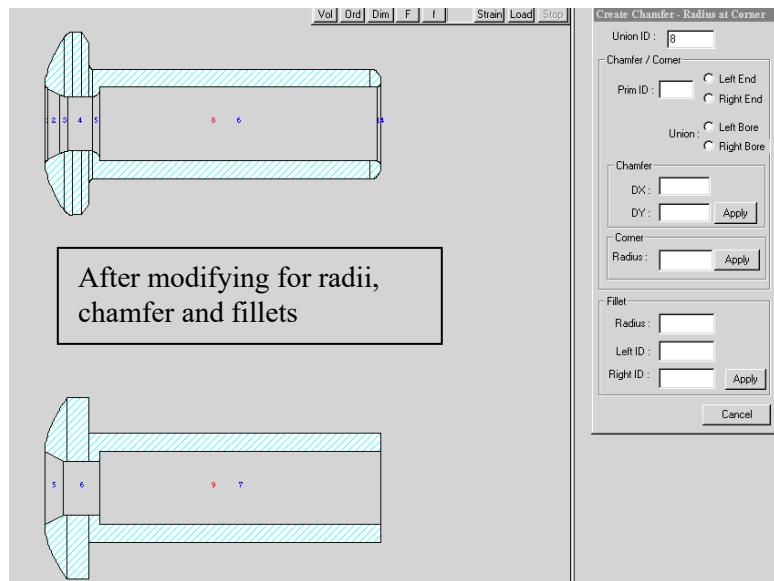
- Create: Create a primitive or a union
- Move: Move a primitive or a union
- Subtract: Subtract primitives from a union.
- Modify: Modify a primitive or a union
- Scale: Scale a primitive or a union. This can be used to convert dimensions in mm to inches and vice versa.
- Chamfer: Put chamfers, corner radii or fillets on a union. Union can be constructed first without any chamfers, radii etc which can be put through this ‘Chamfer’ action.
- Delete: Delete any primitive or a union
- Copy: Create a copy of a primitive or a union

- Transition: Under Development. For creating transition primitive joining a non-axisymmetric prim such as a User Defined Prim, prim with flats etc with a Rod type primitive.
- Show: Show dimensions
- Mirror: Create a mirror of a primitive or a union
- Mod. Dim: Modify part dimension by modifying dimensioning
- Explode: Break up a union into primitives
- Renumber: Automatic renumbering of the primitives and unions on the screen.
- DXF_Out: Create a DXF output of the part geometry
- DXF_In: To bring in geometry from a DXF file
- Brep_Out: Under development. Create a BREP file for non-symmetric or symmetric part geometry. This would be used for 3D simulation using NAGSIM.3DS FEA program.

3.4.1 Modify Union to Create Corner Radii, Chamfers and Fillets

The user can create ‘COR’ and ‘Fl’ type primitives to construct the part features of corner radii, chamfers and fillets. This, however, is cumbersome as the dimensions of these features have to be calculated outside of NAGFORM program. In this version, the union can be modified easily in NAGFORM to put in these features.

In the Model dialog box, a ‘Corner’ button is added in the ‘Action’ group. Select UNI (union) as the object and ‘Corner’ as the action. Press ‘Next’ to go to the dialog box shown below.



For Corner Chamfer: Enter Id of the prim, select the prim end (left or right), enter chamfer lengths in X and Y directions (DX, DY) and press ‘Apply’ in the chamfer group box. This can be done for solid as well as subtracted prims.

For Corner Radius (Convex): Same as above except enter corner radius and press ‘Apply’ in the corner group box.

For Chamfer /Corner Radius at ends of Part Bore: Same as above except select union end instead of prim end.

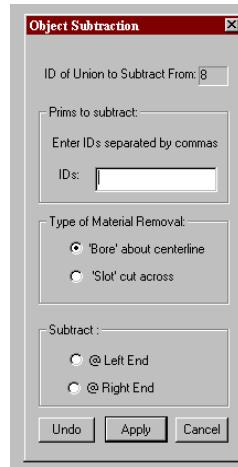
For Fillets (convex or concave) at the junction of two prims: Enter Id of the prim on left, Id of the prim on

right, enter radius and press apply. Note that both the prims must either be solid or subtracted.

3.5 Subtraction of Primitives to Create Hollow Parts

In NAGFORM program, hollow parts can be constructed by first creating a union representing the outside of the part and then hollowing it out by subtracting solid primitives from inside of the union. To subtract a primitive or a number of primitives:

1. Create the outside profile of the part by joining solid primitives into a UNION.
2. Create primitives (solid) to be subtracted from the UNION.
3. Subtract the primitives at left end or at right end of the UNION using the dialog box shown below. This dialog box is called by selecting ‘Union’ as the Object and ‘Subtract’ as the action in the Object-Action dialog box of ‘MODEL’ module.
4. Enter the IDs of primitives to be subtracted, separated by comma. ID of the end prim to be written first, then next and so on.
5. Choose left end or right end and press ‘Apply’
6. You can undo if you made a mistake,
7. Option to remove material as a ‘Slot’ is not implemented yet.

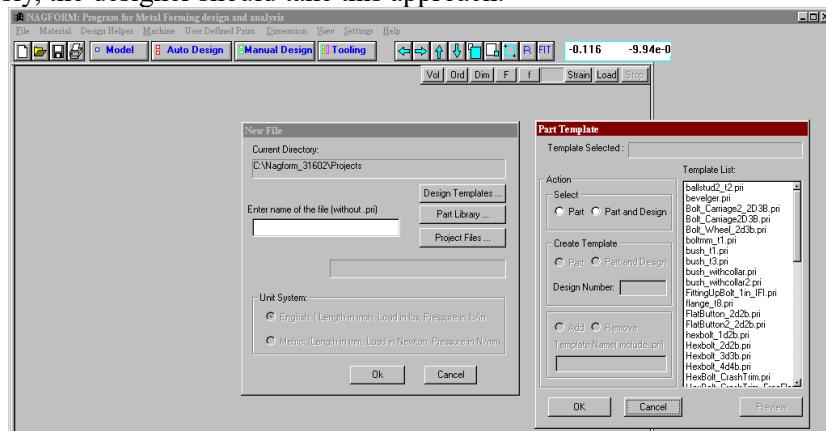


Note that:

1. Only solid primitives can be subtracted. You can not subtract a tubular primitive.
2. A union can not be subtracted from another union.
3. A primitive can not be subtracted from another primitive. Primitives can be subtracted from a union only. If the outside of the part requires only one primitive, you must still go through the process of creating a union with one prim.

3.6 Using Part Template to Construct Geometric Model

Another approach to construct geometric model of a part is to use the template feature of NAGFORM program. For family of parts or where the part in question is similar to a part that was designed previously, the designer should take this approach.



Any part geometry and its design can be stored in a template part and design. When a new file is opened,

click on the 'Template' button on the 'New File' dialog box. A 'Template' dialog box will open as shown in the figure above. The list box in this dialog box shows the names of all templates available. Select the template you wish to use, and then click OK. This template dialog box will disappear and the name of template selected will appear on the 'New File' dialog box. When you close this form, the template part will appear on the graphics screen. Now use the Model dialog box to modify the geometry as explained earlier.

3.7 Using Existing Project File in 'Projects' or 'User DB' to Construct Model

Another approach to construct geometric model of a part is to bring in a copy of a existing project file at the time of starting a new file. This would bring in a copy of the part in the new file. Modify the part as needed. This procedure is similar to windows 'Save As' procedure.

3.8 Bring 'Geometry' from another CAD System to Construct Model

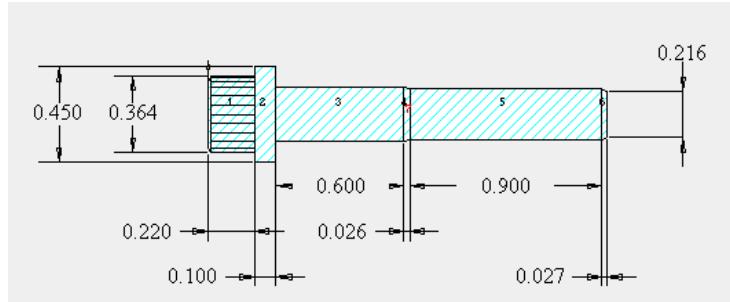
Another approach to construct geometric model of a part is to bring in geometry of the part from another CAD system. The acceptable file format for bringing in geometry is the DXF format. Read the section 'Import Drawing in DXF Format into NAGFORM' for details.

3.9 Dimensioning

Two types of dimensioning is available in NAGFORM.

- Ordinate Type
- Conventional

In conventional dimensioning, Auto dimensioning is now included. For conventional automatic dimensioning, you can press 'Dim' button in the small tool bar. You can also select 'Auto Dimension' from the 'Dimension' menu and from the 'Settings' – Design dialog box. An example of Auto Dimensioning is shown below.



You can hide or display a dimension by going to menu 'Dimension' and then 'Modify' selection. The size of the dimensions can be changed in 'Setting'- Design menu dialog box.

4. Material

For estimating the forming loads for various operations, the required material properties are:

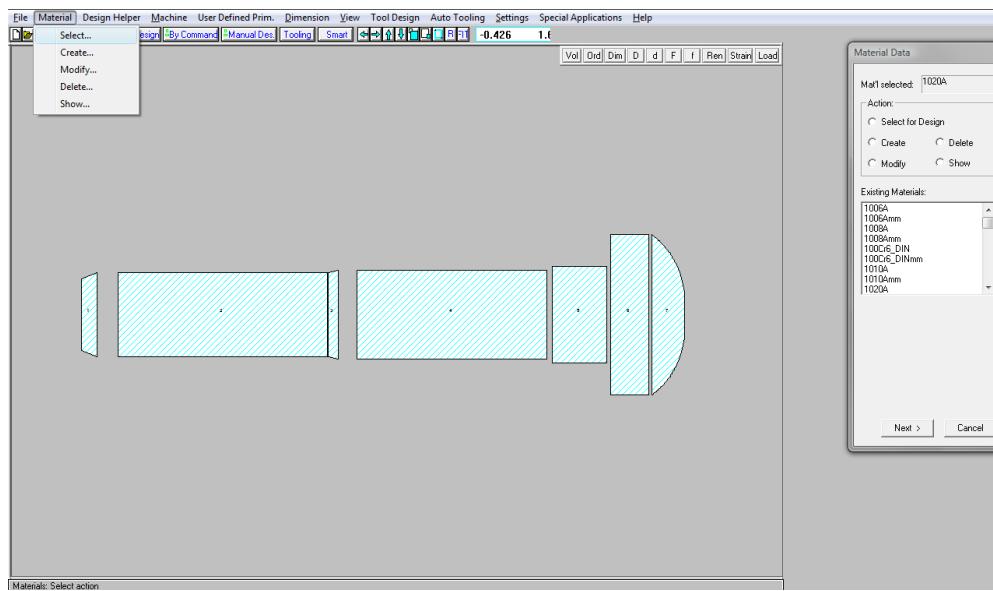
- . Yield strength
- . True stress - true strain data in form constitutive equation

In NAGFORM, properties of a material are stored under its name in a material database. Before design, the user must specify the part material by selecting it from the material database. If the database does not contain the material, the user would need to establish the material properties through testing, and then create and store the material properties in the database.

Important Note: The material properties depend upon the initial condition of the material. Also there are lot-to-lot variations in material composition and properties. An 80 % spheroidized annealed 5120 has quite different properties than a cold rolled 5120 steel. The material data provided with NAGFORM is derived from literature and therefore is approximate and crude representation of the actual materials. The user should obtain accurate data by standard tests for better estimation of loads.

In NAGFORM, the user can perform following actions using the 'Material data' dialog box. This dialog box comes up when you select any sub menu from 'Materials' from the menu bar.

- Select a material from the material database for a part
- Create a new material and store it in the material database
- Modify an existing material in the database
- Delete an existing material from the database
- Display the properties of an existing material in the database



4.1 Select a material for a part

Choose 'Material' from the main menu and then choose 'Select Material'. A 'Material Data' dialog box will open as shown in the Figure above. 'Select for Design/Analysis' button is chosen as the default selection. The dialog box shows the current materials in the database in a list box. By scrolling through the list, you can see if a particular material is in the database or not. Select

the material in the list box and press OK.

4.2 Create a New Material in the Material Database

1. To create a new material, the user must have the property data for the material. Choose ‘Material’ and then ‘Create’ from the menu. The ‘Material Data’ dialog box will appear. Select ‘Create’ if not selected already. The dialog box will change to include more options as seen in the figure.
2. Enter material’s name in the material name edit box. If a material with the same name exists in the database, a warning message will appear. You can change the name entered or accept to write over the existing data.
3. Select ‘Constitutive Equation’ format for entering the plastic stress-strain data of the material
4. Click ‘Next’.
5. The ‘Material Data’ dialog box will close. Another dialog box ‘Material-Create’ will appear where the data in form of constitutive relation is to be entered.
6. Enter in the property data including modulus of elasticity, yield strength, and plastic stress-strain data in form of constitutive equation (k, n). Note that ‘Formability Index’ that appears on this dialog box is not being used presently. A formability index of 45 is being used for all materials. In future, this index will be used to change default values of the rules for different materials.

4.3 Modify an Existing Material

1. To modify an existing material, that material must be in the database. Choose ‘Material’ and then ‘Modify’ from the menu. The ‘Material Data’ dialog box will appear. Select ‘Modify’ if not selected already. The dialog box will change to include more options (Figure above).
2. Select material by clicking on it in the list box.
3. Select the form in which you want to enter the plastic stress-strain data of the material
4. Click ‘Next’.
5. The ‘Material Data’ dialog box will close. Another dialog box ‘Material-Modify’ will appear.
6. Change as needed, the values of modulus of elasticity, yield strength, and plastic stress-strain in form of constitutive equation. Click OK.

4.4 Deleting a Material from the Database

1. To delete an existing material, that material must be in the database. Choose ‘Material’ and then ‘Delete’ from the menu. The ‘Material Data’ dialog box will appear. Select ‘Delete’ if not selected already.
2. Select the material in the list box and press ‘OK’.
3. A message will appear asking you to confirm that you wish to delete that material.

4.5 Material Properties in Metric Units

In material database, materials with properties in metric units have been added for design in Metric units. To distinguish these materials from materials with properties in English units, their name includes ‘mm’ at the end. For example 1020Amm name specifies that the material is 1020 annealed and its properties are in Metric units.

5. Progression Design

Progression design includes determination of:

1. Sequence of operations to achieve final part configuration.
2. Geometry of the preform at each operation.
3. Forming loads for each operation.
4. Deformation strains at each forming step and total strains in the formed part.

Like any other design problem, a preform-design problem for forming has multiple solutions i.e., it is possible to find a number of sequence of operations that would yield the final part configuration. The challenge is to find that design solution which meets the requirements within the manufacturing and other constraints. The process of preform design thus consists of two steps:

1. Finding all possible sequence of operations within the constraints of the problem.
2. Determining the optimal solution out of these possible solutions.

It should be noted that because the constraints may differ for different organizations, the optimal preform design for the same part would invariably be not the same for different manufacturing facilities.

5.1 Possible Sequence Designs:

Possible sequence designs are limited by one or more of the following:

1. Available equipment: Types of machines available and its specifications limit the processes that can be performed.
2. Workpiece material: Properties of the work-piece material such as yield strength, strain hardening and total deformation before fracture determine which processes can be employed.
3. Strength of the tooling: The tooling must be able to withstand the forming loads. The forming operation and the extent to which the part can be formed in an operation is limited by the strength of the tooling.
4. Other considerations: Lubrication, ability to transfer between stations, accuracy achievable by a process etc. also affects the selection of a forming process.
5. Size and shape of the starting blank: Starting shape of the blank whether it is a rod/wire or a tubular blank would affect the selection of the forming operations.

Unless these constraints are too restrictive, there is usually a large number of possible ways that a part can be formed. It is not practical to consider all the possible preform designs. In practice, the designer may consider a few possible designs and select one of these based upon his experience and experience of others involved in the project. He/she may also conduct simulation of metal flow and tool stresses using a finite element program such as NAGSIM.2D to decide on the sequence.

5.1.1 Approach to Find Possible Progression Designs

The approach employed in NAGFORM is based on a combination of Forming Rules and Analysis. Analysis is used where possible and practical to employ. Rules are used where analysis is not easily applied, is too time consuming, requires input data that is not available or requires analytical background not easily found in a designer. In NAGFORM, analysis is used to **estimate** the loads and strains during forming.

Because not all of the input information required for accurate analysis is available up-front, assumptions are made to simplify the analytical calculations which provide only **estimates** of the loads and strains.

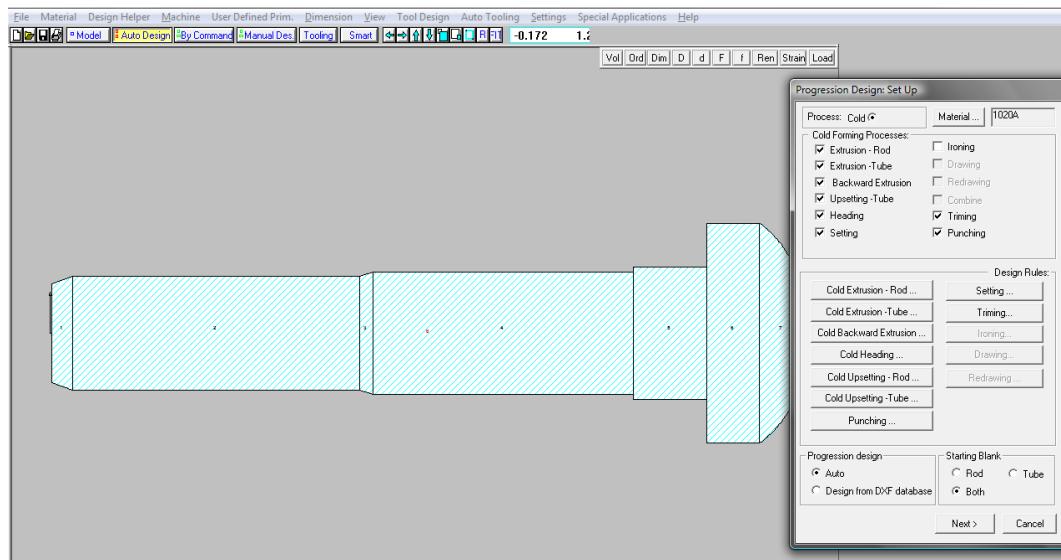
At the present time, NAGFORM considers parts or preforms that are axisymmetric or nearly axisymmetric. The processes considered are cold and hot forming processes **though no analytical analysis of temperatures and heat transfer is performed**. Temperature influences the design in two ways. Firstly the temperature affects the flow stress of the material used to calculate the forming loads and secondly, the rules used to form are different for forming hot compared to forming cold by the same process. Thus, within the framework of a rule-based approach, by suitably modifying the rules, hot forming processes are analyzed.

5.1.2 Processes

The various processes considered in NAGFORM are:

1. Forward extrusion of solid sections
2. Backward extrusion of solid sections
3. Forward extrusion of tubular sections
4. Backward extrusion of tubular sections
5. Upsetting of solid sections
6. Upsetting of tubular sections
7. Cold heading of wire
8. Punching
9. Setting- cold

10. Trimming
11. Punching
12. Ironing

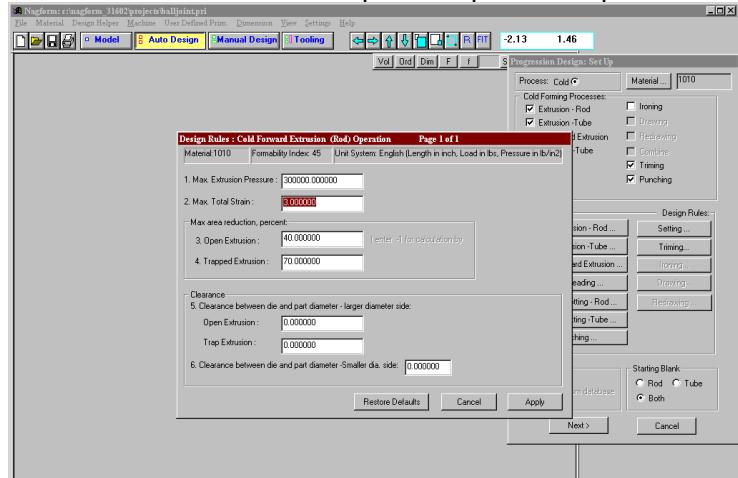


5.1.3 Process Rules and Constraints

With the objective that it should be possible to change the forming rules easily, rules and constraints are written separately for each of the above processes. In a new file, default rules are written automatically that can be easily changed by the user. Next figure shows the dialog box for the rules and constraints of the forming process of forward cold extrusion of solid sections. For this process, limits are set for:

1. Maximum extrusion pressure.
2. Maximum strain.
3. Maximum area reduction for open extrusion
4. Maximum area reduction for trapped extrusion.

5. Clearances between the die and the part for open and trap extrusion.

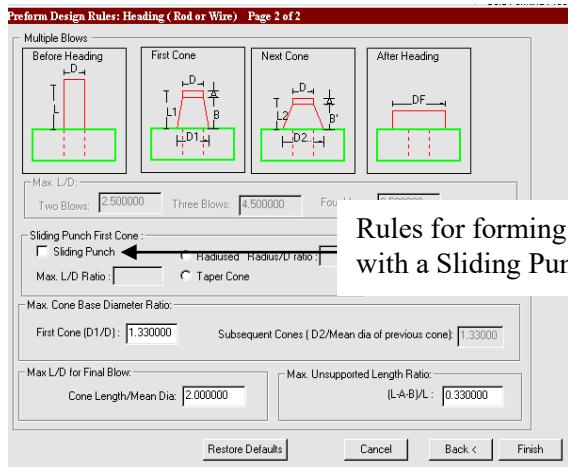
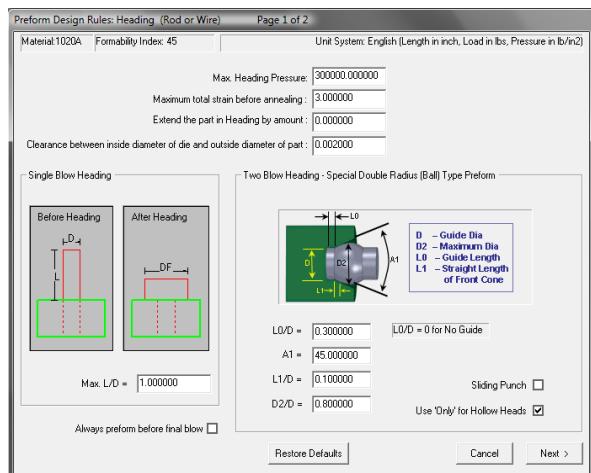


Similar to the dialog box shown here, there are dialog boxes for other processes. By changing the input values, the designer can obtain a preform design according to his/her rules.

In sequence design, forming processes of ‘Heading using a Sliding Punch’ and ‘Crash Trimming’ to form polygonal shapes have been added recently. The rules to define the limits of these two processes are included in the design rules.

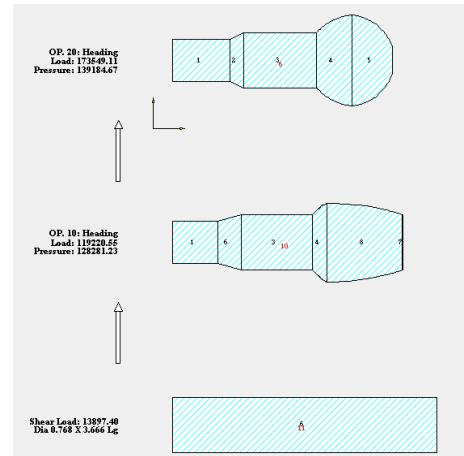
‘Heading’ Design Rules:

Rules are defined in two pages as shown below. A selection namely ‘Always preform before final blow’ is added in the rules to define user’s intent to preform the part before final heading blow even when the L/D ratio is below the limit for one blow heading. This is to reflect the actual practice of preforming before heading to obtain certain part shapes.



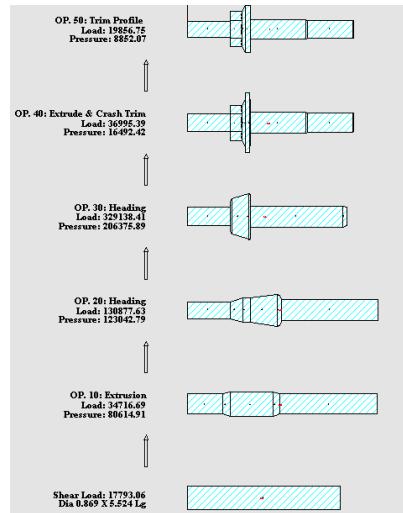
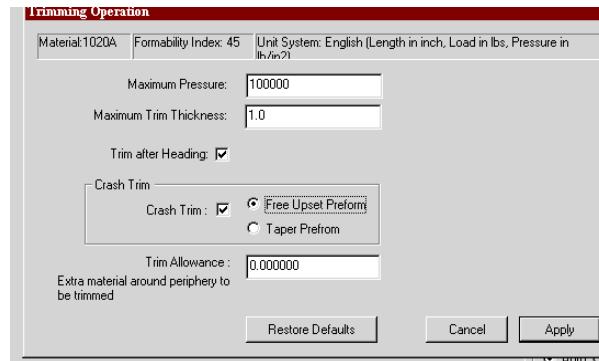
Rules for forming with a Sliding Punch

In heading design, the practice of gathering more material using a sliding punch is now included. To define the limits of this process, its ‘Rules’ have been added in the ‘Heading’ process rules. If the user selects this process, he/she needs to enter the max L/D ratio for two blow heading (with sliding punch) and the shape of the preform. If a convex preform shape is selected, the ratio of (Radius/Wire Dia.) needs to be specified. An example of a design obtained using a radiusued sliding punch is shown.



Trimming Rules:

The process of ‘Crash Trimming’ to obtain well defined polygonal shapes such as a ‘Hex’ or 12 point with flange has been added recently in the sequence design. The rules for crash trimming are included in the Trimming rules as shown.

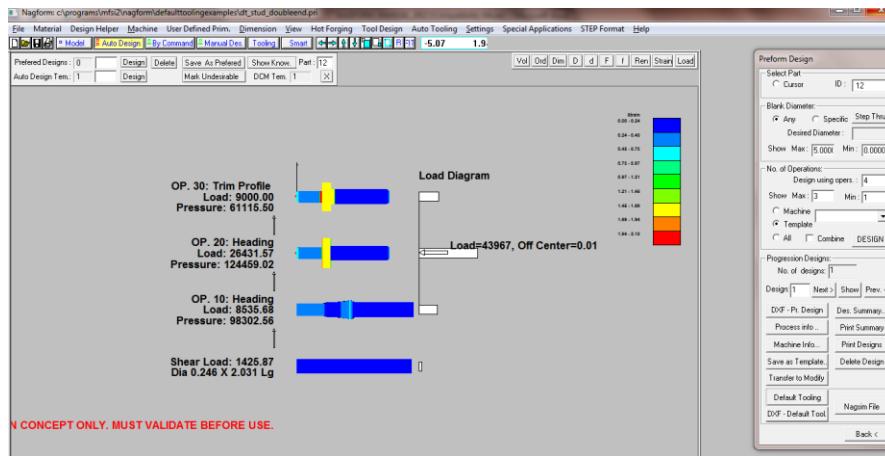


To obtain designs using ‘Crash Trimming’ process, the designer must select this option in the trimming rules. Also the type of preform needs to be selected. NAGFORM can design Crash Trimming process using a taper preform or a free flow type preform as done in actual practice. The design shown is an example of sequence design using crash trimming process

5.1.4 Determination of Progression Designs – Auto Design

Determination of preforms depends upon the desired shape and size of the starting blank. For tubular parts, the starting blank can be a solid rod or a tube. For solid parts, the starting blank can only be a rod. The designer may prefer to use a particular diameter rod or a tube of a particular inside and outside diameters. In NAGFORM, the designer can specify the desired type and size of the starting blank.

Starting from the part shape, NAGFORM determine which processes would yield the part shape starting from the blank. For these processes, analytical and geometric analyses are performed to determine if the processes would satisfy the rules and constraints set for that process by the designer. If the processes are feasible, a design is obtained. Normally, a number of alternative forming sequences that satisfy the constraints are determined.



To determine progression designs:

1. Enter Part ID. There is more than one part, enter ID of the part to be designed.
2. Choose 'Any' or 'Specific' wire diameter selection. For 'Specific' selection, enter the wire diameter. Note the wire diameter that the program determines will be slightly less than this diameter as it includes clearances at various stations. Enter range of diameters within which you want to see the designs.
3. Enter range of number of stations (specify max. and min.). 'Design Using Oper' should be two (2) more than the maximum number of stations.
4. Select 'Combine' option.
5. Select 'All' or Template or Machine to specify whether you want all designs or design according to the template or design for a specific machine. If you choose 'Machine', you must select the machine type from the Drop Down list box.
6. Press 'DESIGN'.
7. Bottom half of the dialog box would now appear showing you the number of designs and other options to look at the results.

5.2: Optimum Progression Design

The next step in the preform design process is to select the optimum progression design out of the possible design solutions. This selection depends upon many considerations including:

1. Equipment requirement and availability
2. Number of operations required
3. Robustness of the forming operations
4. Strains in forming. It is of concern specially in cold forming. The extent of deformation must be within limits of formability of the part material
5. Number of annealing steps required if any
6. Cost
7. Past experiences

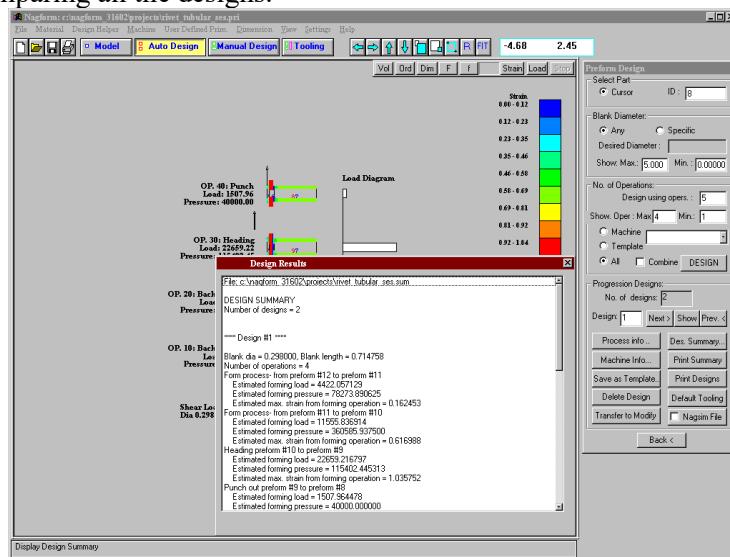
Some of these considerations are beyond the scope of NAGFORM. The role of NAGFORM in this selection process is to make available all the information generated from the analyses to the designer so

he or she can make the selection. The pertinent information is provided in form of (1) a design summary and (2) machine selection summary.

5.2.1 Design Summary & Machine Selection Summary

The figure below shows a design summary provided by the NAGFORM program after completion of the preform design. The design summary provides a summary of the results of possible designs. It includes the following information:

- 1 . Number of operations
- 2 . For each operation, type of operation, maximum load, maximum strain and maximum strain differential.
- 3 . A table comparing all the designs.



Machine selection summary gives a summary of machines that can be used for each of the alternative designs. The details are given in section ‘Machine Database and Selection’.

5.2.2 Progression Designs in DXF Format

NAGFORM can output the forming sequences in DXF format. The dimensions are not displayed but the preforms are drawn to scale. The designer can use CAD programs such as AutoCad, Pro-E etc to dimension the progression sequence.

To obtain a DXF output of sequence designs, you must first select this desired output in ‘Design Settings’ dialog box. Choose ‘Settings’ in main menu and then ‘Design’. The ‘Design Settings’ dialog box would appear. Select ‘Write DXF Design File’ and click on ‘Ok’. Now design again. A DXF file of the designs would be created by NAGFORM.

5.2.3 Transfer A Auto Design to Manual Design Module for Modification

To transfer a design from ‘Auto Design’ to Manual design module, first display this design in the graphics area when you are in the ‘Auto Design’ module. Press the button ‘Transfer to Modify’ in the ‘Preform Design’ dialog box. A copy of the sequence design being displayed would be created and added to the manual designs.

In Manual Design Module, this copy can be modified as required using the modeling capabilities of the program. Please note that the changes made in the ‘Manual Design’ module are not analyzed in any way. Those changes are simply done by the program as directed by the user. No calculation of loads or pressures or checks against the forming rules is done.

5.2.4 Removal of Similar Sequence Designs

NAGFORM ‘Auto Design’ sometimes gives designs that are only slightly different. When there are too many designs, it becomes difficult to go through all the designs. To alleviate this problem, a selection ‘Remove designs similar above percent similarity xxx’ has been added in the ‘Settings’ – Design menu dialog box. To remove similar designs, select this option. A 90 percent similarity limit is a reasonable value to use for removing similar designs. When this selection is made, NAGFORM would compare the designs for similarity and if the next design is similar within the specified similarity limit, that design would be removed.

5.2.5 Only Trimmed Designs

When trimming after heading is selected, ‘Auto Design’ creates designs with the trimming process and without the trimming process. Because the designs without trimming require less number of operations, most of the designs obtained are without trimming and only a few with trimming are created. If the user is interested only in sequence designs with trimming, it becomes difficult and cumbersome to use ‘Auto Design’ and ‘Design Helper’.

A selection ‘Show only Designs with Trim Operation’ has been added in the ‘Settings’ – Design menu selection dialog box. Select this if you are interested in trimmed designs only. NAGFORM would remove all designs that do not include a trimming operation. If you do not get any design for the specified number of operations, you can now go to ‘Design Helper’ to assist you in getting designs with trimming.

5.2.6 Design Helper

If no designs are obtained or NAGFORM gives designs with more number of operations than desired, you can go to the ‘Design Helper’ for help. ‘Design Helper’ is explained in the next section.

5.2.7 Template Design

When starting a new file with an existing template, the template sequence is written in your file. If ‘Template’ button is selected in place of ‘All’ or ‘Machine’, the program would attempt to design the current part according to the template design sequence. If all rules are obeyed, the program would come up with a single progression design for the current part as per the template design. If any of the rules can not be satisfied, no design would be given by the program.

5.2.8 Automatic Designs for A Specific Machine

Another enhancement made in current version of ‘NAGFORM’ is to generate progression designs for specific machines. In the ‘Auto Design’, the user can select to design for any one of the following:

- (a) For a specific machine
- (b) According to the template
- (c) Obtain ‘All’ designs

When the ‘Machine’ selection is made, the user must select one of the machines listed in the Drop-Down list box. When a particular type of machine is selected, the program generates only those designs that can be done on the specified type of machine.



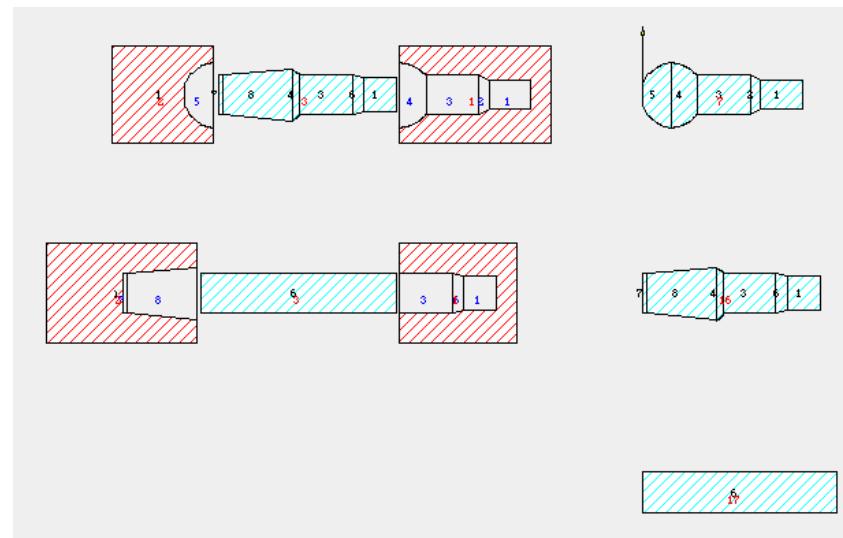
5.3 Creation of ‘Default Tooling’ in ‘Auto Design’ – In Development

In ‘Auto Design’, we have initiated the procedure for creating simple tooling for the sequences generated by the NAGFORM program. The purpose of this simple tooling drawn around the forming sequence is to:

- Show how the part tooling may need to be to form the part.
- Use this tooling in simulation of the forming process in NAGSIM.2D program.

This simple tooling is termed ‘Default Tooling’ here. Please note that this feature will be gradually completed over a period of one year or so.

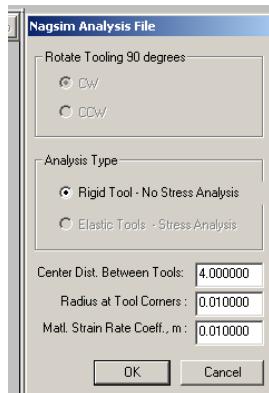
To create default tooling for a sequence, press the ‘Default Tooling’ button in ‘Preform Design’ dialog box. If the program can, it would generate default tooling as shown below for a part



To obtain a DXF drawing of the ‘Default Tooling’, select ‘DXF OUT’ before you press ‘Default Tooling’ button. A DXF drawing of the sequence and tools will be created.

5.4 Automatic Creation of Analysis files for Simulation of Forming Sequence using NAGSIM.2D – In Development

The procedure of automatic creation of the analysis file for simulation for any forming sequence has been started. It would however take a year or more before NAGSIM.2D analysis files can be generated automatically for most of the sequence designs in NAGFORM. This task requires the completion of the previous task of creating default tooling automatically. To create a NAGSIM.2D model file (.cad), select ‘Nagsim File’ and then click on ‘Default Tooling’ button.



The dialog box shown above would appear. Note that you may have to design first to ‘enable’ the ‘Nagsim File’ selection. The ‘Default Tools’ drawn have sharp corners. If you want to fit radii at the

corners and fillets, enter the radius value. Note that same radius would be fitted by the program to all corners and fillets. The material properties in NAGFORM do not include strain rate sensitivity index, m , for the material. You need to specify ' m '. Also enter the distance you want to keep between tools of adjacent stations.

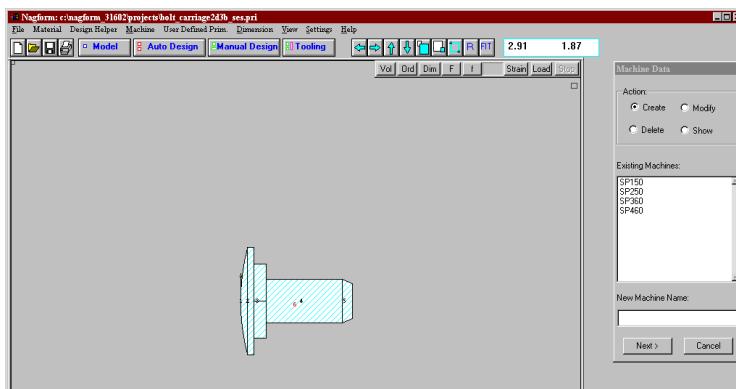
NAGFORM would modify the default tool drawings to have right half only, would fit the radii and write the complete model file (.cad) with all the information.

Note that this automatic generation of the CAD file of NAGSIM.2D program is in preliminary stage of development. Currently, only for simple progressions with maximum three forming stations and having only a punch and a die at each station with no pins, NAGSIM.2D file may be created.

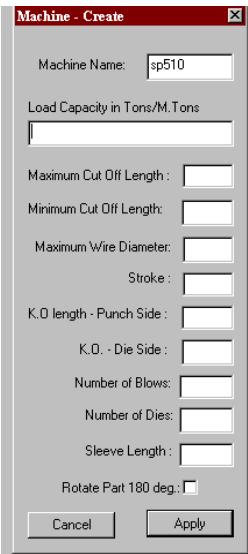
Presently you can use the DXF drawing of 'Default Tooling' created by NAGFORM to create the tool drawings for NAGSIM.2D program. Modify the DXF drawing to draw only the right half of the parts and put each part-geometry in a layer with the same name as the part name. You can then use it along with the NAGSIM.2D templates to create the analysis file for simulation easily and quickly.

5.5 Machine Database and Selection

The objective is to help the user to select machines that may be used for forming a part by the forming sequences created in Automatic Design Module. As a first step, the User must create a database of available machines. To create and maintain a database of available machines, the form shown below is provided. This form can be accessed from the Main Menu. Click on the 'Machine' title and then 'Set up' and the form shown below will appear.



To add a machine to the database, select 'Create' radio button, then enter the name of the machine and press 'Next'. The machine data form shown below will appear. Fill in all the information on the machine and then press 'Apply'. This machine data will be added to the existing Machine Database. The data on machines can be modified and also deleted in needed.

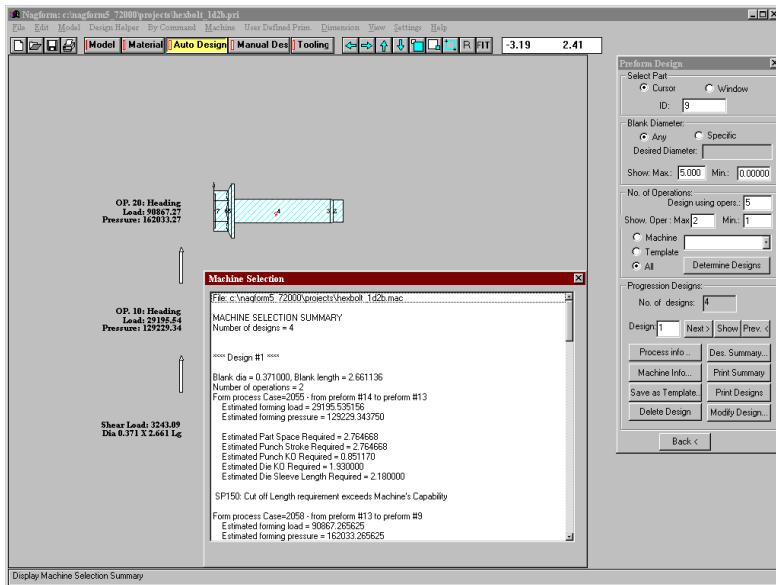


To determine the machines that may be suitable, in the ‘Automatic Design Module’, go to ‘Setting’ on Main Menu, then Design selection. When the ‘Settings’ dialog box appear, select the option ‘Machine Selection Summary’ and press ‘OK’.

Now when the program determines the designs, it would also create a ‘Machine Selection Summary’. This summary gives the information on the machines that may be suitable for a particular design. It also gives the reasons why the other machines in the database can not be used. You can see this summary by pressing ‘Machine Info..’ button in ‘Preform Design’ dialog box.

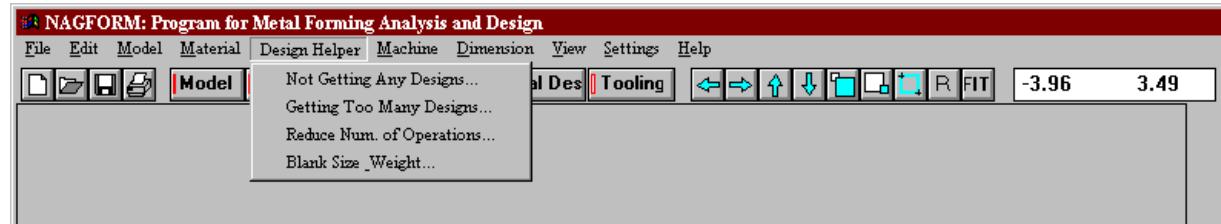
Please note that this machine selection is based upon certain assumptions made by the program regarding the way the part would be formed. It is quite possible for a good designer to come up with alternative arrangement of tooling to the make the part differently and use a machine this was considered unsuitable by the program.

Use of machine selection by the program should be considered only a ‘rough’ estimation.



6. Advanced Design Helper

In NAGFORM, the main menu has a separate title ‘Design Helper’ as shown below.

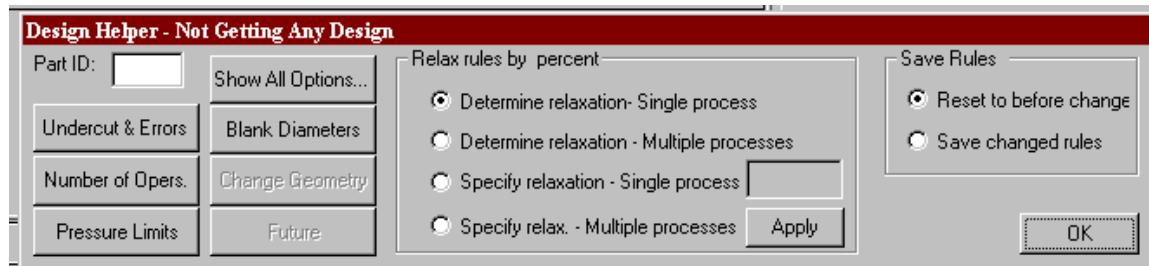


The purpose of the ‘Design Helper’ is to assist the user in automatic design of the forming progressions. It is a diagnostics tool as well. Specifically, the Design Helper can be used in the following situations:

1. When NAGFORM program does not give any design.
2. When number of designs are too many
3. When Designs with less number of operations than those being given are desired
4. To obtain blank dimensions and weight

6.1. Menu Selection ‘Not Getting Any Design’

This selection can only be made when the user is in ‘Auto Design’ module. The above menu selection will bring up the form (dialog box) shown below.



Using this dialog box, the designer can quickly determine if the reason for not obtaining a design is any of the following:

1. Part has undercut
2. Less number of operations
3. Proper blank diameter is not selected
4. Limits set on loads and pressures in the ‘Rules’ dialog box are being exceeded
5. Any rule or any combination of rules need to be changed (relaxed)

In this ‘Design Helper’, the program automatically iterates for different values of the design parameters to look for a solution. When you click on ‘Undercut and Errors’ button, the program checks to see if the part has any undercut that does not allow any design. It would also check for other errors such as wrong setting of maximum and minimum wire diameters, number of operations etc.

When ‘Number of Operations’ button is pressed, the program would iterate automatically, up to ten operations, to determine if designs are obtained with higher number of operations.

When ‘Pressure Limits’ button is chosen, the program determines if raising the pressure limits up to twice the limit values would yield a design. It would find the least change in pressure limits required to obtain a

design if one is obtained.

When ‘Blank Diameters’ button is clicked, the program iterates to find a wire diameter at which a design may be found.

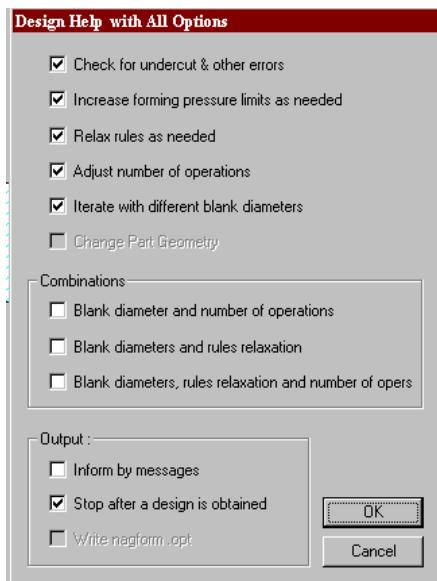
When ‘Relax Rules’ options are selected, the program would go through each rule and all rules of a given process to determine if relaxing of the rule(s) is required. The program would point out the new rule limits at which design(s) are obtained. The options here are:

1. Program to change rules of a single process such as extrusion or heading
2. Program to change rules of more than one process at a time
3. User to specify the percent change in rules for any single process
4. User to specify the percent change in rules of multiple processes.

In all of the above cases, the program designs a number of times with different parameters. It may take from a minute to a few hours of computer time to get an answer.

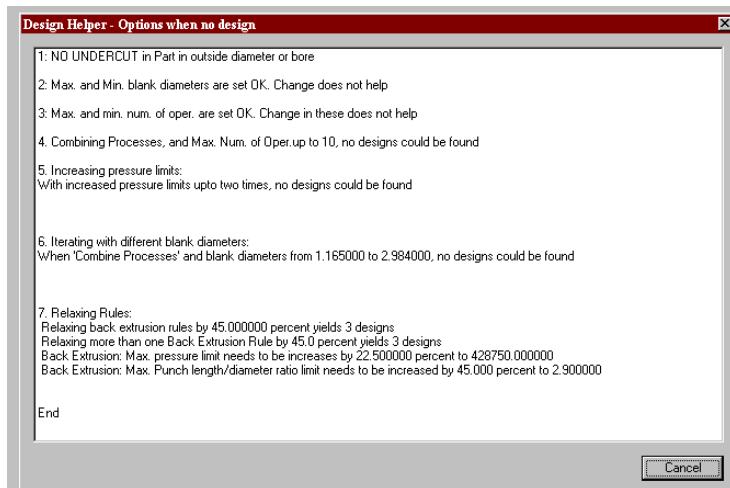
Please note that it is not guaranteed that the program will find a design. Within the limitation of the process parameters, there may not exist any design or the logic of the program can not find the solution. The advanced Design Helper is however looking at a much broader range of process parameters to find a design.

You can also ask the ‘Design Helper’ to look for designs for a number of options. For this, press ‘Show All Options’ button. This would open the dialog box shown below.



In this dialog box, you can select the possible options that the program should go through to search for a design. Note that some of the options take considerable time to execute. It may form one minutes to 5 hours to complete the search for a design if you select all of the options. So you may want to go through one or two options at a time.

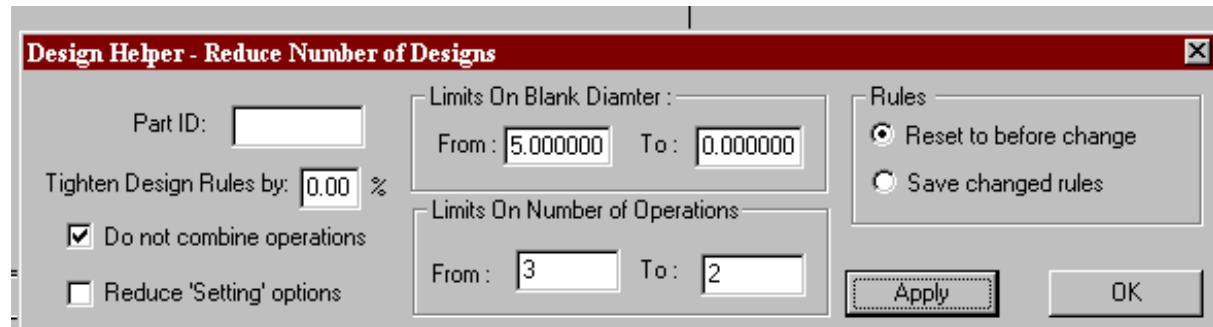
If you are interested in designs obtained with different options, deselect the ‘Stop after a design is obtained’ check box. The program would not stop after it finds the first solution. It would continue investigating till all the selected options have been searched. In the end, the design information is presented in a file named ‘nagform.opt’. The contents of this file are also displayed in a dialog box as shown below.



If a design is obtained and you wish to accept the changes in rules, select ‘Save changed rules’ before exiting out of the Design Helper.

6.2. Menu Selection ‘Getting Too Many Designs’

When this selection is made, the following dialog box comes up.



This form would restrict designs :

- To desired blank diameter within the specified range. Only those designs that give a blank diameter within the specified diameter range would be found.
- To desired number of operations.
- By certain selections such as ‘Not combining processes’ and reduced ‘Setting’ options that reduce the number of designs
- By restricting the design rules further to reduce the number of designs

Another way to reduce number of designs is by eliminating similar designs. In ‘Settings’ menu, when you choose ‘Design’, a ‘Design Settings’ dialog box appears. There is a selection ‘Remove Designs Similar Above Percent Similarity’. Choose this selection to eliminate duplicate designs.

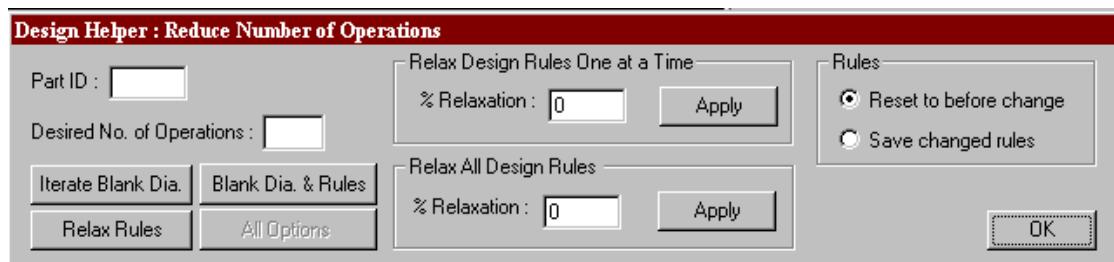
6.3. Menu Selection ‘Reduce Number of Operations’

The purpose of this part of ‘Design Helper’ is to help the designer reduce the number of operations to form a part.

You can also use the first option of ‘Not Getting Any Design’ to find designs with less number of operations. Run Auto Design specifying the desired number of operations. When you do not get any design, go to ‘Design Helper’ option of ‘Not Getting Any Design’ to find designs within the specified number of operations.

To reduce number of operations, enter the desired number of operations and select the option. You can investigate through:

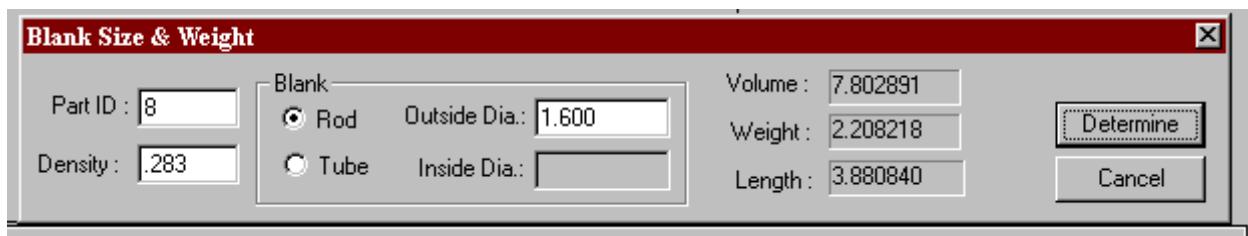
- Selection of a blank diameter
- Relaxation of design rules
- Combination of a blank diameter and rules relaxation.



Keep in mind that the program determines the solution by changing blank diameter and/or rules incrementally. It can take considerable time for the program to determine a solution. As before, you can also specify the percent relaxation of rules to determine a solution.

The following helps to reduce the number of forming operations:

- ‘Combine processes’ selection
- Design with higher number of operations than the actual number required. The program has a better chance of reducing number of operations by combining and recombining operations
- Relaxing rules which make the difference. Through iterative process, the program would determine the changes in rules required to get designs with desired number of operations



6.4. Menu Selection ‘Blank Size & Weight’

This form is designed to obtain the dimensions of the blank and its weight.

7. Design Example

In this section, a simple part is designed to help the user understand the operation of ‘NAGFORM’.

Cold Forming of a Ball Stud:

Problem:

Determine design sequence for cold forming a ball stud from a cylindrical blank.

Inputs:

Final part geometry

Part material: 1020 annealed

Forming temperature: Ambient

Starting blank: Cylindrical blank

Machines available: Cold former / Cold forging press

Rules: Rules for cold forming operations

Constraints: Maximum load and pressure for each cold forming process.

Outputs:

Number of possible process design sequences

For each preform design sequence:

- . Total number of forming operations required
- . Forming process at each operation
- . Shape of the part (preform) at each operation
- . Starting blank diameter and length
- . Load estimates for each operation
- . Estimates of strain distribution in the part at each operation

Comparison of possible design sequences

Forming sequences in DXF format.

Machine selection summary

Design using NAGFORM

The preform-design process using NAGFORM consists of the following steps:

1. Open a new project file.
2. Construct a geometric model of the part
3. Select part material.
4. Set rule limits for the various forming processes.
5. Perform automatic progression design.
6. Select a design or a number of designs from the choices presented by NAGFORM
7. Modify design(s) if needed

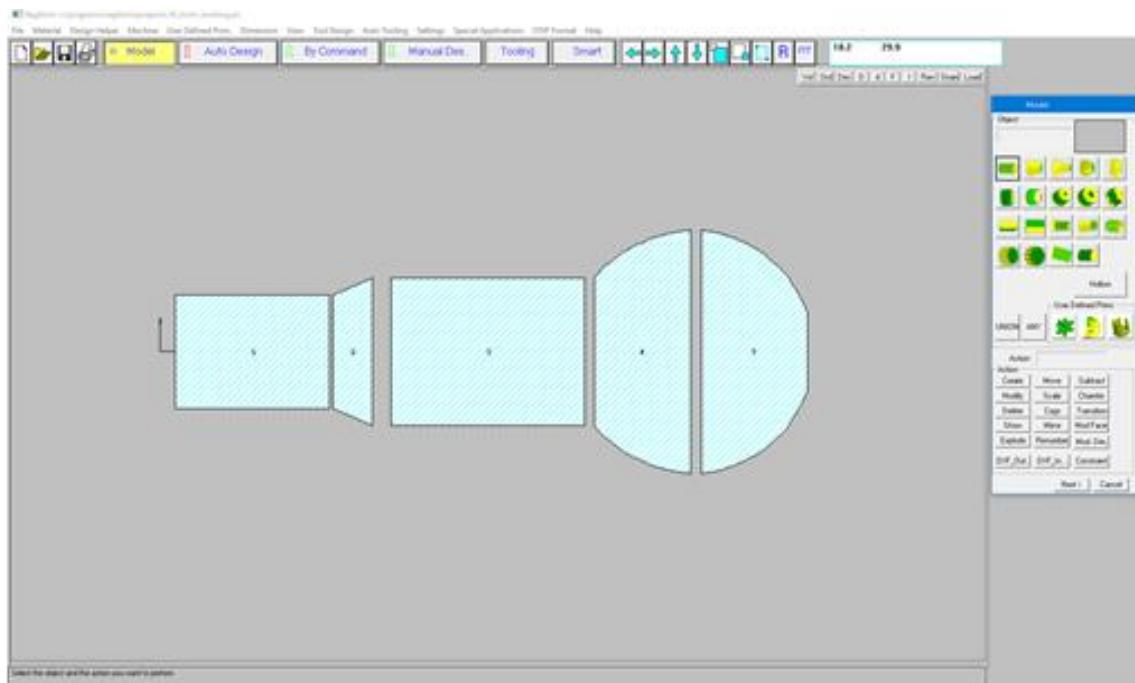
Open a new file:

In this step, the designer opens a ‘New’ geometric model file (with .pri) under a suitable name. This name is used throughout the design process. All results are stored in this file.

Click on the New File icon or select ‘New File’ from the ‘File’ menu. A dialog box will appear on the screen. This dialog box shows the current directory. Enter the name of the file without .pri suffix and press OK. In this example, ‘Balljoint’ is entered as the name. The file name including the path shows up the title block of the main window.

Construct a Geometric Model

In this step, a geometric model of the part is built. The geometry of this part consists of ‘Rod’, ‘Taper’ and ‘Corner’ primitives as shown in the figure below. First create the primitives and then use a ‘Union’ action to join these primitives together to form a ‘Union’.



Create Primitives

1. Click on the MODEL button to open the Object -Action dialog box.
2. Click on the 'R' object button. A drawing of a ‘Rod’ primitive will appear in the dialog box. Now click Create button and then NEXT button.
3. This Object-Action dialog box closes and another dialog box ‘Create Rod’ opens. Create the rod primitive using the full or the short format.
4. Press BACK button to return to the previous dialog box for selecting the next object and action.
5. Repeat the procedure to create other primitives.

Join Primitives (Create a ‘Union’)

1. In the Object-Action dialog box, choose ‘Union’ and ‘Create’.
2. ‘Create Union’ dialog box will appear. The objects to be joined should be included in the list first. The order in which these objects appear in the list is the order in which they will be joined to form a Union .
3. Type 1:5 in the edit box to include primitives with IDs 1 to 5. Press 'Enter'. The IDs 1 to 5 will appear in the object list box.
4. Press ‘Union’. The objects in the graphics window will reappear as a single ‘Union’ object. Note that union ID is written in red.

Note: If the ‘Union’ is not what we want, this operation can be undone by pressing ‘Undo’ button. Also any object in the list can be removed by clicking the object in the list and then pressing ‘Remove’ . By using ‘Remove’ and ‘Add operations’, the list can be modified as desired.

Select Part Material

The material to be selected should be in the database before it can be selected. If it is not, then it must be ‘created’ first in the database. Material for the part is ‘1020 annealed’ which is available in the database. This material can be selected using the ‘Material’ button and its associated dialog box or we can go to the ‘Design’ step and select the material there, which is a simpler approach. We will use the simpler approach of selecting the material in the ‘Design’ step.

1. Click on the ‘Design’ button in the main window. The 'Design Set-Up' dialog box shown in the figure below will appear. This box has a ‘Material...’ button at the top.
2. Click the ‘Material...’ button. A dialog box showing the list of materials in the material database will appear. Click on the ‘1020A’, click select and then ‘OK’ . This dialog box will close and the previous ‘Design Set-UP’ dialog box will reappear. The static test box next to the ‘Material...’ button will now show ‘1020A’.

Set Forming Rules for Various Operations

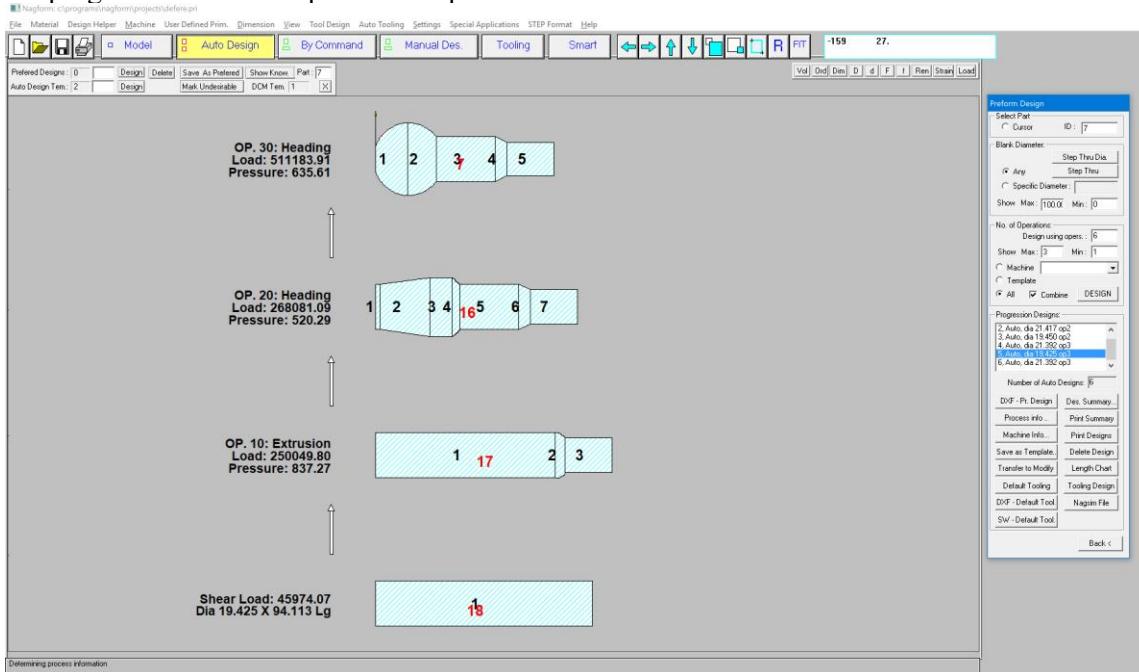
The next step in the design process is to set the design rules that should be used for progression design. Note that default rules are written automatically. Therefore the task here is to modify any rule if needed. To demonstrate how these rules can be changed, we would go through the exercise of changing the rules for the cold heading operations.

To set rules for ‘Heading’ operation, click on the ‘Heading’ button in the Rules portion of the Set-Up dialog box. A dialog box containing the rules will appear. The dialog box contains the default values of the various forming limits for heading in one step and in multiple steps. The designer should enter in the values he/she wants to use for design. After entering in the desired values, click Apply. Repeat the above step for other processes.

Perform Automatic Sequence Design

1. Set-up for design that includes selecting the operations and defining the rules was completed in the previous step. Another selection to be made here is the starting blank shape. For this part, it can only be a ‘Rod’ blank. Click on the ‘Rod’ to select this shape for the starting blank.
2. Click on the menu 'Setting - Design' which would open the 'Setting' dialog box. Check options (1) Write DXF Design File, (2) Write Design Summary and (3) Machine Selection Summary. Press OK.

3. After Design Set-up is completed, press OK button. This closes the Set-Up dialog box and opens the ‘Auto Design’ dialog box.
4. Click on the part to select it as the part to be designed.
5. Enter preferred size of the starting ‘Rod’ blank if needed. We have not used this feature.
6. Select maximum number of operations allowed. We have set it to be three (3). Set Minimum number of operations to be one (1) and Design using operations equal to five(5). Select ‘Combine’ to allow the program to combine operations if possible.



7. Press ‘DESIGN’ button. The determination of preforms starts and would continue till various designs are obtained. It should only a few minutes for the program to determine the possible designs.
8. The total number of designs is shown in the dialog box. The user can view any design by entering the design number or use ‘Next’ to look at the next design or ‘Prev’ to look at the previous design

To look at the design summary, press the ‘Design Summary’ button. A dialog box will appear with the summary of the designs. At the end of this design summary, a table presents the results of all designs in a tabular form. The user can delete any design by selecting ‘Delete Design’. Machine selection information can be viewed by clicking on ‘Machine Info’ button. Click on ‘Transfer to Modify’ if you want to take a copy of the design being shown to the ‘Manual Design Module’ for modification. Click on ‘Process info’ button to see information on reductions, loads etc. Click on ‘Default Tooling’ to view simple tooling drawn by the program.

10. The design process is complete. Results of this design are in three files. File with .pri suffix has part geometry, material data, global variables/settings, rules and the design information. The file with .sum suffix is a text file containing the design summary. File with .dxf suffix contains the preform design in DXF format.

8. Manual Design Module

8.1 Purpose

The purpose of this module is to allow the designer to design forming sequence according to his/her own concept. Various features are provided to assist the designer in constructing the forming sequence as quickly as possible.

There are two manual design options:

1. Design Manually
2. Design by Command

The first option allows the user to create a progression design manually using the modeling capabilities of the NAGFORM program. Commands of the user are not remembered by the program. The design is not checked for loads, pressures or forming rules.

The second option of ‘Design by Command’ is an interactive design process that is remembered by the program. The user defines its design intent through a question and answer session. The session file created can be used to design similar parts automatically by the program.

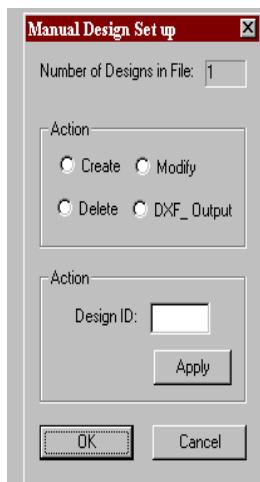
8.2 Design Manually

Modeling capabilities of NAGFORM used in designing manually are:

- Display of volumes of primitives.
- Replacing a primitive with another type of same volume.
- Adding a primitive to a union (preform).
- Changing length of a primitive to obtain specified volume.
- Create copies of any preform.
- Arranging the preforms in a specified order.
- Change dimensions of a preform.
- A number of manual designs can be created and stored in project file.
- Designer can display any manual design saved in project file.
- DXF output of any design can be obtained

8.2.1 Construction of a Manual Design

To construct a manual design, click on the 'Manual Des' Button on the desktop. Other buttons must be in 'Up' position. This will bring up a dialog box with the two options of manual design. Select ‘Design Manually’ and click ‘Ok’. The dialog box would appear.



As seen in the picture, the designer can:

- Create a new manual design
- Modify an existing design
- Delete a design stored in file
- Obtain DXF output of a design

Choosing any of the above and if required pressing the 'Apply' button would either perform the desired task or bring up the next dialog box.

For 'Create' and 'Modify' actions, the dialog box shown below will come up. This dialog box is also used for modifying a copy of the automatic (computer generated) design.

The functions of various buttons on this dialog box are as follows:

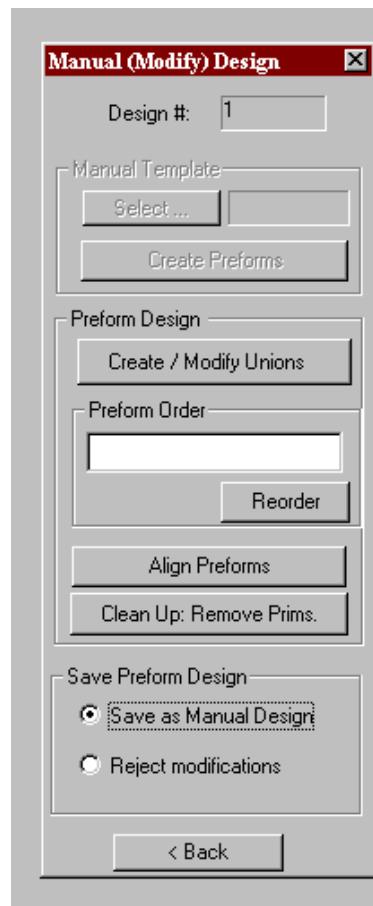
Create/Modify: Pressing this takes the program to 'Union Modification' dialog box where the designer can create copies, modify any preform using special features outlined above, to assist in the modification process.

Reorder: Arranges the preforms (unions) in the order specified by the designer in the edit box. For example, if the unions with IDs 1,30,12,5 are on the screen and the designer writes the order as 1,12,5,30, the program will place preform no. 1, then preform no. 12 and so on and would also align them.

Align: Pressing this button aligns the unions on the screen.

Clean Up: This deletes any primitives that are not used to create a union.

Save as Manual Design: If this radio button is selected, the design changes will be saved in the project file.



8.2.2 Preform Modification

The dialog box shown below is used extensively to modify a union (preform).

The functions of various controls in this dialog box are as follows:

Select: This is to select the union (preform) you intend to modify.

Change/Remove: To change or remove a primitive of the selected union, the user must first identify that primitive of the union by selecting it using ‘Cursor’, ‘Window’ or ID. ‘Change’ takes the control to ‘Primitive Modification’ dialog box. Pressing ‘Remove’ button removes the primitive from the union.

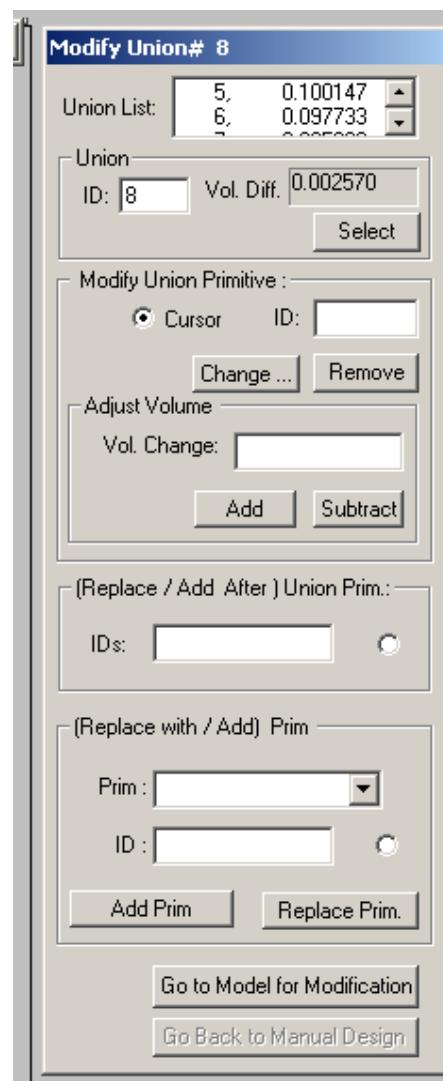
Vol Change /Add/Subtract: Enter volume to add or subtract from the prim selected above and then press Add or Subtract. The length of the union prim will be changed to change its volume.

Replace Comp: This allows any union component to be replaced by a default primitive of another type (or by a primitive created earlier). The default primitive will have the same volume as the primitive replaced. The default primitive type is selected from the drop down list box (or the ID of the prim created previously is entered in the edit box).The ID of union prim to be replaced is entered in the edit box above or it can be selected by clicking on the radio button and then on the prim. Pressing ‘Replace Prim’ would replace the union prim.

Add Prim: is used to add a default primitive (or a primitive that was created earlier) after the selected union primitive.

Go To Model: Takes you back to Object-Action dialog box of the Model

Go Back To Manual Design: Takes control back to the manual design dialog box shown above



8.2.3 Modification of Auto Design

Any computer-generated forming progression design can be modified in the manual design. As the modified design is saved in the project file as a manual design, it can be viewed at any time using the manual design module.

To modify an automatic design, you need to transfer a copy of the design to the manual design module. This transfer is done when you are in the Auto-Design module. Once a copy of the design exists in the Manual Design Module, select that design to modify it as discussed above.

8.3 Design by Command - Manual Design Module

Purpose:

1. User can create his/her own design from scratch in a few minutes through commands to the program.
2. The progression can be saved as a template (session file). Progression for similar parts can be obtained in a couple of minutes using the template (session file).
3. Process information such as estimated forming load, pressure, strain, percent reduction etc. are generated for User's design.
4. The Program warns of any violation of forming rules in User's design.
5. There is no need to do any volume or other calculations.
6. DXF (AutoCad) file of forming progression is generated within a minute.

8.3.1 How does it work:

1. In an interactive session, the program determines the design intent of the User through a series of questions and answers, and creates the design.
2. The program presents a multiple-choice question and the User makes a selection. The subsequent question by the program depends upon the response by the User in the previous question. At times and as needed, the program asks for specific design dimensions to finalize the design.
3. This interactive session is recorded in a 'Session' file.
4. This session file can be edited like a text file to change the design.
5. For a similar part, the program can 'Run' through the session file to create a progression design. The User can also 'Step' through the session changing the design parameter values (User's response) at any step.

8.3.2 Implementation:

'Design by Command' is implemented as a part of 'Manual Design' Module. When the User presses the 'Manual Design' Button , a dialog box appears.



To create a sequence design by Command, select the radio button 'Design by Command' and then press 'OK' button. This dialog box will close and another dialog box ' Design by Command Set Up' shown below would appear.

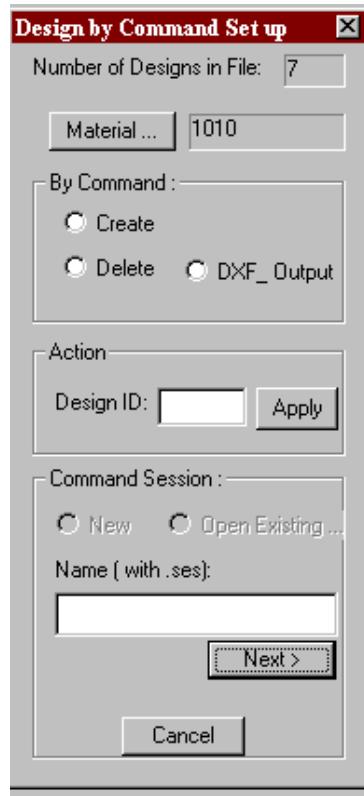
8.3.3 'Design by Command Set Up' Dialog Box:

Explanation for various items on this dialog box is as follows:

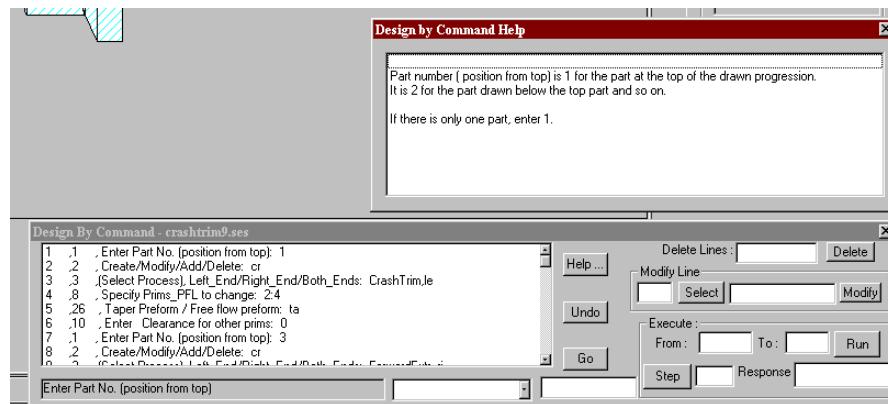
- Number of Designs in File - Shows number of manual designs in the project file.
- Material - Click on this button to select the material if not selected already. The selected material is shown.
- Create - Select to create a new design.
- Delete - Select to delete an existing manual design. To delete, enter the design ID in the edit box and press 'Apply'
- DXF_Output - To create a DXF file of any design, select this radio button. Enter Design ID in edit box and press 'Apply'. A DXF file of the design will be created.
- New - Select this when you want to create a new design using a new session file. Enter name of

the session file and press ‘Next’ button. A new session file will be created and the ‘Design by Command’ dialog box would open at the bottom of the desktop.

- Open Existing - Select this when you want to create a new design using an existing session file. A Windows ‘Open File’ dialog box would open. Select the session file you want to use. Press ‘Next’ and ‘Design by Command’ dialog box would open at the bottom of the desktop.



8.3.4 Design By Command Dialog Box:



The interactive session of questions and answers between the program and the User is recorded in the session file using this dialog box. Explanation for various items shown on this dialog box is as follows:

- Dialog Box Title: The session file name appears here. This file is saved in ‘Session’ sub directory of the ‘NAGFORM’ program.

- File List Box: The session file consists of ‘Command’ lines. This list box shows the command lines of the session file as it is written or changed. Each command line contains (a) the line number, (b) an internal code number used by the program, (c) question or instructions to the user, and (d) User’s response.
- Delete Lines: Enter numbers of the lines to be deleted from the session file. To delete one line, enter the line number and press ‘Delete’ button. To delete multiple lines, uses colon between the start line and the end line number. Note ‘comma’ between line numbers is not allowed or interpreted anywhere in this dialog box.
- Modify Line: Enter line number to modify and press ‘Select’ button. The existing ‘response’ of the User would appear in the edit box. You can change this response by entering new response and pressing ‘Modify’ button. The line in the session file will be changed by this action.
- Execute Group Box: The items in this group allow the user to ‘Step’ through a set of command lines or whole session, or ‘Run’ through these. The difference between the two modes is that in ‘Step’ mode, the command lines are executed one by one and the user has the option to change the response. In ‘Run’ mode, the program executes the command lines from start line to end line without stopping. The user does not have the opportunity to change any command.
- Run mode: Enter the ‘Start/From’ line number, ‘To/End’ line number and press ‘Run’ button. These command lines will be executed.
- Step Mode: Enter the ‘Start/From’ line number and the ‘To/End’ line number. Enter the line number to start the Step mode. Pressing ‘Step’ button once would show the program question/instruction and pressing it again would show the response. Change the response if you want to before pressing the ‘Step’ button again. Note that changing the response here does not change the file. To change the response in the session file, use ‘Modify Line’.
- Go: Enter the response in the edit box below this button and press this button to execute the action. The command line is written in the session file.
- Undo: You can undo the last command by pressing this button. Repeatedly press it to remove/undo a number of commands from the end.

‘Design by Command’ procedure for creating forming sequence manually now has a ‘Help’ section. If at any time doing the interaction with the program, you are not clear about the question or the response you should give, press the ‘Help’ button in the ‘Design by Command’ dialog box. A ‘Help’ dialog box as shown below would appear and help information would be displayed.

8.3.5 Example Session Files

In NAGFORM program files, there is a directory called ‘Session’. This directory has a number of session files. These session files are for the parts that are in the ‘Template’ directory. As an example, start a new file bringing in any template say ‘Ballstud2_t2.pri’. This would bring a copy of the ‘Ball Stud’. Modify this part to have a longer shank. Then go to ‘Design by Command’ to create a sequence design for this part. The session to use is ‘ballstude2_t2.ses’. Follow the procedure discussed above to ‘Run’ through and ‘Step’ through the session file to create forming sequences.

For another part started from the Ball Stud template, write your own ‘New’ session file following the ‘ballstud2_t2.ses’ file. Note that all the session files are text files, so you can read them in ‘WordPad’ program and print them.

8.4 Database for Representative Parts – for ‘Design by Command’

In NAGFORM, there are two methods to obtain sequence design, namely

- Auto Design
- Design by Command

‘Design by Command’ is a manual design procedure that is automated through a ‘Session’ recorded in a session file (.ses). Users can utilize ‘Design by Command’ procedure to create their own sequence design and then use the created Session file to design similar parts.

We have created a number of Session files to illustrate the use of ‘Design by Command’ procedure. These files are stored in ‘Session’ subdirectory of the NAGFORM program files. A separate database for storing representative parts to be used with ‘Design by Command’ session files has been created. In this database, the representative parts for various session files created earlier have been added. The User can start from a representative part, and modify it to the desired shape and dimensions in place of starting from scratch. Also if the number of primitives and general shape are kept same in modification, ‘Design by Command’ session file for the representative part can be used to get the sequence design for the new part.

In ‘New File’ dialog box, a ‘Design by Command’ button has been added as shown in Figure 1.



Figure 1

To bring in a representative part in the new file, click on this button. A dialog box shown in Figure 2 would appear. This dialog box shows all the representative parts and their session files that are available.

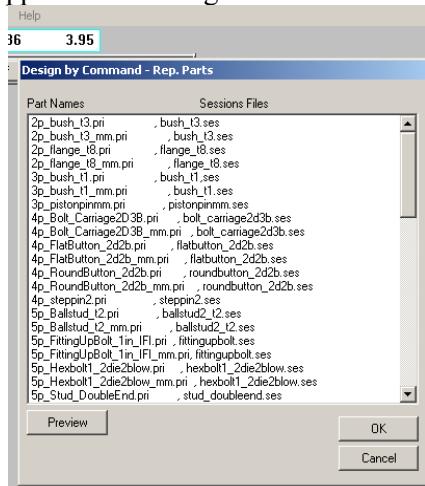


Figure 2

To preview any part, select the part in the list box and click Preview. To take this selection to ‘New File’

dialog box, click OK. This dialog box will close and the part name and the session file name would appear in the ‘New File’ dialog box. A copy of the part model will be brought into your new file.

This procedure of bringing in a representative part in a new file is similar to starting from a ‘Auto Design’ template. The difference is that no sequence design is written into the file when you use this database. You would need to go to ‘Design by Command’ dialog box and use the appropriate Session file to get a sequence design.

You may notice that the part names have xp_ or xpy_ in front. This is to indicate the total number of primitives and number of subtracted primitives in a part. For example, 6p1s_ in front of the name indicates that the part has six(6) primitives out of which one primitive is subtracted. The reason for specifying the primitives in the name is that a particular Session file can be used only if your new part has the same number of total and subtracted primitives for which the Session file was written. You cannot change the number of primitives when you modify the representative part to your new part. You can , in some cases, make a primitive insignificant by reducing its length to near zero (say .0001) but you can not delete it.

Example on Use Of ‘Design By Command’ Representative Part and Session File

Suppose you want to create a sequence design for the part shown in rendered view below. The part has a round with two flats. In the sequence design, the flats are trimmed from a round.

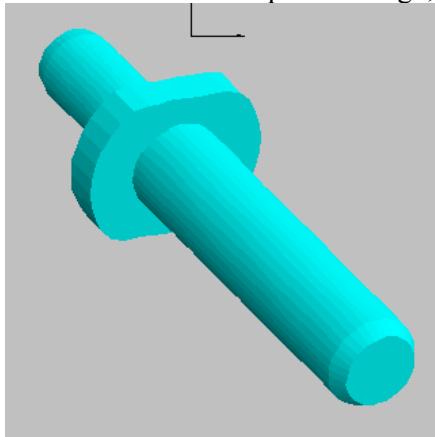


Figure 3

This part is similar to the Representative part ‘5p_Stud_DoubleEnd’ below. This representative part has a Hex that is trimmed. The session file for Design By Command is ‘stud_doubleend.ses’.

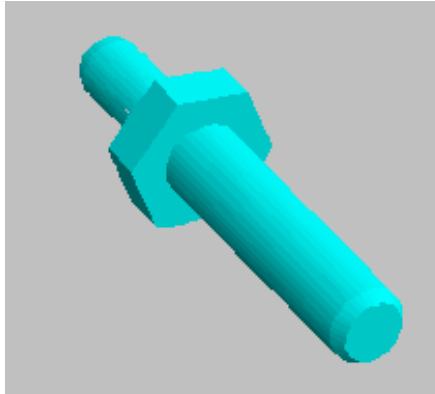


Figure 4

The procedure for getting a sequence design for part shown in Figure 3 using ‘Design By Command’

would be as follows:

1. Select ‘New’ from File menu. Dialog Box shown in Figure 1 would appear.
2. Click on ‘Design by Command’ button. This would bring up the dialog box shown in Figure 2.
3. Select ‘5p_Stud_DoubleEnd.pri’ in the list box. Click OK.
4. This dialog box would close and you would be back to ‘New File’ dialog box. The names of the representative part ‘5p_Stud_DoubleEnd.pri’ and session file ‘stud_doubleend.ses’ would be displayed in the ‘New File’ file dialog box. Enter name of the new file and select Units. Click OK.
5. The ‘New File’ dialog box will close and the representative part as shown in Figure 4 would appear on the screen.
6. Go to ‘Model’ and modify this representative part to the new part as seen in Figure 3. Note that you do not have to ‘Explode’ the union to replace the Hex prim with the ‘Rod with Flat’ prim. You can replace the primitives in ‘Modify Union’ dialog box without ‘Explode’. Modify the dimensions as needed. Make sure you do not delete any primitive.
7. Go to ‘Manual Design’ : Select ‘Design by Command’ Option.
8. Select ‘Create’ and then open the existing session file ‘stud_doubleend.ses’.

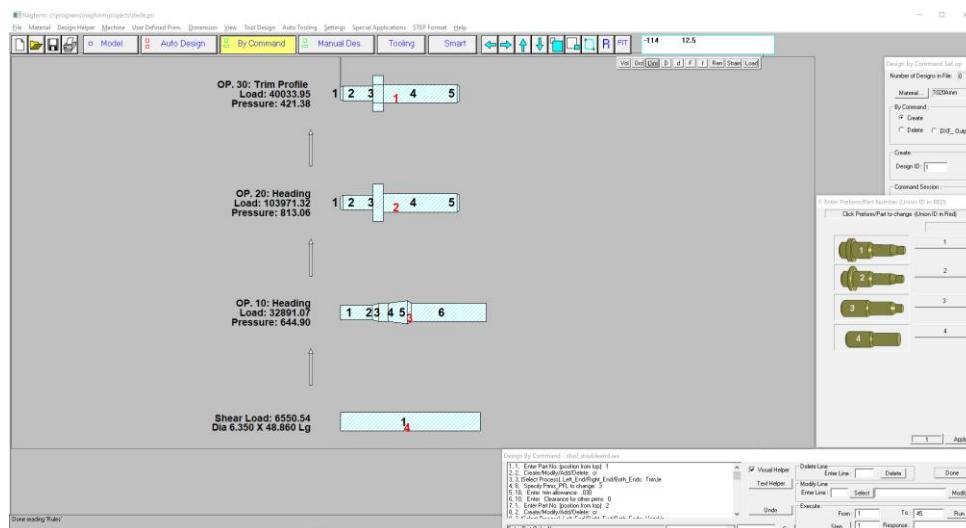


Figure 5

9. In ‘Design by Command’ dialog box, enter the first line number in ‘From’ and last line number in ‘To’ edit boxes. Click ‘Run’ to execute the whole file or ‘Step’ through by repeatedly pressing the ‘Step’ button. You have to enter 1 in Step edit box before you step through.
10. The sequence design based on the Session file will be done in a few seconds.
11. Close the Design by Command’ dialog box.
12. You would be asked if you want to save the Session file. Select ‘No’. Then you would be asked if you want to save the manual design. Select ‘Yes’.
13. Close the dialog box to end Manual design.
14. The manual design you created using ‘Design by Command’ procedure has been saved in your project (.pri) file. You can access it by going into ‘Manual Design’ module. You can get a DXF file output to view the design in a Cad system such as AutoCad.

9. Import and Export of DXF Files

9.1 Import Drawing in DXF Format into NAGFORM

8.1.1 Instructions:

Before importing a drawing in DXF format into NAGFORM, make sure that the drawing has been edited as follows:

1. Remove all dimensioning.
2. Remove borders and text.
3. **All parts should be drawn symmetric about horizontal axis (Centerline).**
4. A single arc crossing the centerline must be broken in two arcs so that the part is symmetric about the horizontal axis (centerline).
5. Arrange all parts in a horizontal pattern such that a vertical line would cut only one part.
6. All corners must be connected properly for NAGFORM to convert a drawing into a union.
7. There should not be any duplicate line or arc.
8. Remove centerline from the part.
9. For hollow parts, draw the bore with a dashed or dotted line.
10. Save the drawing in AutoCad12/LT2 [*.dxf] DXF format by selecting this format in ‘Save as type’ selection.
11. NAGFORM recognizes only lines and arcs. All rectangles, polylines and blocks should be broken into ‘Lines’ and ‘Arcs’.
12. Special shapes such as hexagon, splines etc. drawn in a 2-D drawing can not be recognized except as noted below. So these shapes would need to be added within NAGFORM.
13. Some shapes such as hexagon can be brought in provided these are drawn in the same way as NAGFORM draws them when it outputs these shapes in DXF format.

9.1.2 Import:

To import, press ‘DXF_in’ button in the ‘Model’ or ‘Manual Design’ dialog box of NAGFORM. This would open a dialog box titled ‘Bring In Geometry from DXF File’. Select ‘Prims with bore’ to have primitives/union drawn in tubular primitives or select ‘Subtracted Prims’ to have the union drawn in as solid with subtracted primitives. Note that NAGFORM’s further development is with subtracted primitives only. So it is preferred that you choose “Subtracted Prims” selection.

Pressing ‘Next’ will bring up the window’s ‘Open file’ dialog box. Select the DXF file that has the drawing and press ‘Ok’. NAGFORM will read the ‘Entities’ section of the DXF file and convert the entities into ‘Primitives’. NAGFORM will also join the adjoining primitives into ‘Union’.

Note that when NAGFORM converts a drawing into ‘Union(s)’, the union ID is written in red. If a red ID does not appear, it means that NAGFORM did not convert the primitives to a union. This may happen if the primitives were not joined properly or there was only one prim.

More information on ‘DXF_IN’ can be found in ‘Help’ menu of the program and in the topic ‘Create And Use a Historic Knowledge Database of Parts and Forming-Sequence Designs in NAGFORM’ written in this manual.

9.2 Output NAGFORM Designs in DXF file

9.2.1 Instructions

NAGFORM allows you to save the following in DXF file:

1. Part (s) created in ‘Model’ module
2. Automatic designs created by the program.
3. Manual designs created in ‘Manual Design’ module and those created by modifying automatic designs.
4. Tooling components and assembly designs created using ‘Tooling’ module.

To obtain DXF file of a manual or tooling design, press DXF_output button. Program will create a DXF file of the design shown in the graphics window. The DXF file created has the same name as the project file with last letter replaced by a numeral or letter to create a different name from the existing DXF files.

In the Automatic design module, open the ‘Design Setting’ dialog box from ‘Settings’ menu on the desktop. Select ‘Write DXF design file’ option. A DXF file of all automatic designs will be created when designs are determined.

9.2.2 Reading DXF Files Created by NAGFORM

To read DXF file created by NAGFORM in AutoCAD:

1. Start a ‘New’ drawing or ‘Open’ an existing drawing in AutoCad
2. At the command line of AutoCad, type ‘DXFIN’
3. AutoCad will bring up ‘Open File’ dialog box. Select the DXF file and press ‘Ok’.
4. Select ‘Extents’ in to bring all entities in view.

You can also double click on the drawing in ‘Explorer’ to open the DXF drawing in ‘AutoCad’.

9.2.3 DXF Representation of Non-Symmetric Part Features – In Development

Currently, only 2D part geometry in DXF format can be brought into NAGFORM program. Any feature that requires more than one view to draw cannot be brought in. Attempt is being made to bring in Non symmetric part features such as hex through DXF.

Some non-symmetric part primitives such as a Hex can now be brought in provided these are drawn in the same way as NAGFORM does when it outputs these in DXF format. The lines and the arcs must be drawn in the same way.

9.2.4 Examples:

‘DesignFromDXF’ directory contains a number of DXF files that have non-symmetric part features. These files are being read in for design in ‘Design From DXF Database’ procedure. The non-symmetric features are drawn in a standard way so NAGFORM can read and interpret the non-symmetric shapes.

9. ‘SMART’ Database

9.1 Introduction to ‘SMART’

Objectives of ‘Smart’ Database:

- Create and maintain a historic database of parts designed and manufactured.
- Capture knowledge gained from design, tooling and manufacturing experiences.
- Use the knowledge to improve sequence design, quoting and manufacturing processes.
- Reduce/eliminate duplication of sequence design and tooling efforts.
- Reduce time and cost to design, quote and manufacture a part.

How Does ‘Smart’ Work:

Information on the formed part, its sequence design, quoting and manufacturing processes is stored in the ‘Smart’ database. For any new part, the existing information in ‘Smart’ database is searched and analyzed to assist in design and manufacturing of the new part.

Information Stored in ‘Smart’ Database:

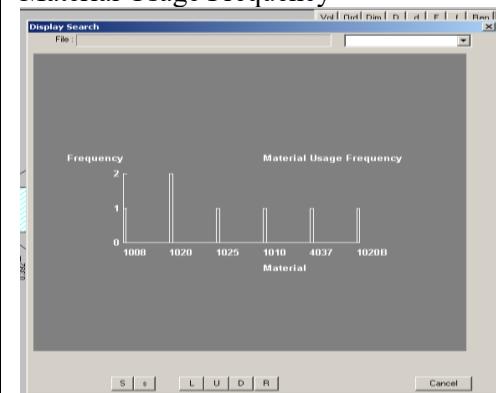
The data stored is date, part geometry, part material, and sequence design from Auto Design, ‘Design by Command’ design or manual design. The user-entered data includes date, part material, sequence design, forming machine, quantity, cost per piece, tool life and lessons learnt.

Search and Analysis:

‘Smart’ has powerful search capabilities to assist in the sequence design, tooling and manufacturing of a part, and to eliminate duplication of effort:

1. *Part:*
 - a. Parts in database similar to the new part and extent of similarity.
 - b. Parts that have a single or group of geometry features such as Hex, taper, lobes within a specified size range.
 - c. Parts within a specified range of length, diameter or volume.
2. *Design* (Auto, ‘Design by Command’, Manual):
 - a. Existing design, if any for the new part.
 - b. Existing designs (Auto, ‘Design By Command’ and/or Manual) for parts similar to the new parts.
3. *Wire Size:*
 - a. Wire sizes used over a specified period of time.
 - b. Frequency of use of various wire sizes.
 - c. Parts that used wire within a specified size range.
 - d. Range of wire size used.
4. *Forming Machine:*
 - a. Forming machines used over a specified period of time
 - b. Frequency of use of various forming machines
 - c. Usage of a particular machine
5. *Tool Life:* Tool life data for parts similar to the new part.
6. *Cost:*
 - a. Percent difference between Quoted and Actual costs.
 - b. Average and Maximum difference.
 - c. Parts with cost difference greater than a specified limit.
7. *Lessons Learnt:* *Lessons learnt on similar parts within specified similarity.*

Material Usage Frequency



Cost Comparison

Search Smart.db

Materials	Wire	Forming Machine	Parts																					
Design	Tool Life	Lessons	Cost																					
Date	Sort by Date																							
Starting: 06/16/06	<input checked="" type="radio"/> Ascending																							
End: 07/30/06	<input type="radio"/> Descending																							
Search																								
<input checked="" type="radio"/> % Difference - Manufacturing and Quoted <input type="radio"/> % Difference - Max <input type="radio"/> % Difference - Average <input type="radio"/> Different Greater than % <input type="text"/>																								
Determine																								
Results																								
<table border="1"> <thead> <tr> <th>File</th> <th>Record ID</th> <th>Cost Difference %</th> </tr> </thead> <tbody> <tr> <td>bolt_hexbolt.pri</td> <td>5</td> <td>16.666667</td> </tr> <tr> <td>bolt_trimmed.pri</td> <td>6</td> <td>0.000000</td> </tr> <tr> <td>bolt_wheel_2d3b_ses.pri</td> <td>4</td> <td>-25.000000</td> </tr> <tr> <td>dxf_bush_3d3b.pri</td> <td>20</td> <td>5.263158</td> </tr> <tr> <td>dxf_hexagon2_2d2b.pri</td> <td>22</td> <td>-18.181818</td> </tr> <tr> <td>dxf_hexbolt_3d3b.pri</td> <td>21</td> <td>-8.108108</td> </tr> </tbody> </table>				File	Record ID	Cost Difference %	bolt_hexbolt.pri	5	16.666667	bolt_trimmed.pri	6	0.000000	bolt_wheel_2d3b_ses.pri	4	-25.000000	dxf_bush_3d3b.pri	20	5.263158	dxf_hexagon2_2d2b.pri	22	-18.181818	dxf_hexbolt_3d3b.pri	21	-8.108108
File	Record ID	Cost Difference %																						
bolt_hexbolt.pri	5	16.666667																						
bolt_trimmed.pri	6	0.000000																						
bolt_wheel_2d3b_ses.pri	4	-25.000000																						
dxf_bush_3d3b.pri	20	5.263158																						
dxf_hexagon2_2d2b.pri	22	-18.181818																						
dxf_hexbolt_3d3b.pri	21	-8.108108																						

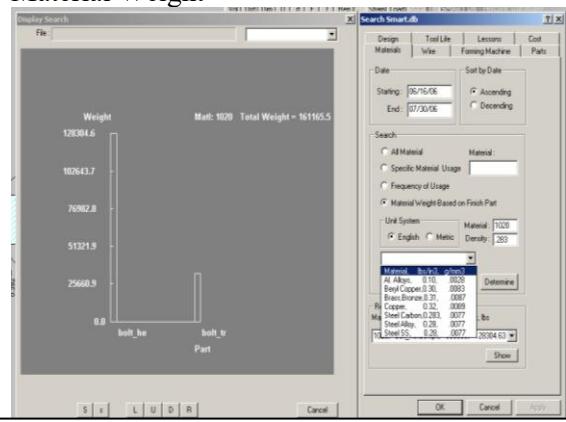
Lessons Learnt

Search Smart.db

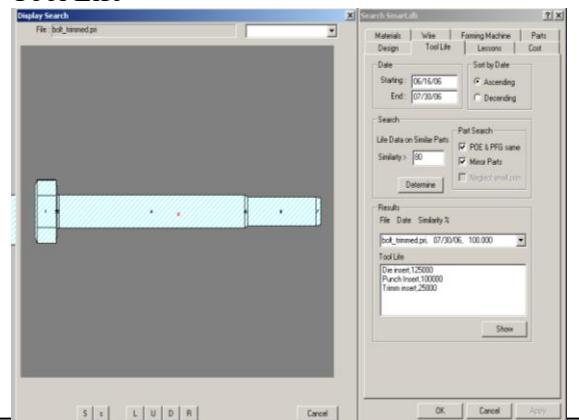
Materials	Wire	Forming Machine	Parts
Design	Tool Life	Lessons	Cost
Date	Sort by Date		
Starting: 06/16/06	<input checked="" type="radio"/> Ascending		
End: 07/30/06	<input type="radio"/> Descending		
Search			
Lesson on Similar Parts Similarity > <input type="text" value="30"/> Part search <input checked="" type="checkbox"/> POE & PFG same <input checked="" type="checkbox"/> Mirror parts <input type="checkbox"/> Neglect small prims			
Determine			
Results			
File Date Similarity % bolt_wheel_2d3b_ses.pri. 06/16/06. 38.571			
Lesson Lesson 1: Increasing the insert shrink fit from .006 to .01 did not increase the life Lesson 2: Using M4 instead of Carbide did not change the cost. So used M4 to reduce cost			
Show			
OK Cancel Apply			

Smart Database Module

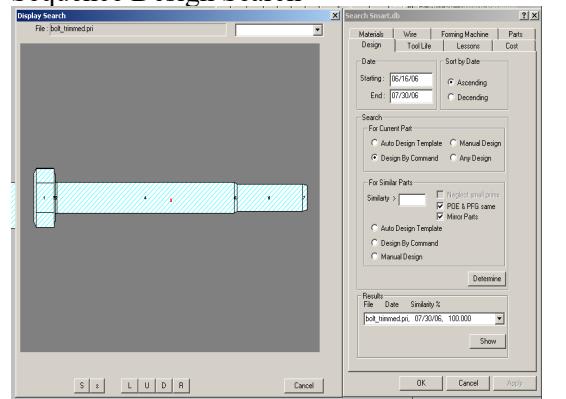
Material Weight



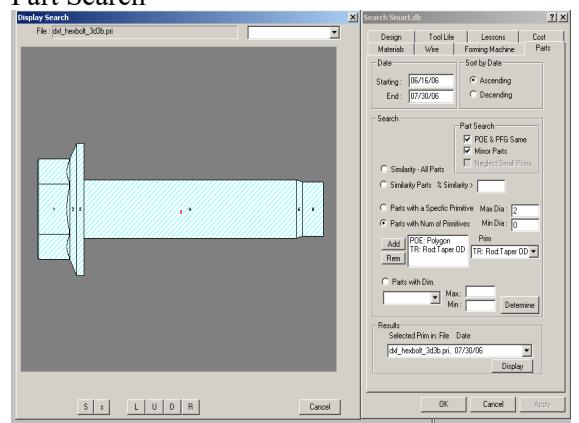
Tool Life



Sequence Design Search

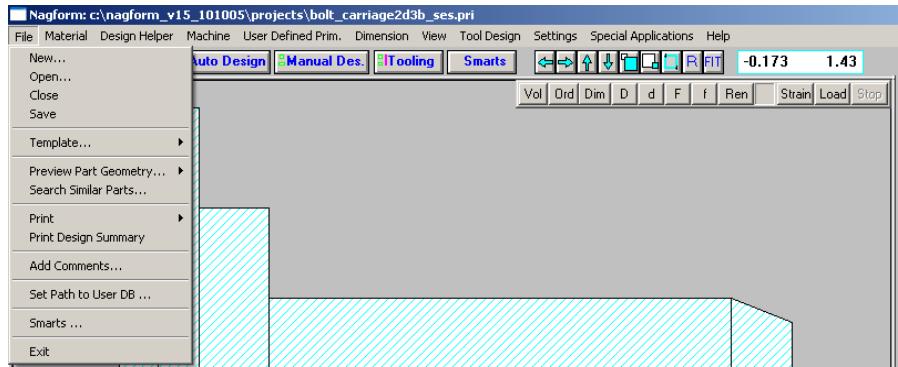


Part Search



9.2: Creating and Modifying ‘SMART’ database

To work in ‘Smart’ database, go to ‘File’ menu ->Smart... option or click on ‘Smart’ Button on the Main window as shown below.

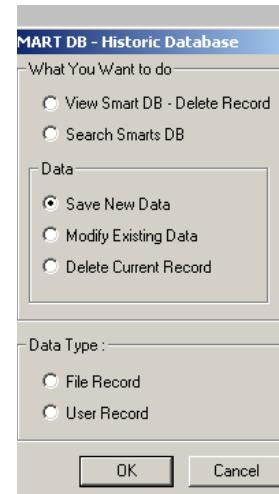


Choosing any of these options would open up the ‘SMART DB – Historic Database’ dialog box on the right as shown below. This is the main ‘Smart’ dialog box.

The various options are:

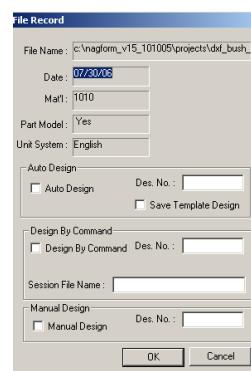
1. View Smart DB summary / Delete any record
2. Search Smart DB
3. Save New Data (File Record or User Record)
4. Modify Existing Data (File Record or User Record)
5. Delete the Current Record (File & User or User alone)

File Record here refers to the information put in from the NAGFFORM project file (.pri) and User Record here means the data put in by the User at (a) Quoting Stage and (b) after or during Manufacturing stage. Information in User Record comes from the User.



9.2.1: Save New Data (File Record, User Record)

New File Record: To create a new record (Save New data) of a part, the User must have opened the project file (.pri) of that part. Select ‘Save New Data’, then select ‘File Record’ or ‘User Record’. If you select ‘File Record’ and press ‘OK’, a dialog box would appear on the right side of the main window as shown.

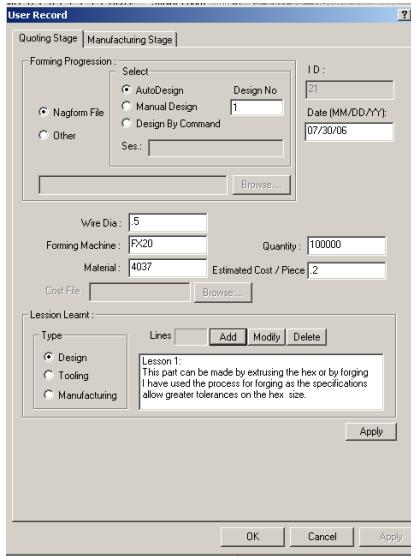


Some of the information such as date, material, and unit system would show up automatically. This is read by the program from the project file.

To save sequence design in the ‘Smart’ database, check mark the appropriate Check Box for the type of design and specify enter the design number. Note that you can enter more than one design type like ‘Auto

Design', 'Design by Command' or 'Manual Design. When you click OK, the program would read the design from the project file and copy it into the Smart DB.

Save New User Record: To save new User Record, select 'Save New Data' and 'User Record' in the main 'Smart' dialog box. When you press 'OK', a multiple-dialog box with 'Tab Control' would appear on the right side of the main window.



Note that you must save 'File Record' for the part before you can save the 'User record'. If File Record does not exist, the program would display a message box asking you to first save the file record. It is not necessary to have a User Record for every File Record in 'Smart'.

The User Record has two stages, namely, Quoting Stage and Manufacturing Stage. When you are quoting the part, enter the information in 'Quoting Stage' dialog box. After the Quoting process, if you manufacture the part, fill in the 'Manufacturing Stage' data. You can omit filling in the data on any stage.

At 'Quoting Stage', you can enter the following information:

- Date
- Forming Progression Design
- Wire Diameter
- Forming Machine
- Material
- Quantity
- Estimated cost /piece
- Lessons learnt

Note that the program assigns an ID (Identification Number) to each part. Same ID is used for both the File Record and the User Record of the part. This ID number is assigned automatically. Once assigned, it is not changed.

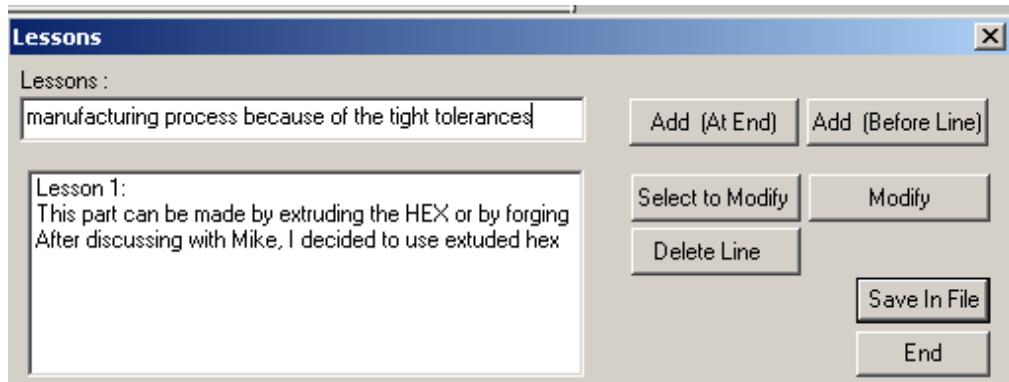
The date is automatically entered but the User can change the date.

Forming progression here means the part progression layout you used in estimating the cost of the part. It can be a design created by the 'NAGFORM' program or it can be your own progression. If the Forming Progression is from NAGFORM, select 'Nagform File' and enter the information about the design such

as the type of the design (Auto, Design by Command or Manual), and design number. If you chose ‘Design by Command’, you would need to enter the name of the Session File you used to create the ‘Design by Command’ design. If you have your own progression, select ‘Other’.

To enter the ‘Lesson Learnt’, click on the ‘Add’ button. A dialog box as shown below would appear. You would enter the lesson learnt in this (Lesson) dialog box and press ‘Save in File’. When you close ‘Lesson’ dialog box, the text you wrote there would appear in the list box of the ‘User Record’ dialog box. Click on ‘Modify’ button to modify the contents of the lesson. It would take you back to the ‘Lessons’ dialog box. You make the changes there and Save again. The modified text would be displayed in ‘User Record’ list box. To delete a lesson altogether, press ‘Delete’ button.

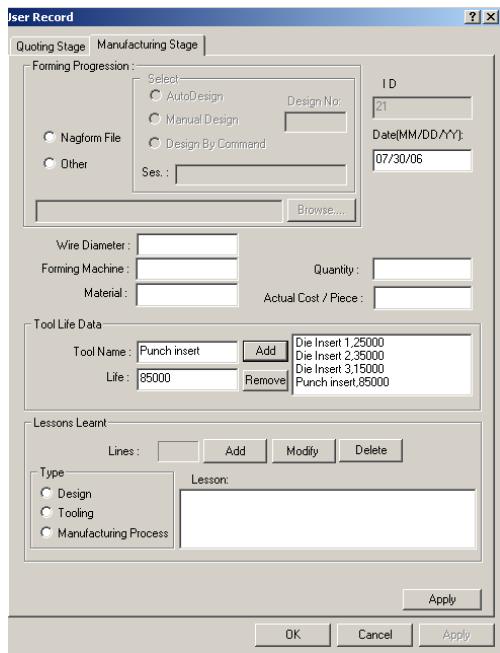
After you have entered all the information, press on ‘Apply’ button. The ‘User Record’ would be written in the ‘Smart’ database file.



To write the lesson in the ‘Lesson’ dialog box, write the text line in the upper edit box. Click on ‘Add (At End)’ button to put this line of text at the end of the contents of the list box. If you wish to add this written text line before a certain line in the list box, select that line in the list box and then click on ‘Add (Before Line)’ button. This would add the written line before the selected line in the list box.

To modify the text in list box, select the line to modify in the list box. Press ‘Select to Modify’ button. The selected line would now appear in the edit box above the list box. Make changes and click on ‘Modify’ button. The selected line of the list box would be changed. After you are done writing or modifying the lesson, press on ‘Save in File’ button. The list box contents would be saved and written in the parent dialog box that called this ‘Lesson’ dialog box.

To save User Record in the ‘Manufacturing Stage’, use the Tab control in the ‘User Dialog Box’, to display the ‘Manufacturing Stage’ dialog box. The data entered here should be the actual data used in /during the manufacturing process.



At ‘Manufacturing Stage’, you enter all of the same type of data as you entered in ‘Quoting Stage’. In addition, you can enter the tool life data as shown.

To enter the tool life data, enter the tool name and the tool life in the two edit-boxes. Press ‘Add’ to move this data to the list box on the right. If you want to modify or delete, select the data in the list box and click on the ‘Remove’ button. The selected data in the list box would be removed. You can enter the new data again. When the ‘User Record’ is saved, only the tool data that is displayed in list box is saved. Press ‘Apply’ to save the User Record.

9.2.2 Modify Existing Data (File Record, User Record)

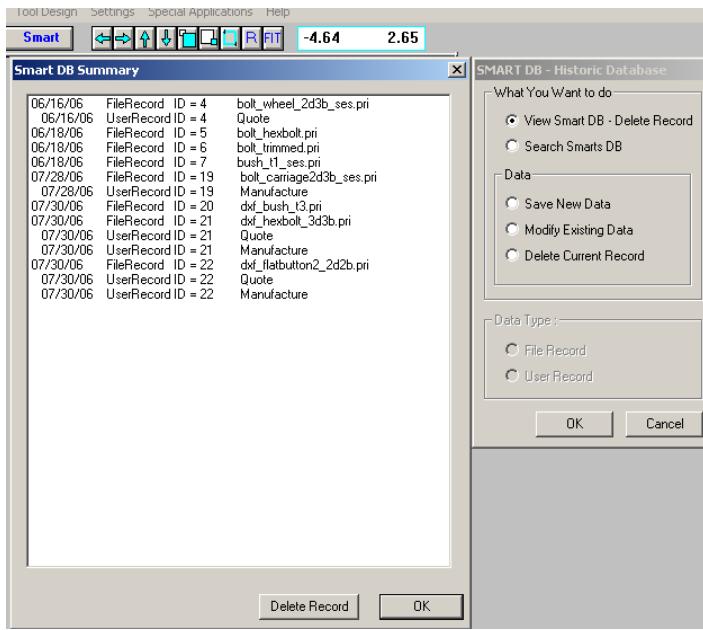
To modify an existing ‘File Record’ or ‘User Record’, in the main ‘SMART DB – Historic Database’ dialog box, select ‘Modify Existing Data’, then select ‘Modify File Record’ or ‘Modify User Record’ depending upon which record you want to modify. When you click OK button, the same dialog boxes that were used for entering ‘New’ data would appear. The existing data of the record would be displayed in the respective dialog boxes. Make the modifications as needed and press ‘Apply’ button. The modified record would be written in the ‘Smart’ database.

9.2.3. Delete Current Record (File Record, User Record)

To delete the File Record or the User Record of the current part, select ‘Delete Current Data’, then select ‘Delete File & User Record’ or ‘User Record’ depending upon which record you wish to delete. Note that the User Record cannot be in the Smart database without the File Record. Therefore when you delete the File Record, the User Record is deleted as well.

9.2.4. View Smart DB Summary And Delete Records

To view a list of the File and User Records currently in Smart database, select ‘View Smart DB – Delete Record’ option in the main ‘SMART DB – Historic Database’ dialog box. A dialog box ‘Smart DB Summary’ would appear next the main dialog box as shown below.



The information listed is Date, type of record, its ID and the NAGFORM project file name.

The list is sorted by the date of the project File Record. User Records of ‘Quoting Stage’ or ‘Manufacturing Stage’ are listed after the File Record, irrespective of the dates of these records.

Note that the File Record and the User Record for the same part (Project file) have the same ID.

To delete any record from the Smart database, select the record in the list box and click ‘Delete Record’ button. If you delete a File Record, the User Records would also be deleted.

9.3. Search using Smart Database

The current version of NAGFORM_start has some powerful search capabilities especially related to the part geometry. However, the data in User Record – Quoting Stage is not being used except for the estimated cost. The search capabilities will be extended further in the next version after getting feed back from the Users.

To search ‘Smart’ database for information, select ‘Search Smart DB’ option in main Smart dialog box and press ‘OK’. A multiple-dialog box with ‘Tab Control’ as seen below would appear.

User can perform search related to any of the following:

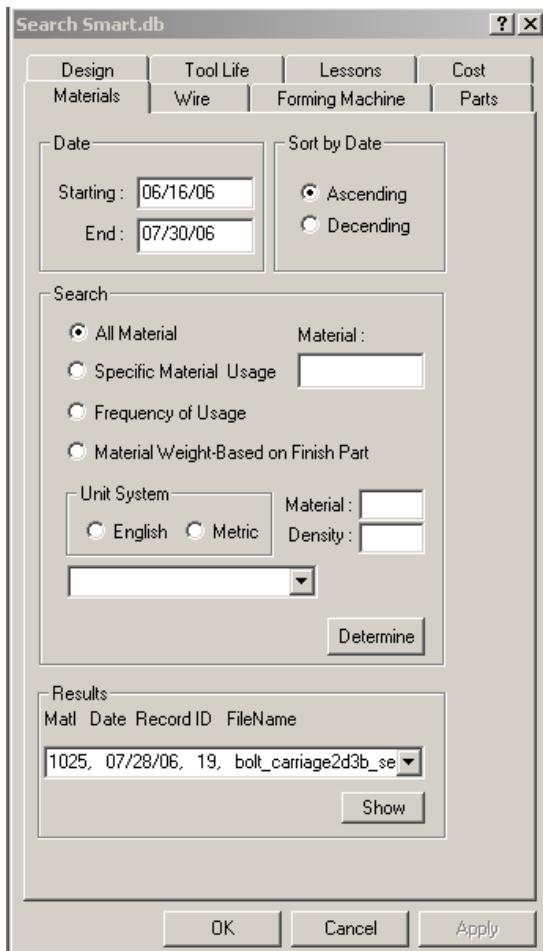
- Material
- Wire
- Forming Machine
- Cost
- Tool Life
- Lessons learnt
- Sequence Design
- Parts

Choose any of the above by clicking on the respective tab. Currently the data search is done from the following records.

For Part Geometry:
File Record

For Wire, Forming Machine, Design, Tool Life, Lessons:
User Record – Manufacturing

For Cost:
User Record – Quote
User Record - Manufacturing



In most searches, where relevant, the Start and End dates of data to be searched can be specified. All dialog boxes have Start Date and End Date edit boxes. Also where relevant, the results can be presented sorted in ascending or descending date. Each dialog box has Ascending and Descending sort selection.

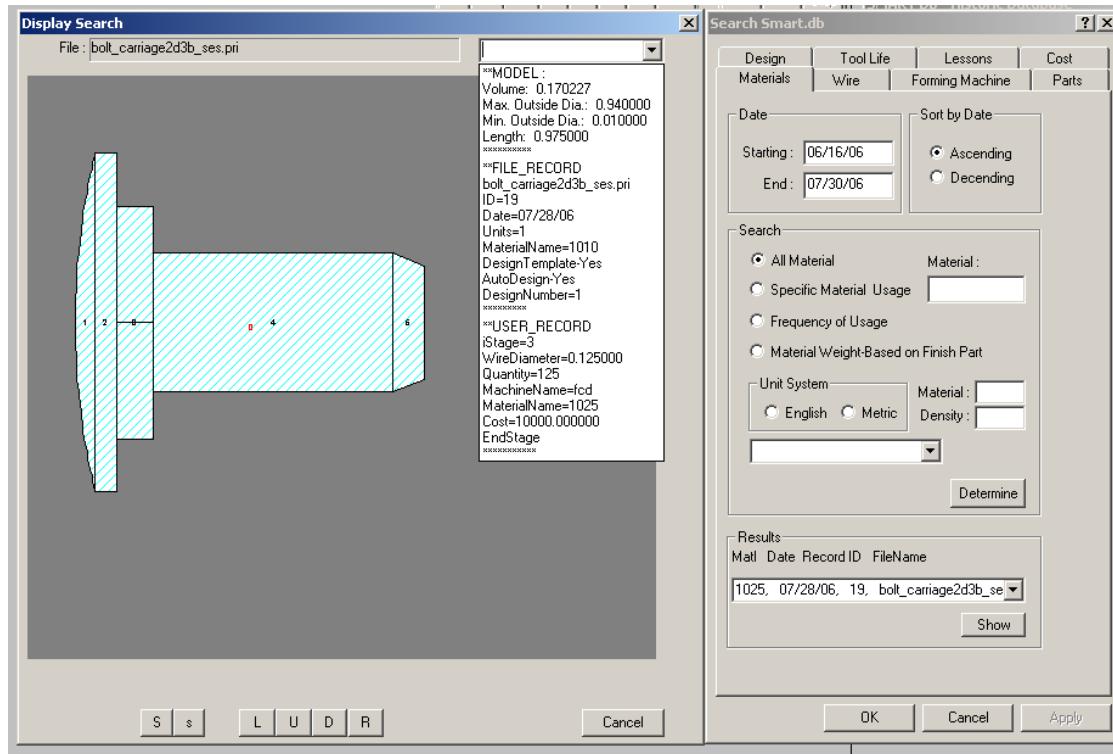
9.3.1: ‘Material’ Search

Click on ‘Material’ tab to perform search related to the part material. The search options are:

- All Materials
- Specified Material Usage
- Frequency of usage of all different materials
- Weight of a material used (based on finish part weight)

The search is done using the User Records – ‘Manufacturing Stage’ in the Smart database. For all searches, enter the search period by entering the Start and End dates. Also select ‘Ascending’ or ‘Descending’ sort by date.

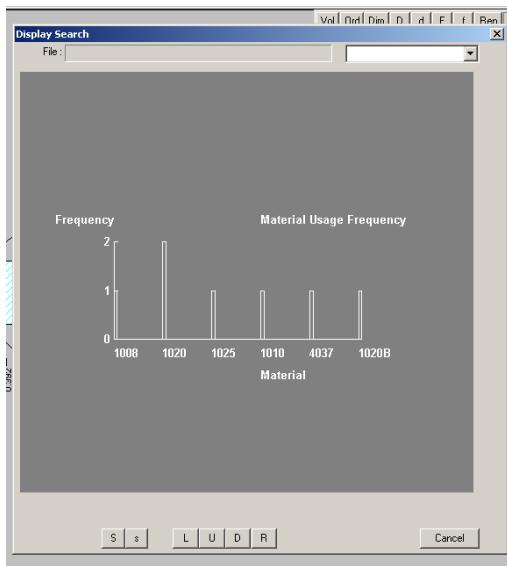
When you select ‘All Materials’ option and press ‘Determine’, the program determines the material of all the parts for which User Record – Manufacturing Stage exist in the Smart database. The results are listed in Drop-Down list box marked ‘Results’. To display any of the listed part, select it in the list box and press ‘Show’ button. This would display the part in a new window ‘Display Search’ as shown below.



You can see a summary of the part dimensions, File Record and the User Record using the Drop-down list box of the new window. The part graphic display can be moved to Left, Up, Down and Right using the buttons marked ‘L’, ‘U’, ‘D’ and ‘R’ respectively. Use ‘S’ to zoom in and ‘s’ to zoom out. Pressing ‘Cancel’ will close this ‘Display Search’ window.

To search the usage of a particular material, select this search option in ‘Material Search’ dialog box and enter the material name in the edit box next to this selection. Pressing the ‘Determine’ button would list all part files where this material is used, as determined from the User Records.

The ‘Frequency of Usage’ means the number of times a material is used. Selecting this option lists all different materials and the number of times they were used in the Results Drop-down list box. You can get a visual representation by selecting any of the files in the Results list box and pressing ‘Show’ button. A plot as shown below would be displayed in a new window next to the ‘Materials’ dialog box.



You can move the plot or zoom in or out using the buttons of the ‘Search Display’ window.

To get the weight of a specified ‘Material’ based on finished part dimensions, select this option, enter the material name, its density and the Unit System in which you have specified the density. Click on ‘Determine’ to get the material weights in the Results drop down box.

To get a visual representation, select any of the items in the Results list box and press ‘Show’. The results will be displayed in the ‘Display Results’ window.

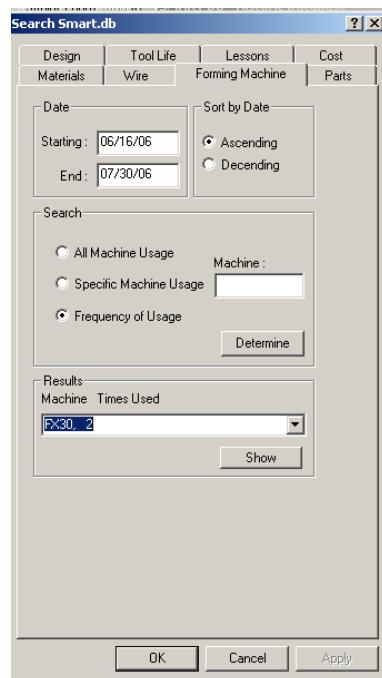
To assist you in specifying the density of the part material, densities of common materials in both English and Metric Units are in the Drop-Down box of ‘Materials’ dialog box as shown. Note that for English units, it is assumed that the part dimensions are in inches. The density is specified in lbs/in³. The weight would be in ‘lbs’. For Metric Units, part dimensions are assumed to be in ‘mm’. The density is specified in g/mm³. The weight is divided by 1000 to present the results in kgs.

9.3.2. ‘Wire’ Search:

The search options on Wire Size are as seen in the dialog box on right. These options are similar to that for the ‘Material’ search. Selecting ‘Wire Diameter Range’ option and clicking on ‘Determine’ button displays maximum and minimum wire sizes used.

9.3.3. ‘Forming Machine’ Search:

The search options on Forming Machines are as seen in the dialog box on right. These options are also quite similar to that for the ‘Material’ and ‘Wire’ searches.



9.3.4. ‘Cost’ Search

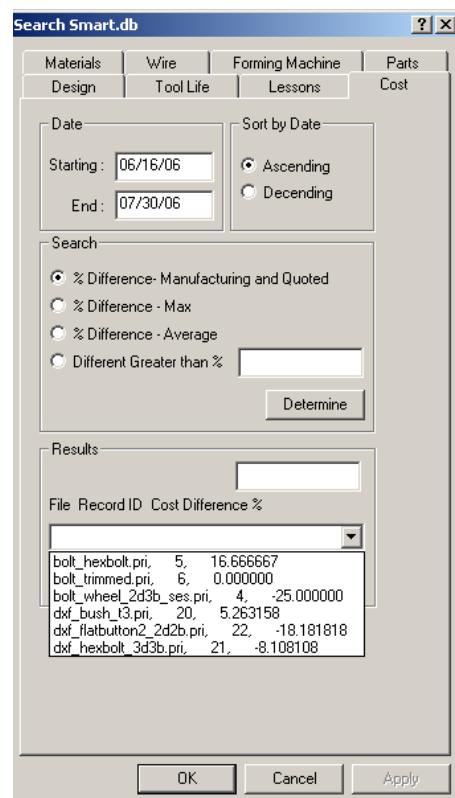
The search options related to ‘Cost’ are shown in the dialog box on right.

Select the option ‘% Difference – Manufacturing and Quoted’ to see the difference in manufacturing cost and quotes price for all parts. Press ‘Determine’ button to display results in the Drop-down list box as shown.

Select ‘%Difference – Max’ option to display the maximum difference in quoted and manufacturing costs for any part. This value is displayed in the edit box in Results. You can also see the part by selecting it in the drop-down list box and pressing ‘Show’.

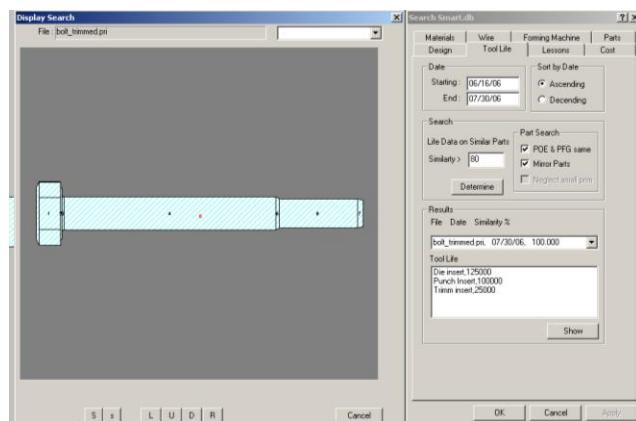
To find average difference in costs, select the ‘% Difference –Average’ option.

To see all parts for which the difference is greater than a specified percent limit, select this option, enter the percent difference limit and press ‘Determine’.



9.3.5. ‘Tool Life’ Search

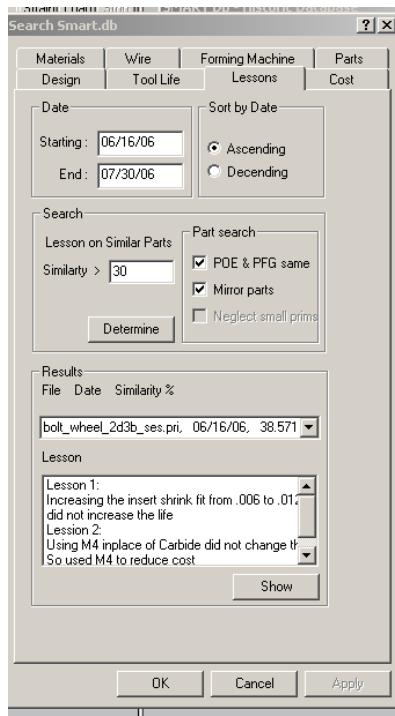
The Tool Life dialog box is used to get ‘Tool Life’ data on parts that are similar to the current part in the project file. Enter the similarity limit for finding similar parts. Select ‘POE & PFG’ same if you want the program to consider these two types of Polygon primitives to be same while calculating similarity. When ‘Mirror Parts’ is selected, the program determines the similarity between the two parts and also between one part and mirror of the other part. The greater of the two similarity numbers is taken as the extent of similarity between the two parts



When you click on ‘Determine’ button, the program finds all the parts that are similar to the current part with similarity greater than the specified limit. To display the Tool Life data for any of the similar parts, select it in the drop-down list box and click ‘Show’. The similar part would be shown in the ‘Display Search’ window and the tool life data will be shown in the Results list box as seen above.

9.3.6. ‘Lessons Learnt’ Search

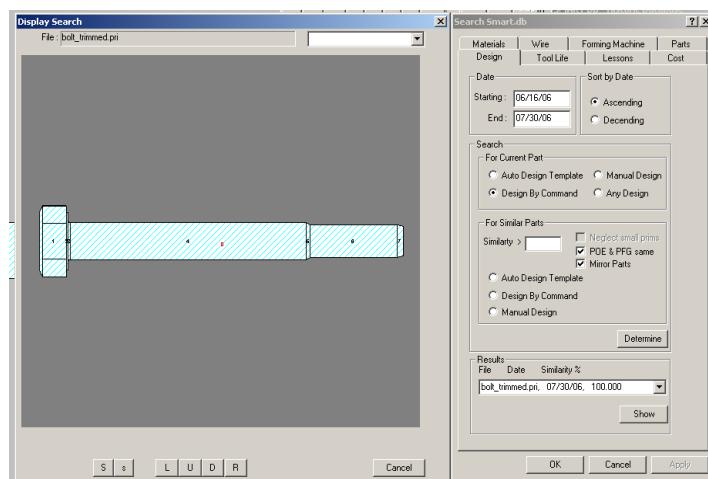
You can search for lesson learnt using this dialog box. The search procedure for this dialog box is very similar to that for ‘Tool Life’ dialog box.



9.3.7. Sequence Design Search

Using this dialog box, you can search for sequence designs in the Smart database for the current part. The various search options are listed in the dialog box.

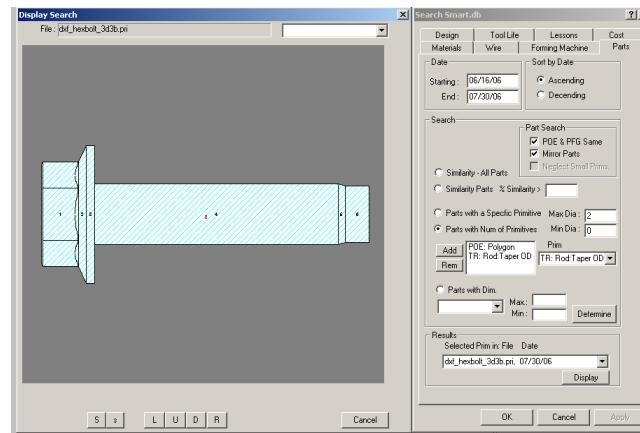
To determine designs for the current part, select the type of design (Auto Design, Design by Command, and Manual) or select ‘Any’. When you click on ‘Determine’, the program searches for parts in database that have very close geometric similarity (> 95 %) to the current part. Then the program determines if the type of design selected is in the database for these parts. Those parts that have the sequence design are listed in the Results drop-down list box. You can display any of the parts in the list box by selecting it and pressing ‘Show’ button.



If no design is available, you can look for sequence designs for similar parts by specifying the similarity limit. Note that the designs of a part with similarity less than 80% would not be directly useful for the current part.

9.3.8. Parts Search

In NAGFORM_Smart, a number of ‘Part Search’ options are provided to eliminate duplication of sequence design, tooling and manufacturing efforts. The options are shown in the dialog box.



Select ‘POE & PFG Same’ and/or ‘Mirror Parts’ on the considerations explained above.

- When you select ‘Similarity –All Parts’ option, the program calculates similarity between the current part and all of the other parts in ‘Smart’ database. The values are listed in the Results drop-down list box.
- To determine parts those have similarity with the current part above a certain percent, select this option and enter the Similarity Limit in the edit box. Pressing ‘Determine’ button would display all those parts.
- To find parts that have a certain primitive within a specified diameter range, select this option. Select the type of prim from the ‘Prim’ drop-down list box, enter the Max. and Min. diameters and press ‘Determine’.
- To find parts that have a number of specified primitives within specified diameter range, enter these types of primitives one by one in the list box by selecting them in ‘Prim’ drop-down list box and pressing ‘Add’. To remove any from the list box, select that prim type in the list box and press ‘Remove’. After you have all the types of primitives in the list box, enter the maximum and minimum diameters and press ‘Determine’. The program would find all parts in Smart database that have all of these primitives within the specified diameter range. Those parts would be in the Results list box.
- To find parts that have ‘Length’, ‘Diameter’ or ‘Volume’ within a specified range, select the part feature (Length, Diameter, Volume) for the drop-down list box next to this selection. Enter the maximum and minimum value in the two edit boxes and press ‘Determine’ button. All parts that meet the criteria would be displayed in the Results list box.

10. Tooling Design Using NAGFORM

10.1 Introduction

‘Auto Design of Tooling’ is being developed in the NAGFORM software program in order to reduce the time required to design and draft tooling components and assemblies.

‘Auto Design of Tooling’ For Cold Forged Parts is being developed in NAGFORM software program to:

1. Reduce the time required to design and draft tooling for any cold forged part.
2. To directly take the sequence design created by NAGFORM or User to NAGSIM.2D program for simulation without the need for preparation of tooling drawings by the User.

‘Auto Design of Tooling’ is being developed in three phases:

1. Auto Design/Drafting of Standard Tooling Components - NAGFORM creates a fully dimensioned Auto-Cad DXF drawing of a tool component from the dimensions of the tool in a few seconds.
2. Auto Design of Tooling Assemblies applicable to any forming machine. - The program has a tooling assembly module that allows the user to create assemblies by bringing in different tools. By setting up different constraints between the tools, many of the assembly dimensions are determined automatically. Once the assembly is complete, the .dxf drawings are generated automatically.
3. Auto Design of Tooling: Part Sequence Design -> Tooling Assemblies -> Tooling Components.

10.2 Phase 1 - Auto Design / Drafting of Standard Tooling Components.

NAGFORM has the capability to provide eighty five standard tooling components to its users. These include:

- Die Inserts
- Punch Inserts, Pins
- Monogram Tools
- Trim dies / punches
- Spacers – fillers
- Die/ Punch casings

This is only a partial list of tooling components that are needed for Auto Design of the Tooling Assemblies. The goal is to increase the number of standard tools to 500 and to give the user the ability to add their own tools to the library.

Library Drawings

Designs:

- PUNCH INSERTS
- TRIM DIES
- PUNCH PINS
- DIE INSERTS
- OTHERS
- SPACERS
- DIE CASINGS
- SPECIAL TOOLS

Punch Inserts

Punch Insert_7	Punch Insert_9	Punch Insert3_11
Hex Punch Insert4_13	Taper Bush_21	Insert_94
Insert_95	Insert_96	Punch Insert_18
Monogram Tool1_47	Monogram Tool2_48	

Trim Dies

Round Trim Die_38	Trim Die Square_55	Trim Die Hex_56
Trim Die Round_35	Trim Punch_50	Trim Die Round_88
Trim Die Square_86	Trim Die Hex_87	

Punch Pins

Punch Pin_14	Punch Pin_16	KickOut Pin_65
Ejector Pin_66	Pierce Pin 6Lb_57	Pierce Pin 12R_58
Pierce Pin_34	Ejector Sleeve_74	Ejector Sleeve_75
Punch Pin Hex_98	Punch Pin_15	Punch Pin_79
Punch Pin_80	Punch Pin_82	Kickout Pin_64

Die Inserts

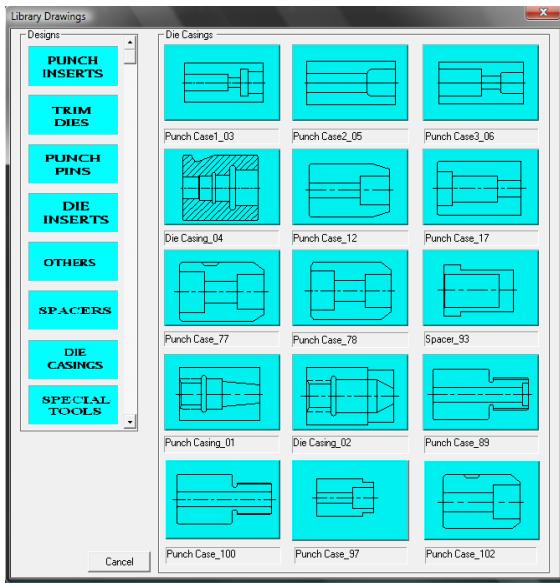
Taper Insert_30	Trap Insert_27	Die Insert with Case_43
Die Insert with Case_39	Taper Die Insert W/Case_44	Die Insert Ext Upset_32
Die Insert_8	Die Insert_10	Insert_68
Split Die Sleeve_83	Split Die Sleeve_04	Die PointInset_51
Die PointInset_52		

Other Tools

Nut_73	Spring_25	Nut_67
Die Hard Plate3_41	Nut_71	Nut_72
Punch Holder_76		

Spacers

Back Filler_40	Bush_63	Spacer with Case_42
Die Filler_45	Die Filler_46	Solid Taper Spacer_49
Solid Taper Spacer_60	Solid Taper Spacer_61	Taper Bush_62
Spacer_90	Spacer_91	Spacer_92
Punch Plug_81	Punch Plug_85	



10.2.1 How to Create Standard Tools

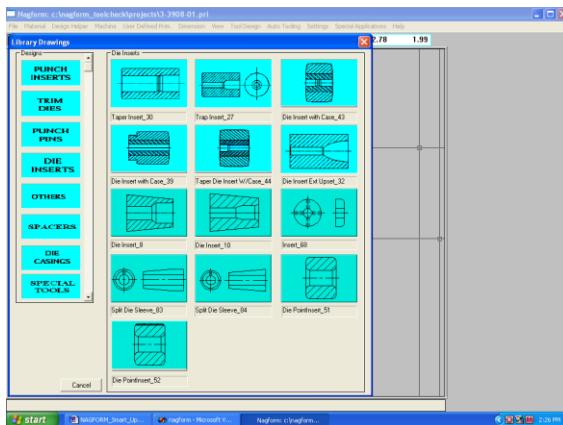
STEP 1 – ACTIVATE TOOLBAR - The tooling section of NAGFORM can be activated by clicking on the Auto Tool Button after which, a tool bar will appear.



The various buttons in this tool bar are:

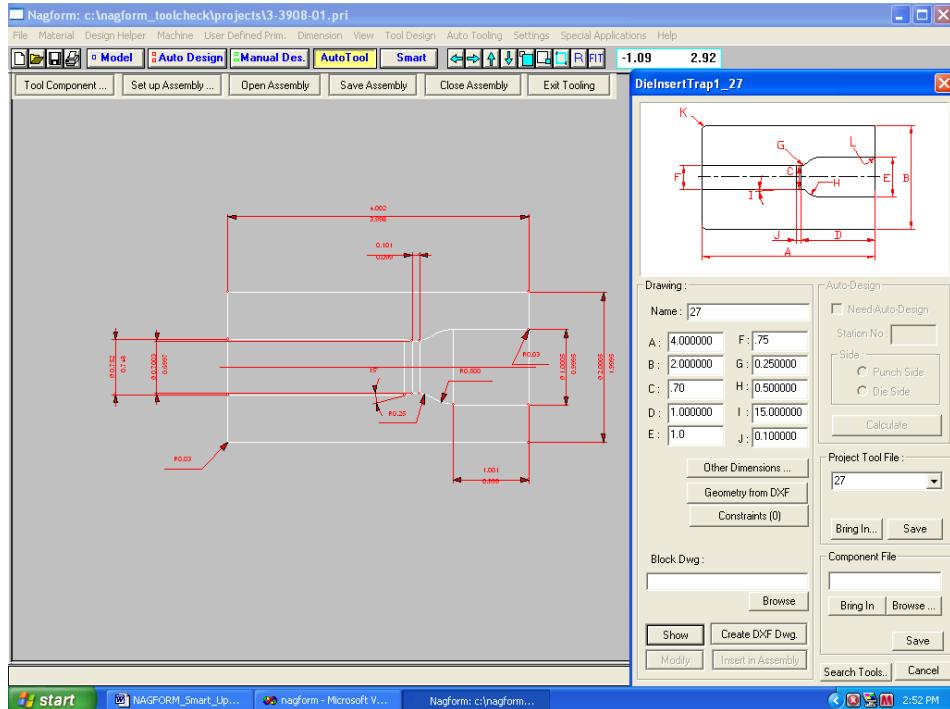
- Tool Component: Auto design/drafting of a Standard Tool Component.
- Set up Assembly: Set up for single station or Multi station subassemblies.
- Open Assembly: Open the tooling assembly from file (.asm).
- Save Assembly: Save assembly in file (.asm).
- Close Assembly: Save assembly and close the assembly module.
- Exit Tooling: Close this tool bar and deselect ‘Auto Tool’

STEP 2 Select Tool Component - In the Tool bar below ‘Auto Tool’, press ‘Tool Component’ button. A dialog box as shown below would open.

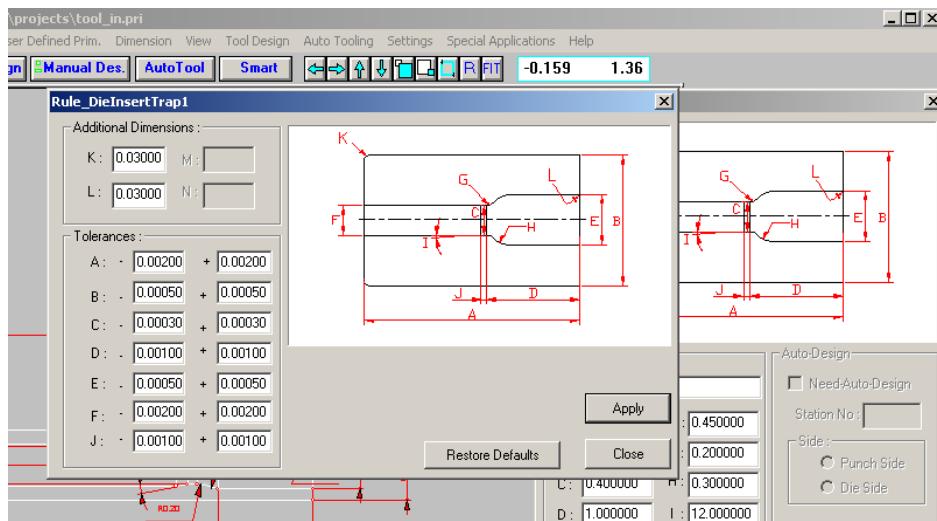


On the left hand side of this dialog box, select the category in which your tool may fit. Currently, the categories are Punch Inserts, Trim Dies, Punch Pins, Die Inserts, Others, Spacers and Casings. As you click on the main category, the tool components that belong to this category are shown on right. Click the tool on the right that you want to draw or save as component file. Here for illustration, we have chosen the trap extrusion die ‘Trap Insert1_27’. A dialog box will open on right as shown below.

STEP 3 Enter Tool Dimensions - This dialog box shows a sketch of the tool with the main dimensions that need to be entered by the User. In addition to these main dimensions, there are other dimensions such as tolerances, corner radii etc needed to draw the dimensioned ‘AutoCad’ DXF drawing. This additional data, that may not change from part to part, is put in a separate ‘Other Dimensions’ dialog box.



If you click on ‘Other Dimensions’ button, the second dialog box showing additional dimensions and tolerances will open as shown below.



Thus for each tool, there are two dialog boxes that have the dimensions needed to draw the DXF Drawing.

The default values in ‘Other Dimensions’ Dialog box are read from a text file such as “Rules27_DieInsertTrap1_in” or “Rules27_DieInsertTrap1_mm” depending upon the unit system chosen for the project (.pri) file. You can modify this ‘Rules’ file in ‘Word Pad’ to change the default values of dimensions and tolerances that appear here. The ‘Rules’ Files are in a directory called ‘ToolCompLib’ of NAGFORM program.

Change the dimensions / tolerances as needed, press ‘Apply’ and then close this Rules dialog box. Now enter main dimensions including the name of the tool in the main dialog box for the tool. If you wish to save these values, click on ‘Save’ button. NAGFORM program will write these values in a file with project file name and ‘.too’ extension. To bring back the dimensions, open the drop down box, pick the tool by its name and click on ‘Bring In’ button. NAGFORM program will read the dimensions from the tool file (.too) and display the values in the edit boxes.

STEP 4 Enter Equation Constraints

With equation constraints, it is possible to

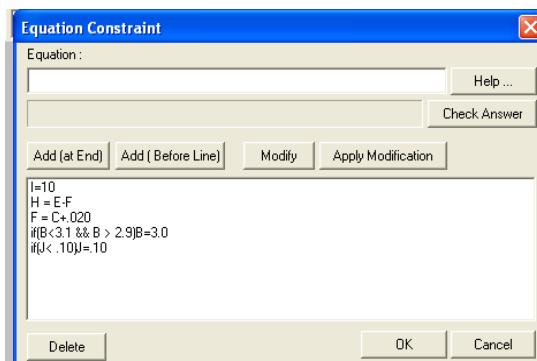
1. Fix values of certain dimensions.
2. Determine dimensions based on equations.
3. Determine dimensions in relation to other dimensions of the tool.

Example 1:

Suppose we want to keep the Angle ‘I’ of the Trap die (DieInsertTrap1_27) to be 10 degree, always. This can be done by setting an equation constraint:

$$I = 10$$

To add this equation constraint to the tool, click on ‘Constraints’ button of the main dialog box of the tool. An ‘Equation Constraint’ dialog box will appear as shown below.



Write $I=10$ in Equation Edit Box, then press Add (at End). Enter OK to apply the constraint to the tool. Press Cancel to close the ‘Equation Constraint’ dialog box. Now in the main dialog box of the tool, enter value of I of 15. If you press Show, the program will apply the above equation constraint and the value of I will be set to 10.

Example 2: You want that the Radius ‘H’ of the trap die to be calculated using the expression:

$$H = E - F$$

Again, Press ‘Constraints’ button. In the Equation Constraint dialog box, enter this equation in ‘Equation’ edit box, then press Add(at tend) button. This constraint equation will be added to the list box. Press

'OK', then Press Cancel to close the dialog box. Now in the main dialog box, enter different values of E and F. The radius 'H' will automatically change according to the above equation.

Example 3: Suppose you want to keep the clearance (difference between F and C dimensions) to be .020. This can be done by setting an equation constraint: $F = C + .020$.

Example 4: Supposing that you want the Dimension 'B' to be a whole number '3' if its value is determined to be between 2.9 and 3.1. This can be done through an equation constraint:

if($B < 3.1 \&\& B > 2.9$) $B = 3.0$

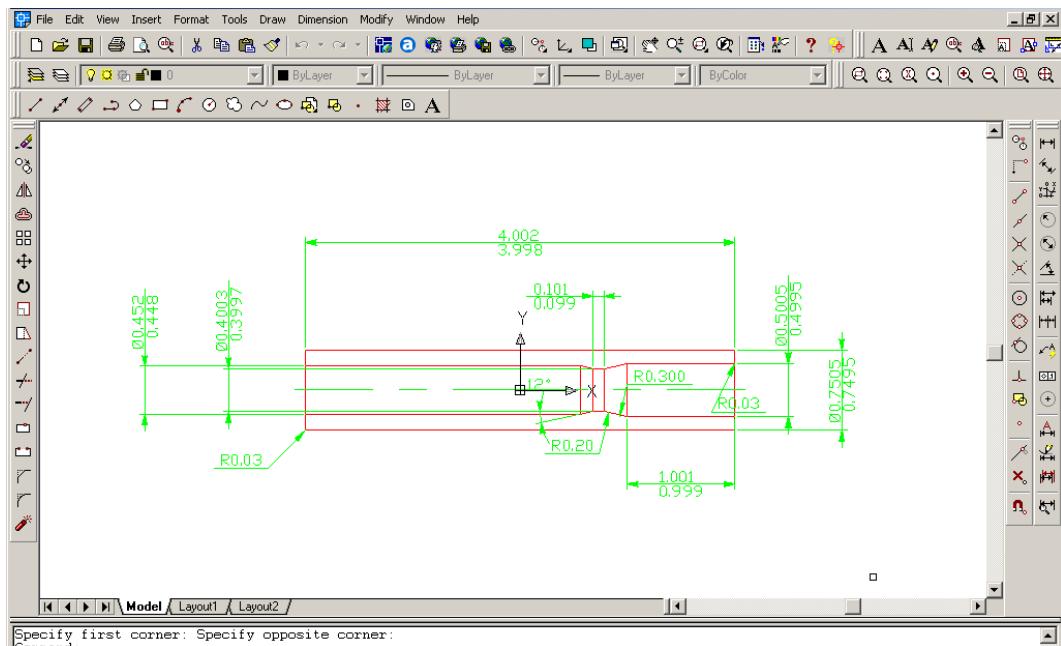
Example 5: Suppose that you want the minimum value of Dimension 'J' to be 0.10. This can be done by setting the equation:

if($J < 0.1$) $J = 1$;

STEP 5 Create Tool Drawing

In the main dialog box, click on 'Show' to display the drawing in NAGFORM graphics area. NAGFORM would display the DXF drawing without creating a drawing. Use NAGFORM graphics buttons to move the display to left, right, up down, zoom in, zoom out, redraw, fit or zoom in using a window. Change the main dimensions or the tolerances as needed.

When you are satisfied that correct dimensions have been entered, click on 'Create DXF Dwg.' button. A DXF drawing of the tool will be created by NAGFORM with the tool name that you had entered. This drawing will be created in 'Projects' directory.



STEP 6 Insert Tool Drawing into Your 'Block' Drawing

NAGFORM program can place the tool drawing in your 'Block' drawing. To place the tool drawing in your Block Drawing, 'Browse' to locate your block drawing. NAGFORM will place the center of its tool drawing at location (0,0) of the Block Drawing. To place it at different location in Block Drawing, use 'Drawing Preferences' dialog box to set the location at which the tool drawing should be placed.

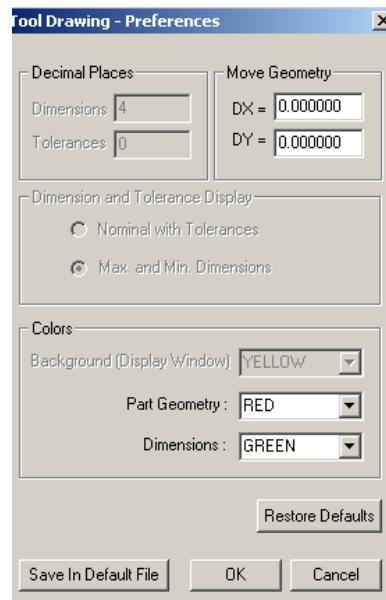
Another approach is to create DXF drawing without using Block drawing. Then use ‘Copy with Base Point’ to paste the NAGFORM tool drawing at the desired location in your drawing.

When you place NAGFORM created drawing in your drawing, it is possible that some of the dimensions would have to be redrawn in the style of your drawing.

10.2.2 – Tool Component Features

Drawing Preferences

You can change the color of ‘Dimensioning’ and that of ‘Tool’ in the DXF drawing by selecting your preferences in a ‘Tool Drawing – Preferences’ dialog box. You can access it through main Menu ->Auto Tooling ->Tool Drawing Preferences.



Select the colors in the drop down list boxes. The Tool Drawing is drawn with center of drawing at (0,0). If you want the center to be at different location, enter DX, DY to move the drawing center to (DX,DY). If you wish to make these preferences permanent, click on ‘Save in Default File’. These will be written in the ‘Preference file’ also located in ‘ToolCompLib’ directory.

Saving Dimension and Equation Constraints in a ‘Component’ File

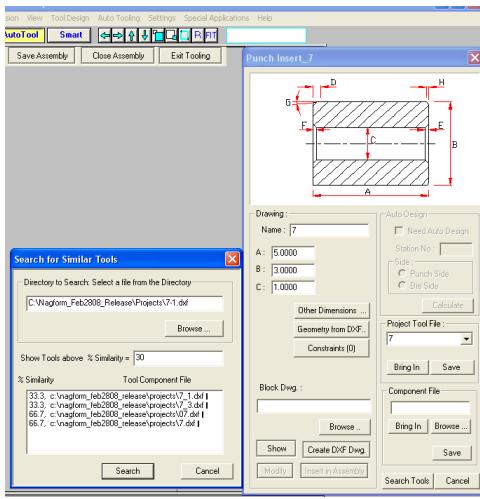
You can save any tool in a Component (.cmp) file for later use. Saving as component file saves the Tool Dimensions such as A, B, C etc and the Equation Constraints. To save as a component file, click on ‘Save’ button in ‘Component File’ group. A Windows ‘Save As’ dialog box will appear. Navigate to the desired directory, enter the file name and press ‘Save’ in Windows ‘Save As’ dialog box. A file of the name you entered will be created and the tool dimensions and Equation Constraints will be saved in it.

You can also use an existing .cmp file to bring in Tool Dimensions as well as Equation Constraints. ‘Browse’ to locate the Component file. Click ‘Bring In’ to read the component file and bring in dimensions.

Search for Similar Tools

In the DXF drawing created by the NAGFORM program, the dimensions (A, B, C etc) of the tool are saved in the DXF drawing. The purpose of storing the dimensional information is to allow search of existing tools of desired dimensions.

To search for tools with dimensions similar to those entered in main dialog box, click on ‘Search Tools’ button at the bottom. A ‘Search for Similar Tools’ dialog box would come up as shown below:



'Browse' to the directory/location you want to search by selecting any file from that directory. Enter the minimum similarity in percentage that selected tools must have. Click on 'Search' button. The NAGFORM program would read all the DXF files in that directory and lists the names of the DXF drawings that have tool dimensions that are similar to the dimensions in the main dialog box. This search for similar tools can only be done for tool drawings created using NAGFORM program.

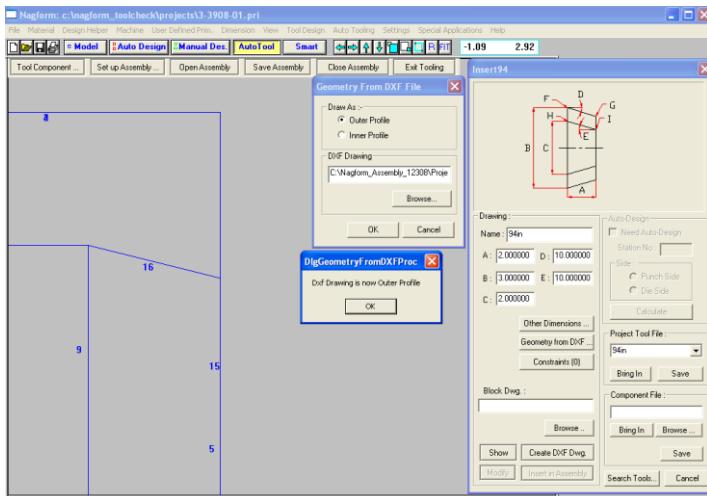
Creating an 'EXTERN TOOL'

'EXTERN TOOL' is a tool with outer or inner profile from a DXF file and inner or outer profile from a Standard Library Tool. The main objective for including an 'EXTERNAL TOOL' type in NAGFORM Tooling Module is:

- Eliminate need to create a 'Standard Library Tool' for tool components who's inner or outer profile does not change but outer or inner profile is available in one of the standard tools. For example, for die casings/ punch casings, sleeves etc. outer profile may not change but inner profile may change to fit different parts. These can be created as an Extern Tool.

To create an 'EXTERN TOOL' from a standard tool, press 'Geometry from DXF' button in main dialog box of the tool. A 'Geometry from DXF' dialog box as shown below will appear. Browse to select the DXF file for outer or inner profile. Press 'OK' button to bring in DXF drawing as outer or inner profile.

Note that the DXF drawing has to drawn with origin set at center of the 'Standard Tool' so that the DXF profile and profile of Standard Tool are aligned.



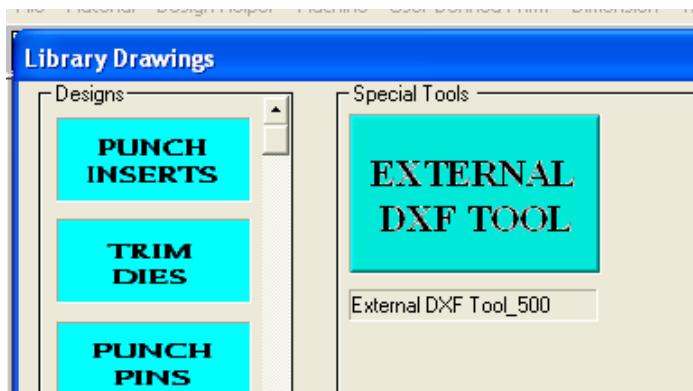
The ‘EXTERN TOOL’ can be treated similar to a Standard Tool with the difference that the profile from DXF file can not be changed. The profile from Standard Tool can be changed by changing the values of applicable parameters ‘A’, ‘B’ etc. The ‘EXTERN TOOL’ can be saved as a Component file just like a normal tool.

Creating a ‘EXTERNAL DXF’ Tool Component

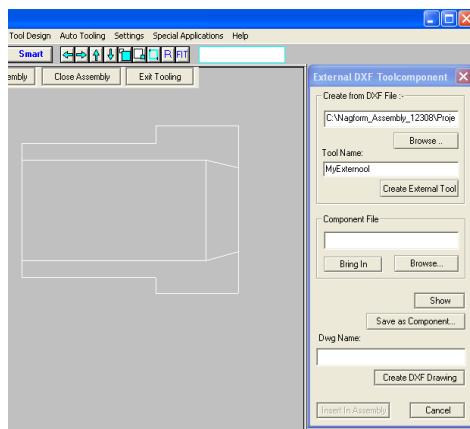
You can create a complete tool from a DXF file. Such a tool is called ‘EXTERNAL DXF’ tool. When the program reads the DXF drawing, whatever is there in the DXF drawing will be read in as ‘EXTERNAL TOOL’. The dimensions or entities of ‘EXTERNAL DXF’ tool can not be changed.

The main objectives for including an ‘EXTERNAL DXF’ type tool in NAGFORM are:

1. Any tool for which NAGFORM program does not have a ‘Standard Library Tool’ can be brought into the Assembly so it can be completed.
2. Eliminate the need to create ‘Standard Library Tool’ for components that do not change. For example, die casings/ punch casings, sleeve etc which are not changed when Assembly is created for different parts, can be created as ‘EXTERNAL DXF’ type tool.



When you select ‘EXTERNAL DXF TOOL’, the dialog box shown below will appear.



To create the tool from DXF, click on ‘Browse’ to select the DXF file. Enter ‘Tool Name’ and press ‘Create External Tool’. The tool will be created. However, to see on screen, you would need to press ‘Show’.

This tool can be saved as a Component File for later use. Press ‘Save as Component’ button to use Window’s ‘Save As’ dialog box to save.

10.3 Phase 2 - Auto Design/Drafting of Tooling Assemblies

In NAGFORM, the capability to create assemblies of a multi station tooling for forming cold forged parts has been added. The assembly information is saved in a tooling assembly file (.asm). The name of the tooling assembly file is the name of the project file with (.asm) replacing (.pri).

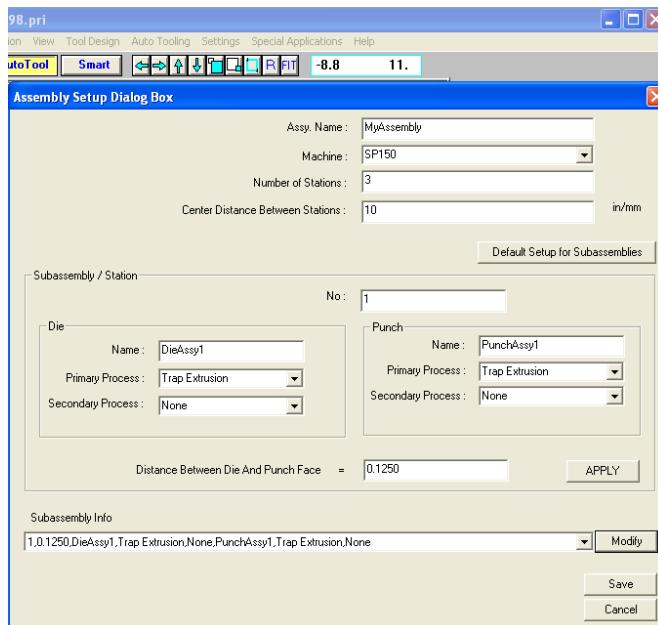
Before constructing the assembly tooling, you need to ‘Set Up’ the tooling assembly. ‘Set Up’ here means defining number of stations (subassemblies), distance between stations, and primary and secondary forming processes for which the tooling is designed. Though the information regarding forming processes is not needed for constructing any subassembly, it would later be used to categorize the tooling assemblies for forming special parts.

10.3.1 Setup for Constructing Tooling Assemblies

Setup needs to be done only once in the beginning. Press ‘Set up Assembly’ button shown below for setup.



This will open the Setup dialog box as shown below:



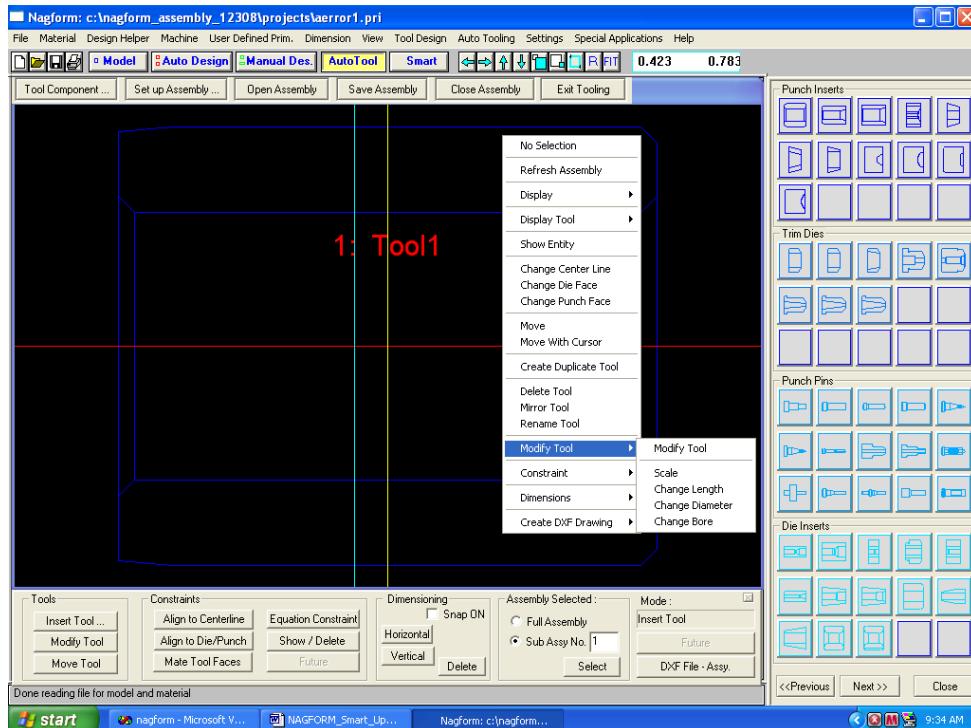
Fill in the information including ‘Assembly Name’, Machine (from drop down box), number of stations and center distance between stations. Now for each station, fill in the name of subassembly, primary forming process and secondary forming process for the die subassembly and the punch subassembly. Then press ‘Apply’ to save this information for each station.

You can also use default set up for subassemblies by pressing on ‘Default Setup for Subassemblies’ button. This will put in default values for subassembly including name, primary and secondary forming processes for each station.

If need be, use ‘Modify’ to modify the subassembly setup information. After you are done, click on ‘Save’ to save this setup in Assembly file. Press ‘Cancel’ to close the dialog box.

10.3.2 Creating Tool Assemblies

To start creating assemblies or continue where you left, press ‘Open Assembly’ button. NAGFORM program read the assembly file (.asm), open a Graphics window and two dialog boxes and display the assembly as shown below. If there were tools created earlier, those will be drawn in the Graphics window along with Die Lines and Punch Lines otherwise the Graphics window would be blank



Popup Menu: When you Right Click Mouse within Graphics area, a Popup menu will come up as shown above. The various menu options are as seen above. ‘No Selection’ or ESC key can be used to get out of previous selection.

Working with Subassemblies for Multi Station Tooling:

For multi station tooling, NAGFORM would draw Centerline (Red), Die Line (Yellow) and Punch Line (Green) for each station. The distance between the Centerlines is taken according to the Assembly Setup information.

To work in any subassembly (station), enter subassembly / station number in edit box next to ‘Sub Assy. No’ radio button in the bottom dialog box. Then click ‘Select’ button. The menu options/commands will now apply to only this selected assembly.

For some of the options, you can work with the whole assembly. Select ‘Full Assembly’ radio button. Please note that many of the menu options such as ‘Delete Tool’, ‘Move Tool’ will not work in ‘Full Assembly’.

Modify Centerline, Punch Line and Die Line:

Use Menu options ‘Change Center Line’, ‘Change Die Face’ and ‘Change Punch Face’ to change the location and dimensions of these lines. When you select any of these options, the cursor will change to a small square. Using this cursor, select the appropriate line. A dialog box will come up showing coordinates of their end points. Modify the coordinates to changes these lines.

Inserting Tools

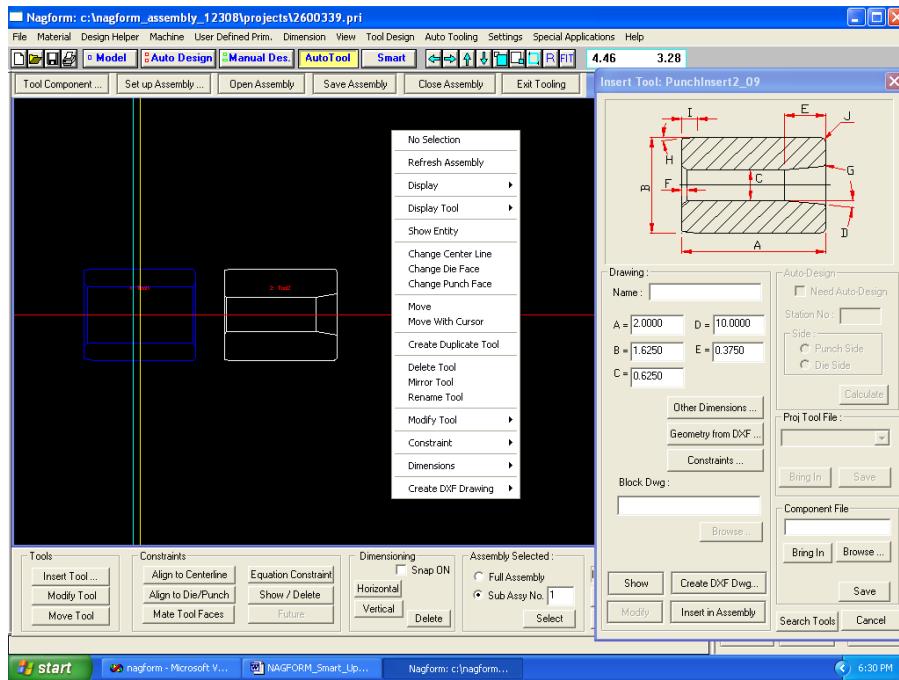
The dialog box on right end is for inserting various standard tools in subassemblies. Each button has a picture of the Standard Library Tool it represents. At the bottom, there are ‘Previous’ and ‘Next’ buttons to move to previous or next set of tools as shown below.



The tool buttons are arranged in the same order as in Standard Tool Library. If you keep the cursor over a tool button, a tool tip displaying the type of the tool will show up.

To insert any tool into the assembly, click over the desired tool button. A dialog box very similar to that used in Standard Library Tool will appear. Some default values of dimensions are entered which can be changed to desired dimensions. You can also insert a tool with default dimensions and later modify it. Also you can bring a tool from Component file.

Click on ‘Insert in Assembly’ button to insert the tool. When the first tool is inserted, the die line and punch lines will show up. Every time a tool is inserted, it is placed on the right of existing tools.



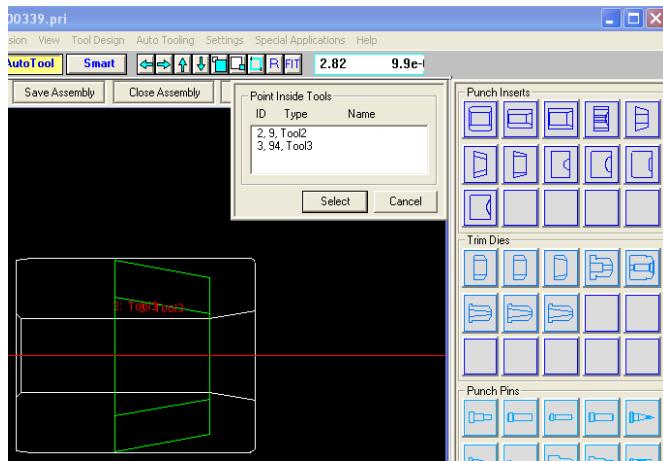
Commands to Facilitate Assembly

Various commands /options to facilitate assembly of tools are:

1. Move Tool
2. Move Tool with Cursor
3. Delete Tool
4. Mirror Tool
5. Duplicate Tool
6. Rename Tool
7. Draw Tool with/without Hatch
8. Show Entity
9. Show Name and ID (one command for all tools)

You can execute these commands by choosing from the Popup ‘Menu’ or pressing appropriate buttons in bottom dialog box.

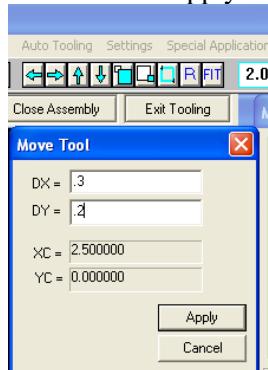
After you choose the command/option, you would, in most cases, need to select the desired tool by clicking anywhere inside the tool. If the clicked point is inside of more than one tool, a small dialog box would appear displaying all the tools that have the clicked point.



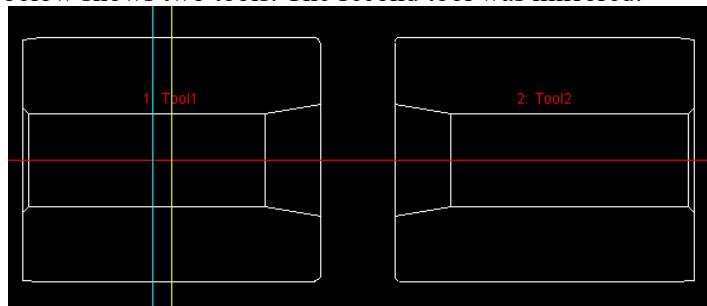
Select the desired tool in the list box and click ‘Select’ button or you can ‘Double Click’ the desired tool to select it.

After the tool is selected, the command/options for the selected tool will be processed by NAGFORM as outlined below:

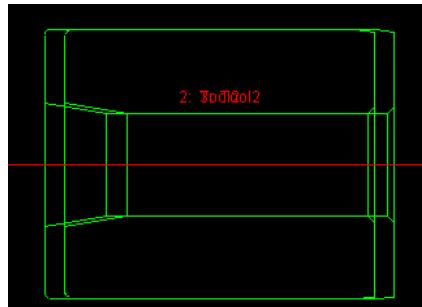
1. Move Tool: The dialog box shown below will appear. Enter the distance to move in X and Y directions and Click ‘Apply’.



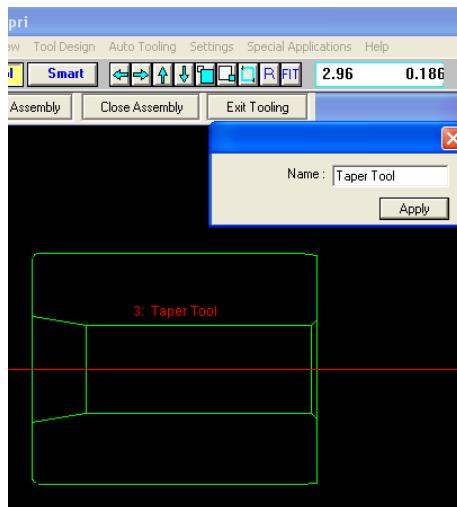
2. Move Tool with Cursor: The selected tool will move with the cursor as long as the left mouse button is kept pressed down.
3. Delete Tool: A Message box will appear asking you to confirm that you want to delete the tool. If you click on ‘YES’ option, the tool will be deleted from the assembly window.
4. Mirror Tool: The tool will be mirrored about the vertical Y axis at the center point of the tool. Note that the original tool is not kept. To bring back the original tool, you can mirror again. Picture below shows two tools. The second tool was mirrored.



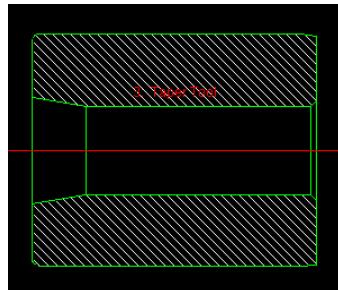
5. Duplicate Tool: A duplicate tool will be created and placed slightly on to the right of the tool that is duplicated as shown below. The duplicated tool will have same dimensions but would not have the tool equation constraints.



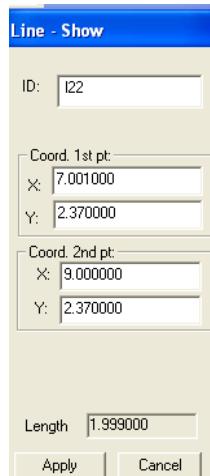
6. Rename Tool: Each tool has an ID (Identification number) and a ‘Name’. The ‘Name’ can be changed. When you select the tool after selecting the ‘Rename Tool’ command, a small dialog box will appear showing the current name of the tool. Enter the new name and press ‘Apply’. The Tool name will be changed to the new name.



7. Draw Tool With Hatch: Depending upon your selection, the tool will be drawn with a ‘Front’ or ‘Back’ hatch.



8. Show Entity: Clicking on any Line or Arc of a tool will bring up ‘Show Entity’ Dialog box which gives coordinates of the end points as shown below.

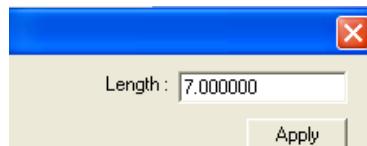


9. Show Name and ID: If name and ID are not being displayed, choosing this option will display Name and ID of all tools. If they are already being displayed, selecting this command would hide the name and ID of all tools.

Modify a Tool

To modify a tool, double click anywhere inside the tool to modify. A dialog box very similar to the ‘Insert Tool’ dialog box will appear on right. Make the changes and then click on ‘Modify’ button to modify the tool.

In addition to modifying the complete tool, there are options to ‘Scale’, ‘Change length’, ‘Change Outside Diameter (maximum diameter)’ and ‘Change Bore’ (minimum inside diameter). Selecting any of these options and selecting the tool will bring a small dialog box with current value of the dimension. Change it to the new dimension and click ‘Apply’ to make the change.



Assembly Constraints

As described earlier, for a tool component, equation constraints can be used to fix values of certain dimensions, determine dimensions based on equations and relate one dimension of a tool to other dimensions of the same tool.

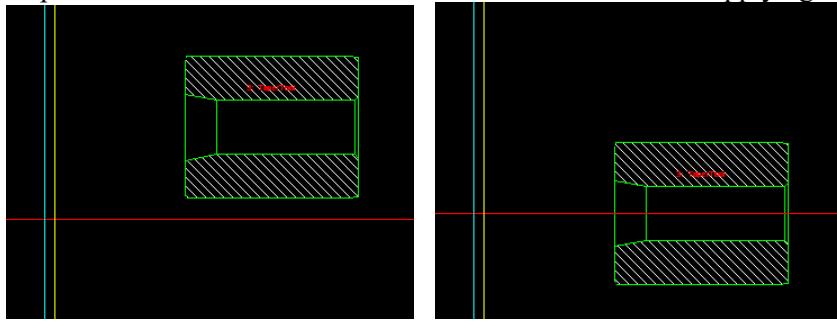
In Assembly, to establish relationship of a tool to assembly centerline, die face, punch face and to other tools in an assembly, ‘Assembly Constraints’ are designed. In NAGFORM, the following Assembly Constraints can be put on a tool:

- Align to Centerline
- Align to Die Face
- Align to Punch Face
- Mate faces of two tools
- Fix Dimensions
- Fix (Complete)
- Equation Constraint between one tool and other tools in an Assembly

‘Align to Centerline’ Constraint:

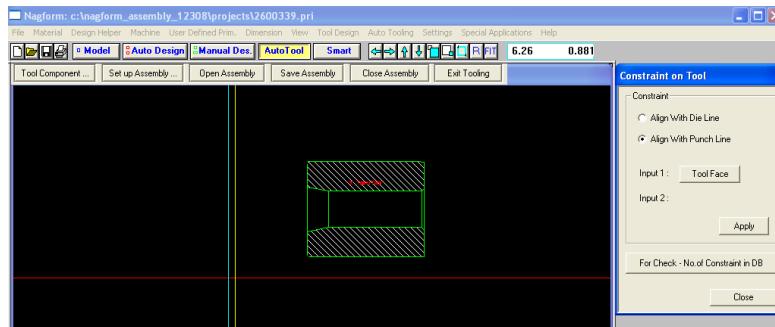
Select this constraint from Popup Menu->Constraints->Align to Centerline. Then select the tool to apply

this constraint. With this constraint, the tool will move to the centerline of the Assembly and would be constrained to stay aligned to the center line. If you move the constrained tool, it would move only along the centerline. The pictures below show the location of tool before and after applying this constraint.



Align To Punch Face:

Select this constraint from Popup Menu->Constraints->Align to Punch Face. Then select the tool to apply this constraint. A dialog box would appear on right as shown below.



Select the radio button ‘Align to Punch Line’ if not selected. Now click on ‘Tool Face’ button. The cursor in Graphics window will change to a small square. Select the vertical face of the tool that is to be aligned to the punch line. Then press ‘Apply’. The tool will move to set the selected face of the tool to the ‘Punch Line’ of the subassembly as shown below.



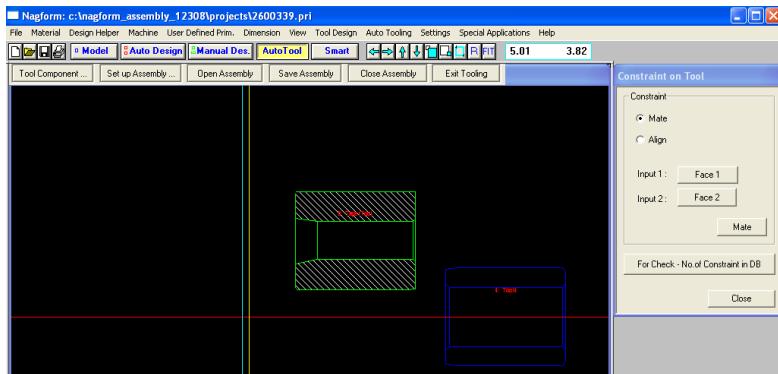
If now you try to move this tool, the tool will move vertically only so that the constrained face of the tool is always set to the Punch line.

Align to Die Face

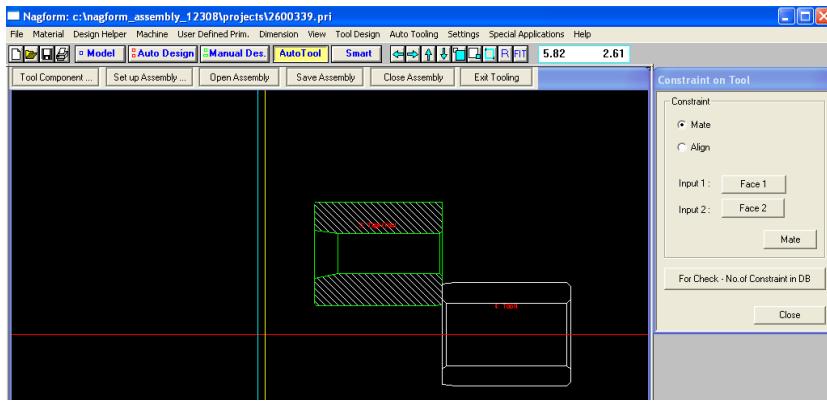
This is similar to the ‘Align to Punch Face’ constraint. The tool face is set to the Die face line of the assembly.

Mate Faces of Two Tools

Selecting this constraint brings up the dialog box shown below.



Press 'Face1' button. The cursor in 'Graphics Window' changes to a small square. Select the vertical face of the first tool. Then press 'Face2' button and select the face of second tool. Press 'Mate' button I the dialog box. The second tool will move horizontally so that the two selected faces are aligned as shown below. Note that the tool is not moved vertically to align centerlines of the two tools.



With this constraint, the faces will always stay aligned. If you move one tool, the other tool will move as well.

'Fix Dimensions' Constraint

Applying this constraint will fix the dimensions of a tool. With this constraint, the tool dimensions can not be modified. However the tool can be moved.

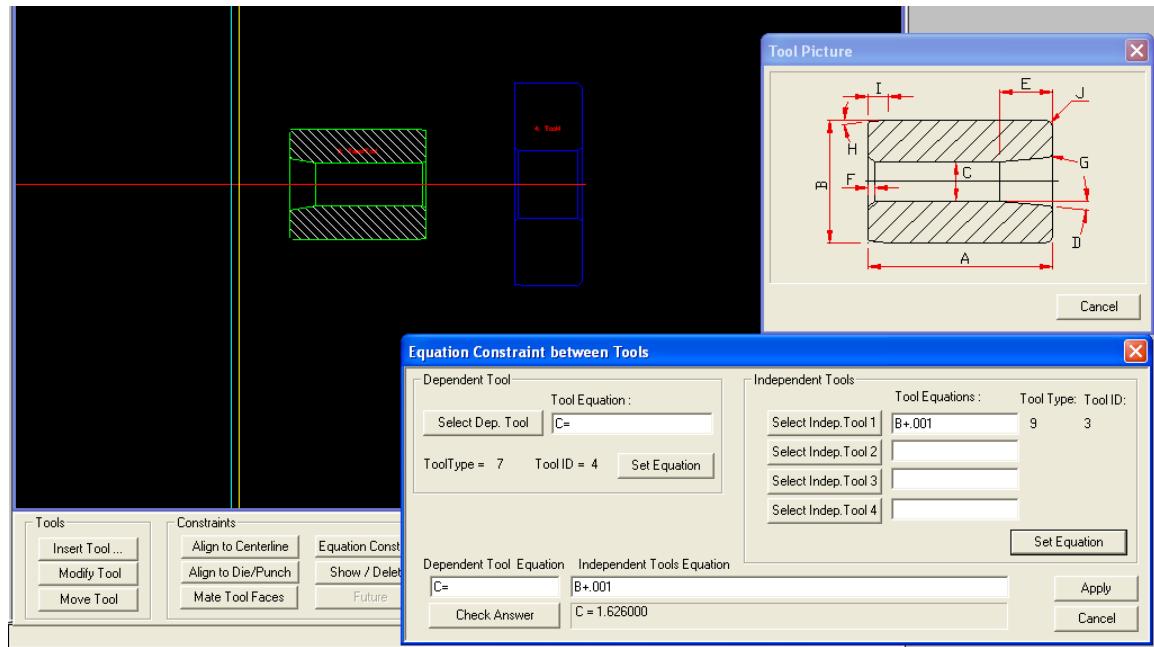
'Fix' Constraint

Applying this constraint will fix dimensions of a tool as well as fix its movement. With this constraint, the tool dimensions can not be modified and it can not be moved.

Equation Constraint Between One Tool and Other Tools in an Assembly

With this constraint, single dimension of a tool (called Dependent tool) can be made dependent upon the dimensions of other tools (called Independent tools). There can be maximum four independent tools.

When you select this constraint, the dialog box shown below would appear. Using this dialog box, equation constraints between tools can be set.

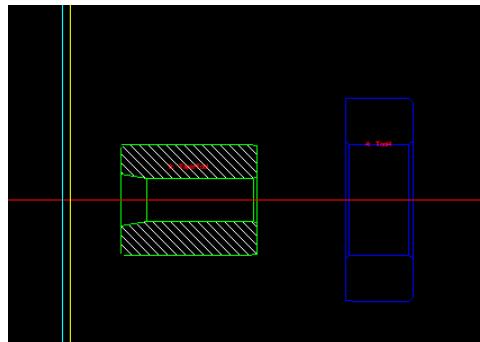


Example: Set ‘Bore’ of tool on right to be equal to outside diameter of the tool on left.

Since the dimension of tool on right needs to be changed, this tool is the ‘Dependent Tool’. Click on ‘Select Dep. Tool’ button. Then click inside of the tool on right to select it. Its type and ID will be displayed in the dialog box. A picture of this tool will also be displayed. Since ‘C’ represents its bore, write the equation ‘C=’ in the edit box and press ‘Set Equation’. The equation of dependent tool will be displayed in the edit box at the bottom of dialog box. Now press ‘Select Indep. Tool1’. Then click inside the tool on right to select it. Its type and ID will be displayed. Also a picture of the tool will appear. Since its outside diameter is ‘B’, write the equation B+.001 in edit box next to ‘Select Indep. Test1’. Then press ‘Set Equation’. This equation will appear below as shown. Press button ‘Check Answer’. Value of C=1.625 is calculated from equation:

$$C = B + .001$$

Where ‘B’ is outside diameter of the tool on left and ‘C’ is the bore of tool on right. As ‘B’ is 1.625, the value of C = 1.626. Click on ‘Apply’ to apply this constraint. The bore of tool on right changes to 1.626 as seen in picture below.

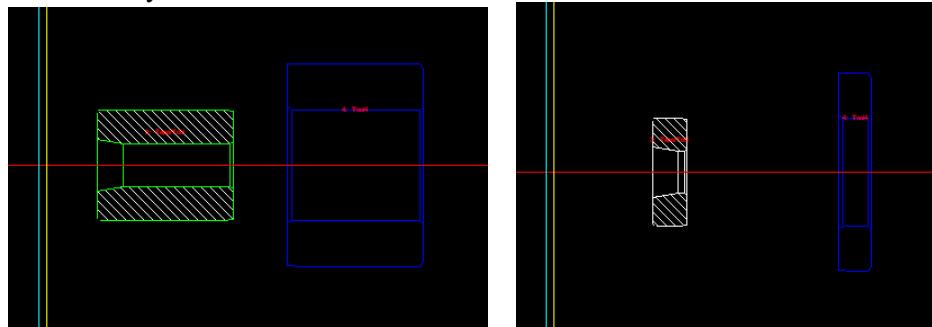


Example: Set ‘Length’ of tool on right to be equal to Length of the tool on left.

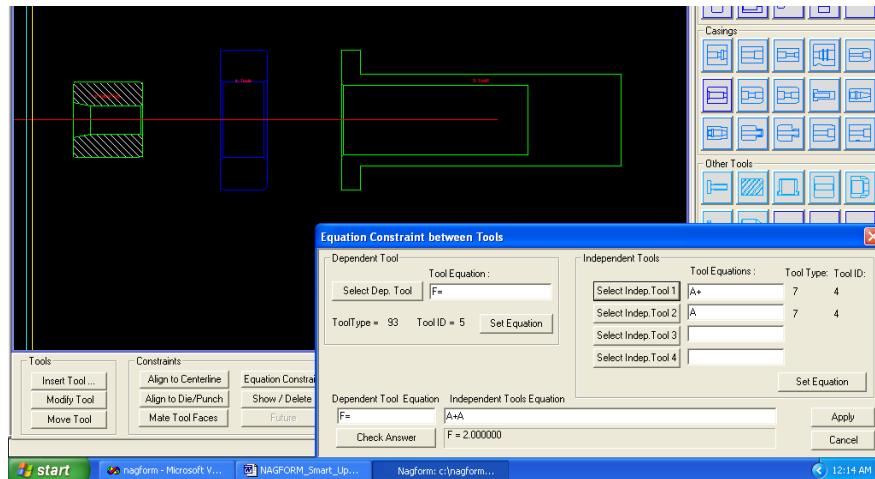
Following the same procedure, set the follow equation:

$$A = A$$

Where 'A' on left of above equation is the dimension 'A' of the tool on right and 'A' on right of the equation is dimension 'A' of the tool on left. When we apply this constraint, the length of tool on right is changed as seen in picture below. If we modify the length of independent tool, the length of dependent tool is changed automatically.



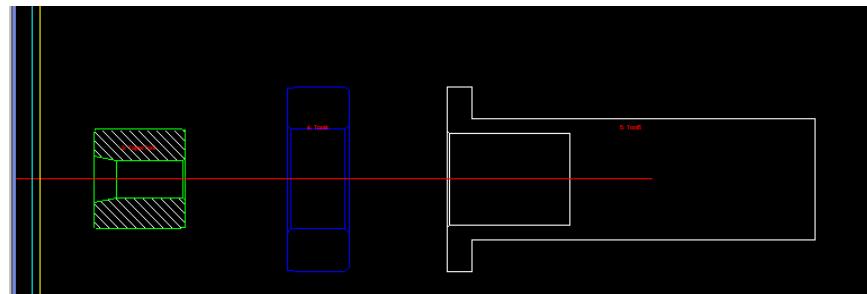
Example: In picture below, Set 'F' (length of hole) of tool on right to be equal to sum of lengths of the two tools on its left.



Following the procedure, set up equation

$$F = A + A$$

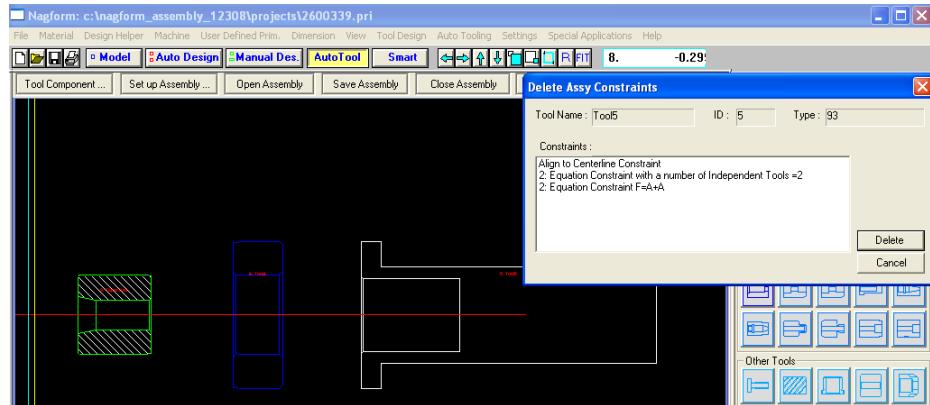
Pressing 'Apply' would put the constraint and the tool on right changes as per the equation as shown below.



Show/Delete Constraints

To see the assembly constraints that have been imposed on a tool or to delete any constraint, select this option from the Popup Menu or from the bottom Dialog Box. Then select the tool by clicking inside it. A dialog box as shown below would appear. The Assembly Constraints that are applied to this tool are listed

in its list box. Incase you want to delete any constraint, select that constraint in the list box and press 'Delete' button. The selected constraint will be removed from this tool as well as from other tools that play any part in the equation.



Refresh Assembly

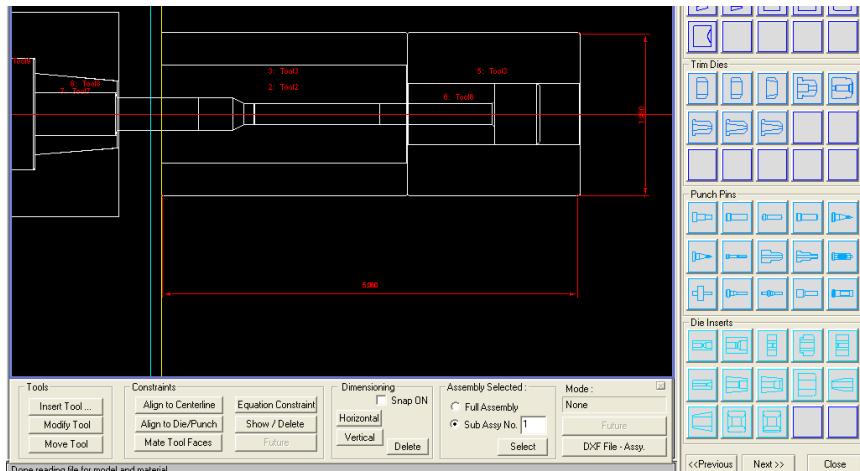
In some cases, the dimensions and locations of the tools in the Assembly may not show the effect of all Assembly Constraints. Selecting this option updates the dimensions/locations of all tools due to Assembly Constraints.

Assembly Dimensions

You can create Horizontal or Vertical Dimension between any two points in the assembly. To pick end points of the entities (line or arc) of any tool, the 'Snap On' can be selected. The program will show the nearest end point of entities by a small square as you move the cursor over various tools. When the desired end point is located by the small square, click left mouse button to select the point for dimensioning.

In the bottom dialog box, press 'Horizontal' button to start the procedure for Horizontal Dimension. Pick first point and then the second point. Horizontal dimension will be displayed. The dimension will move with the cursor. Use the cursor to locate it, then click the mouse left button to place it there.

Use the same procedure for drawing Vertical Dimension.



To delete any dimension, press 'Delete' in 'Dimensioning' group of bottom dialog box. Pick any line of the Dimension to delete it.

DXF Drawing of Assembly and Tools

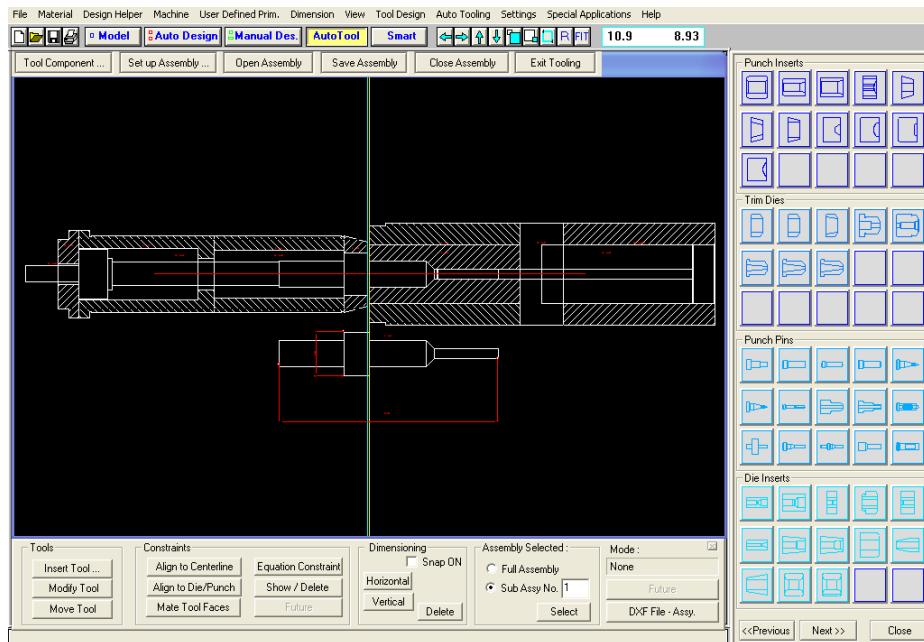
To create a DXF drawing of the assembly, Click on ‘DXF File –Assy.’, a dialog box as shown above will appear. If you want to put the assembly drawing in your Block drawing, ‘Browse’ to select the Block Drawing. Press ‘Save’ button to use Windows ‘Save As’ dialog box for choosing the directory and file name and saving the DXF drawing in it.

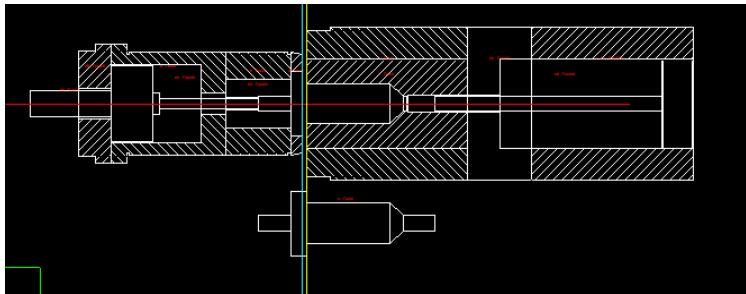


To create DXF drawings of individual tools, use ‘Modify Tool’ option to bring up the ‘Modify’ dialog box of the tool. In this dialog box, you have the ‘Create DXF Drawing’ button. Use this to create the DXF Drawing of the tool.

An Example of Tooling Assembly

The assembly shown below was created using the procedures explained above. The part drawing below the assembly is actually a tool. Equation constraints were set up to relate the dimensions of assembly tools with this part. As the part is changed, the assembly tools change to accommodate the part as shown below





10.4 Phase 3 - Auto Design of Tooling from Part Progression (Under Development)

Part Forming Sequence -> Tooling Assemblies -> Tooling Components.

In the next and final phase, methods will be developed to generate/modify tooling assemblies automatically based on changes in the Part Forming Sequence. The sequence design can be from NAGFORM's Auto Design or from 'Design by Command' procedure.

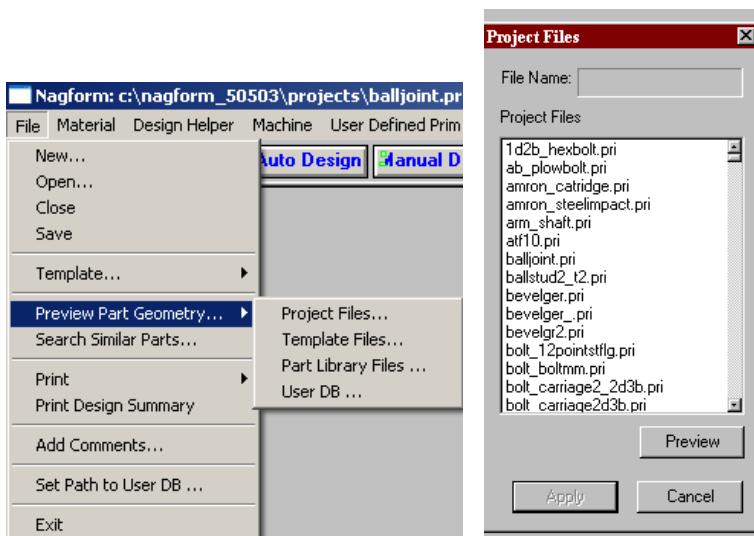
As the part is changed, its forming sequence would change and its tooling assemblies and tools will change automatically. Complete automation in design of tooling can be accomplished for 'Family of Parts' where the part features do not change. Only its dimensions change. Partial automation based upon type of tooling would be possible for any part shape.

11. General Features

11.1 Preview of Part Geometry in Various Files

In the current version of NAGFORM, User can preview the part geometry in various files including ‘Project’ files, ‘Template’ files, ‘Part Library’ and ‘User DB’ files. To preview a file:

1. Select ‘File’ from Main Menu
2. Select ‘Preview’ and then ‘Project’, ‘Template’ , ‘Part Library’ or ‘User DB’
3. A dialog box would appear on the right hand side of the screen. This dialog box would have a list box with names of all the files stored under the selected category.
4. Select the particular file you wish to preview and then press ‘Preview’ button.
5. The part model will be displayed in the graphics window area.
6. After you have viewed the part, click ‘Cancel’ to close the dialog box.
7. Preview selection can only be made when no file is open.



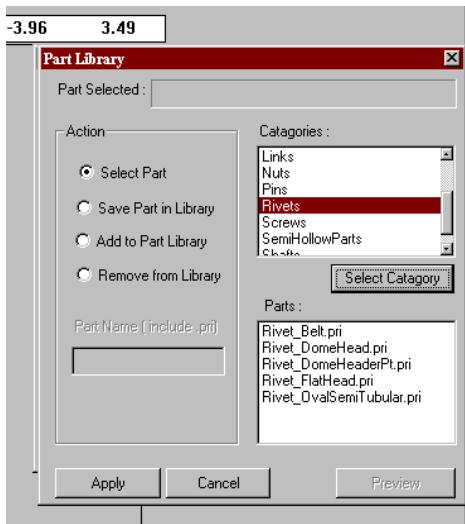
11.2 Part Library

In the current version of ‘NAGFORM’, a part library has been created with the objective of storing part models of commonly cold-formed parts. We would continually add part models to this library so that the users can start from an existing part shape to save time in creating a part model. The users can also add their own parts to this library if they want.

You can use ‘Preview’ feature of the program to preview the part geometry in any of the files in the Part Library.

To bring in geometry from a part in Part Library:

1. From ‘File’ menu, select ‘New’ or click the ‘New File’ button.
2. In the ‘New File’ dialog box, click the ‘Part Library’ button.
3. This would bring up the Part Library dialog box. In this dialog box, there is a list box with names of categories under which various parts are stored. Select the category from the list and press the ‘Select Category’ button.
4. This would bring names of all parts stored under the selected category in the lower list box. Select the part from the list and click ‘Apply’ button.
5. Press Cancel button to close this dialog box and return to the ‘New File’ dialog box.
6. Proceed to start a new project file.



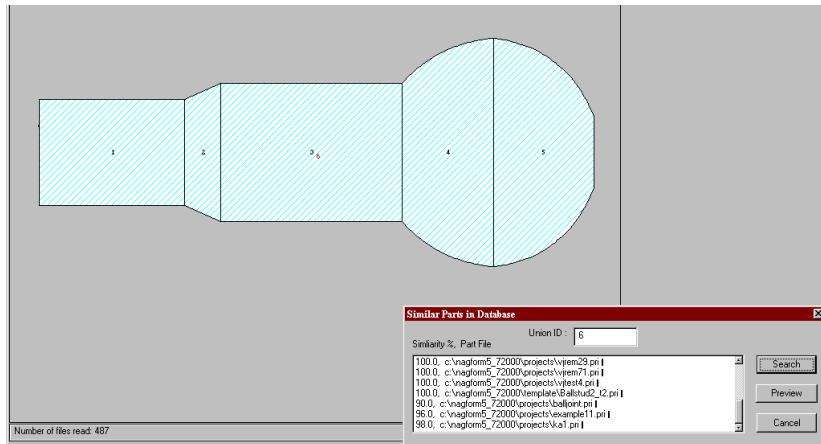
11.3 Search for Similar Parts in Database

In the current version, the user can search for parts similar to the one he/she wants to design. The purpose of providing this feature is to prevent designing a part that has been designed in the past. This capability would also help the designer to study existing designs for similar parts. Essentially, this feature would prevent ‘redesigning the wheel’.

The ‘NAGFORM’ program searches for similar parts in all the files including the project files, template files, part-library files and User DB files.

To search for similar parts, you must be in the ‘Model’ module and the geometry of the part to be designed should have been created.

1. Select ‘File’ from the main menu
2. Then select ‘Search Similar Parts’
3. A dialog box as shown below would appear.
4. Enter the part ID if not in the edit box.
5. Press ‘Search’ button. The program would search through all the files and lists the files with a ‘Similarity’ index. An index of 100 means that the part in database is exactly the same. An index of zero means no similarity. The list box shows only those parts that have a similarity index of 60 or more.
6. If you want to preview any of the similar parts, select that part in the list box and press ‘Preview’ button.



11.4 Six Lobe External and Internal Primitives

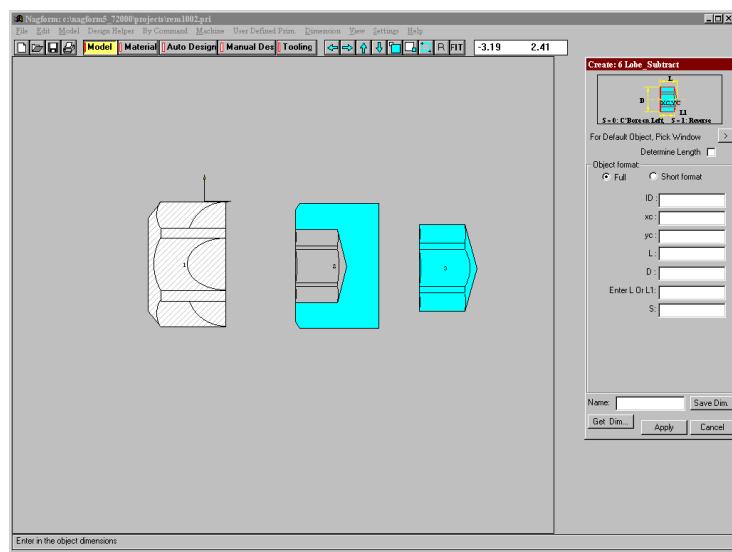
In the current version of NAGFORM, we have three (3) 6 Lobe external and internal primitives as follows:

LOB: External 6 lobes primitive

ILB: Internal 6 Lobe primitive

LSU: 6 Lobe ‘Subtract only’ primitive.

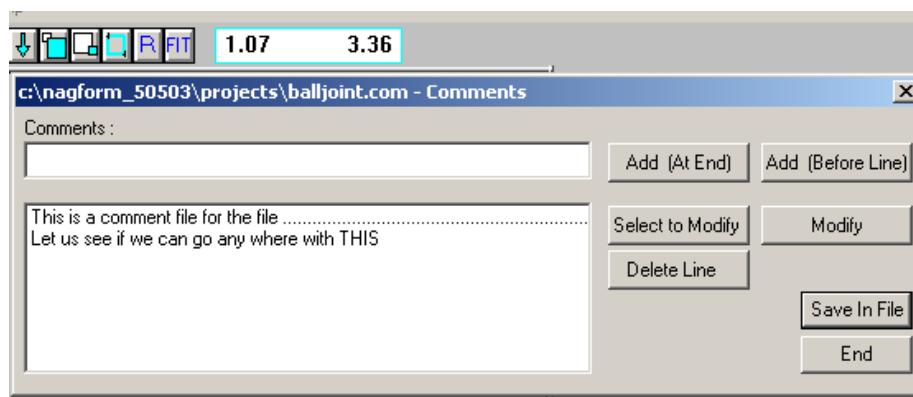
These appear in the group of other primitives and can be used like other primitives. LSU (6 Lobe ‘Subtract only’) primitive should only be used when 6 Lobe impression is to be subtracted from a Union.



11.5 Adding ‘Comments /Notes’ in NAGFORM

In NAGFORM, the user can now add any comments/notes on the project. These notes/comments are written in a (.com) file with the same name as the name of the project (.pri).

To read /write /modify/delete comments, go to File Menu and choose ‘Add Comments...’ selection. Note that you must open the project file first before you can get to the comments. If no comments were added before, a new file with the (.com) suffix will be created. The dialog box shown below would appear. If some comments were added previously, these comments are shown in the list box.



To add new comments, type the comments in the ‘Comments’ edit box and click ‘Add (At End) to add at the end of all lines in the list box. If you wish to add the comments before any line in the list box, select that line in the list box, type comments and then click on Add (Before Line) button. The typed comments will be added before the selected line.

To modify any line in the list box, select that line and press ‘Select to modify’. That line will now appear in the ‘Comments’ edit box. Change the line and press ‘Modify’ to replace the selected line with the changed comments.

To delete any line, select that line in the list box and press ‘Delete Line’. It will be deleted from the list box. Note that you must press ‘Save in File’ button if you want these changes to be saved in the comments file.

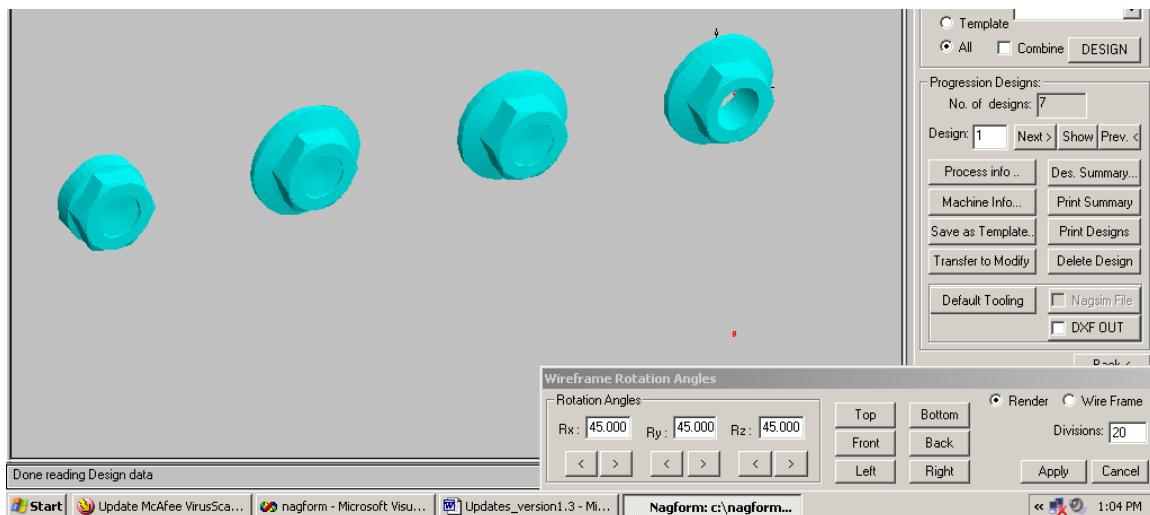
11.6 Rendering

BREP (Boundary representation) means representation of the surface of 3D solid. In NAGFORM, all the primitives and the unions are internally considered as 3D solids. Hollow parts are also 3D solids. No surface representation is needed for the design calculations in NAGFORM.

However, for simulation of metal flow, boundaries consisting of internal and external surfaces are needed. With the objective of exporting 3D surfaces for simulation in NAGSIM.3DS, our upcoming simulation program, BREP representation of primitives and unions in NAGFORM has been developed. Eventually this representation will be used to define the surfaces of the parts and the tools in NAGSIM.3DS.

BREP representation has another use. The surface definition can be used to render any part. This has been done in NAGFORM for parts and progressions.

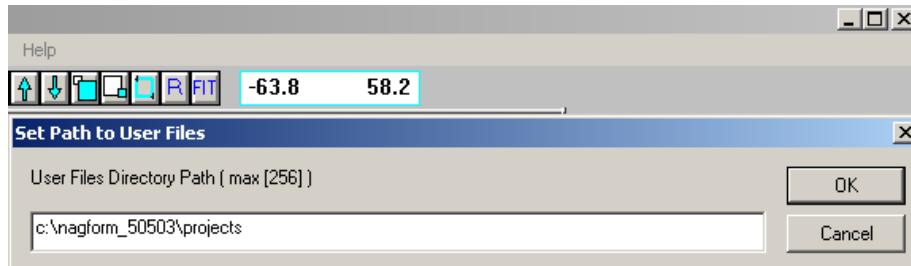
To render any part (Primitive or Union) or groups of parts e.g. design sequence, click on ‘Ren’ button in the small tool bar at the upper right corner of the ‘Graphics’ area. The part(s) will be rendered and a dialog box will appear at the bottom as shown. Using this dialog box, you can change the view by rotating the view point in the 3D space.



When done, click on ‘Cancel’ button to end rendering. Note that ‘Fit’ graphics button does not fit the rendered parts in the Graphics Screen. You would need to use ‘Zoom out’ and then ‘Window (Rect)’ button to select the parts to view.

11.7 User's Database: Search of Similar Parts and Designs

In search for similar parts, NAGFORM looks for similar parts in ‘Projects’, ‘Templates’ and ‘Part Library’ directories. Another place it would search is a User defined directory. The path to this directory is set by the user. To set the path, Select ‘File’ menu, then ‘Set Path To User DB’. A simple dialog box shown below appears where the user can type the location of the directory of his/her files. This location will be included in the search for similar parts.



In Preview, the User can preview the files at this location. Also during opening of a new file, a copy of any part in User’s database can be brought it.

11.8 Tool Design Help

A ‘Tool Design’ option in the main menu of NAGFORM program has been added. There are two selections under this menu option, as shown below in Figure 6, namely:

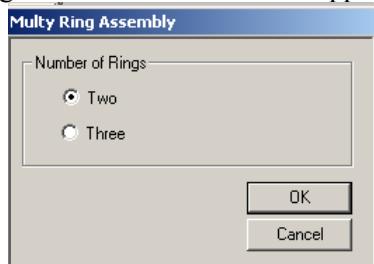
- Multi Ring Assembly: For designing two and three ring press-fit assemblies
- Interference Fit Calculation



Multi Ring Assembly:

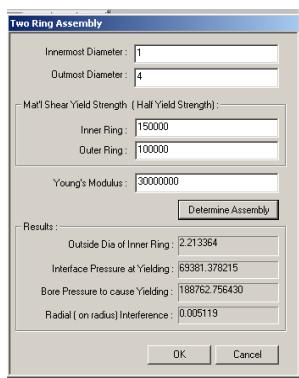
This is based on ‘Cold Forging Data Sheet No 70003’ issued by Cold forging Group of the Institute of Sheet Metal Industry. This is an old publication but we chose it as it has a much simpler input requirement. This provides a starting point for determining the ‘Insert’ and ‘Casing’ dimensions. These dimensions as well as the press fit values can be optimized for any particular part using ‘Interference Fit Calculation’ along with NAGSIM.2D simulation of tool stresses.

To design a multi-ring (two ring or three ring) assembly, select this option from the ‘Tool Design’ menu. A dialog box shown below would appear. Select ‘Two’ or ‘Three’ radio button and then click OK.



A dialog box for two ring or three ring assembly would appear. Shown below is the dialog box for two ring assembly (one insert and one casing)

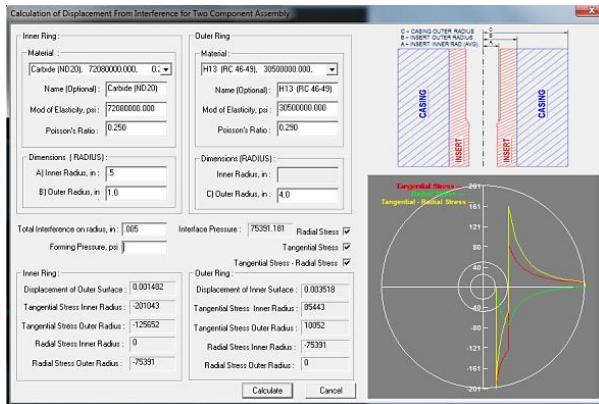
Enter the data:



1. Inner (bore) diameter of the insert. If insert has more than one inner diameter, take the largest bore diameter.
2. Outer diameter of the casing.
3. Shear strength of the insert material (Assume that shear strength = one half of yield strength of the material)
4. Shear strength of the casing material.
5. Enter Young's modulus (Same as Modulus of Elasticity) of the material. If the Young's modulus of insert and casing are different (ex. insert is made of carbide and casing of HSS or carbon steel), you can put in modulus of any material. However, you cannot use the results on Radial Interference.

Click ‘Determine Assembly’ button. The analysis results are shown in the dialog box. The primary result that you would get from this analysis is the ‘Outer Diameter of the Inner Ring (insert)’. Regarding the stresses in the insert and casing from any interference fit, you should use the second selection of Tool Design Menu, namely, ‘Interference Fit Calculation’.

Interference Fit Calculation (Same as in NAGSIM.2D) - Calculation of interface pressure and displacement for 2- piece Shrink-Fit Assembly. When an insert is press fit into a casing, the amount of interference (press fit) is sum of the inwards displacement of the insert and outward displacement of the casing. To include stresses in the insert due to press fit in elastic stress analysis performed by NAGSIM.2D, a ‘Displacement’ boundary condition is specified on the outside surface of the insert in NAGSIM.2D. However, the displacement of insert surface is not known. We must first calculate it from the interference fit of the assembly.



In this version of NAGFORM, calculation of the displacement of the interference on the assumption of press fit of two infinitely long cylinders is provided.

From main menu, select ‘Tool Design’, then ‘Interference Fit Calculation’. The dialog box shown above would appear. Fill in the information on modulus of elasticity, Poisson’s ratio, and dimensions of the insert and casing. Note that dimensions are radii not diameters. Enter the interference amount on radius, then click ‘Calculate’ button. The program would calculate the displacement and stresses in the insert and casing due to interference fit.

12. Technical Papers

12.1 Create and Use a Historic Knowledge Database of Parts and Forming-Sequence Designs

Background

Most of the companies in metal forming business have manufactured a large number of formed parts in the past. The knowledge to form such parts can be found in

- Sample parts saved during the part run
- CAD drawings of forming sequences
- Tool drawings.
- Experiences of personnel who worked on the jobs.

A significant part of this knowledge is lost when the drawings are not kept or are not updated and the experienced personnel become unavailable. Without some sort of computerized system, it is almost impossible to know if any part similar to the part in question was manufactured in the past. This is especially true if a similar part was manufactured a number of years ago. If the organization has many divisions, it is quite likely that the forming knowledge is not shared between various divisions and there is duplication of design as well as manufacturing effort.

Knowledge Database of Forming Parts

The ideal situation is when we can employ all the knowledge gained in previous jobs to the design of forming the part in question. This requires that

- Previous forming knowledge is saved and stored in a database
- This knowledge can be accessed easily.

Purposes of a Historic Knowledge Database

- Prevent redesigning a part that has been formed previously.
- Prevent redesigning the forming process of a part from scratch when similar parts have been designed and formed successfully in the past.
- Prevent duplication of designs when a company has many divisions.
- Prevent quoting a job based on a forming sequence that is not usable.
- Use previous designs to help design new parts and prevent pitfalls.
- Use previous designs to help quote the current part.
- Use existing tooling when it can be used for the new part.

Historic Knowledge Database of Parts and Designs in NAGFORM

NAGFORM has following capabilities that allow the development and use of a knowledge database:

- Search for Similar Parts: NAGFORM searches for similar parts based upon the features of a part.
- Design Template: You can create a template of forming design using the ‘Auto Design’ or the ‘Design by Command’ feature.
- DXF IN and DXF OUT. Designs can be brought in or exported in DXF format that is compatible with AutoCad and most other CAD programs.

Procedure for Creating a Historic Database of Parts and Sequence Designs in NAGFORM

This procedure would create a NAGFORM project file with the part geometry stored in ‘Model’ section and its forming sequence stored in ‘Manual Design’ section of the file. You would need to repeat this procedure for each part for which you have the forming design information.

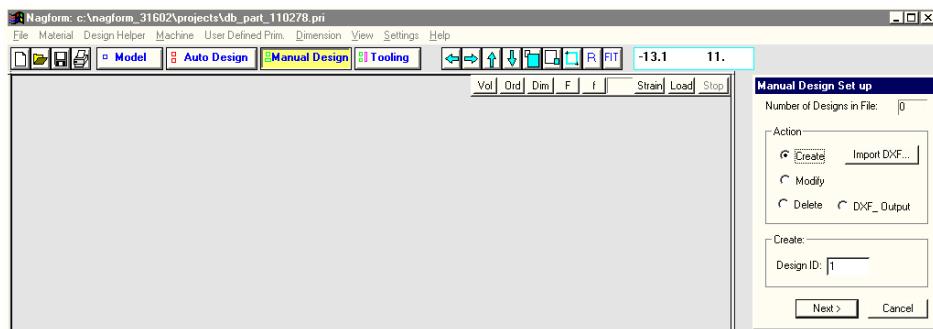
1. Save the existing forming sequence of a part in DXF format
 - As shown below, the drawing should contain only the forming sequence. All other features such

as drawing border, dimensioning etc should be removed.

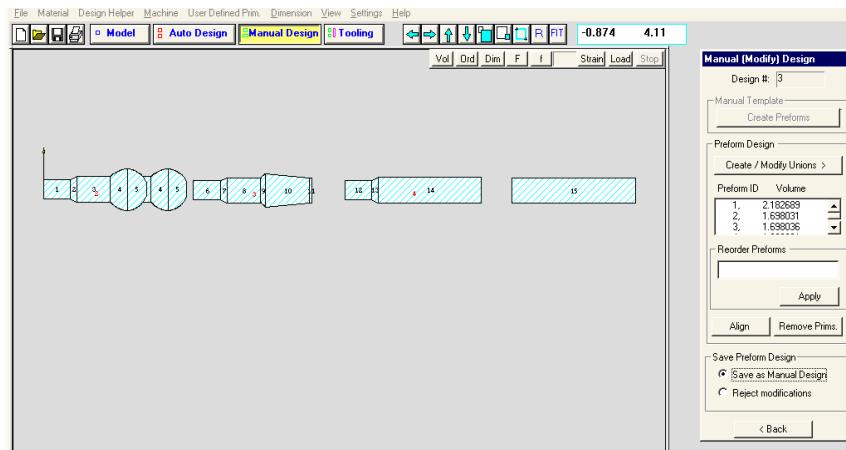
- Remove the centerlines of the parts.
- Arrange the sequence of parts in a horizontal format as shown. The parts do not have to be aligned exactly.
- Save this drawing in AutoCad R12/LT2 DXF format.
- Store this DXF drawing in a directory specially created for this purpose.



2. In NAGFORM, open a ‘New’ file, naming it appropriately to help you identify the part.
3. Click on ‘Model’ button. This would bring the dialog box shown below. Click on ‘DXF IN’ button near the bottom of this dialog box. This would bring up another dialog box. This would lead you to Windows ‘Open’ dialog box. After changing directories as needed, select the DXF file of the part.
4. Clicking to open the file should bring the DXF sequence in the NAGFORM model. You may need to press ‘Fit’ button to view the sequence.
5. You should now have the forming sequence converted to NAGFORM primitives and unions. Delete all unions except the final part union. This is the only part that would be saved here in the Model section.
6. Click on ‘Save’ button to save the file.
7. If the program did not bring in the complete part, you would need to modify the part union in NAGFORM program to complete its geometry.
8. Close the ‘Model’ and click on ‘Manual Design’ button. Choose ‘Design Manually’ option (not the Design by Command). This would bring the ‘Manual Design Set-Up’ Dialog Box as shown.



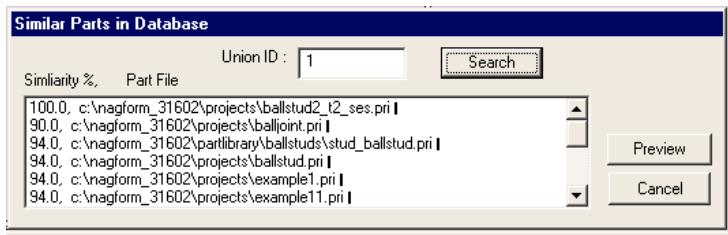
9. Select ‘Create’ radio button. This would enable ‘Import DXF’ button. Click on this button.
10. Go thorough the same procedure as before to bring in the forming sequence. This manual design would require the following modifications:
 - The manual design has duplicate final part unions. You would need to delete one of the part unions.
 - The last ‘Rod’ blank may not be converted to a union by the program. If it does not show an ID written in Red, it has not been converted. You would need to change the ‘Rod’ blank prim to a union.
 - If importing the DXF drawing did not bring in all of the geometry, you would need to complete the manual design.
- These modifications can be made as described in next few pages.
11. Click ‘Next’ button in the ‘Manual Design Set Up’ dialog box. This would bring up the Manual (Modify) Design dialog box shown below. Press the ‘Create/Modify Unions’ button to go to ‘Modify Union’ dialog box.



12. At the bottom of ‘Modify Union’ dialog box, click on ‘Back to Model’ button to go to the ‘Model’ module. Make all the modifications as you do to any part.
13. In ‘Modify Union’ dialog box, click ‘Back to Design’ button to come back to Manual (Modify) Design dialog box shown above. Enter the IDs of the unions representing the sequence in the ‘Reorder Preforms’ edit box. Clicking on the ‘Apply’ button would rearrange the forming sequence in a typical vertical format.
14. Repeat the above procedure to create a project file for each part manufactured in the past for which you have the information.

Using Historic Database of Parts and Designs

- For any new part, create the part model as done normally.
- Go to ‘File’ menu and select ‘Search Similar Parts’ option. This would open the ‘Similar Parts in Database’ dialog box shown below.



Click on ‘Search’ button to search for similar parts. The program would bring up project files of those parts for which similarity index is more than 50 %. Two exactly similar parts have a similarity index of 100 %. The program searches for similar parts in ‘Projects’, ‘Part Library’, ‘Templates’ and ‘User DB’ directories. You can preview the part in any file by selecting the appropriate file in the list box and pressing the ‘Preview’ button. This would bring up the part in that file. When you are done previewing, make sure you note down the names of files you want to investigate further for sequence designs. Press ‘Cancel’ button to close this dialog box and return to the original part.

- If your search bring up a part done in past which is ‘very’ similar (same shape but different dimensions), and a design template for this similar part is available, you can use the design template to design the forming sequence for the new part.
- If the current part is not ‘very’ similar to any part done in the past, review the forming sequences of similar parts to get ideas on forming the current part. As explained earlier, the forming sequences are stored in the ‘Manual Design’ section of a file in the database. For review, you can get DXF output of forming sequence from the Manual Design Module.

12.2 Sequence Design of Specific Components

In general, the following would help to get designs:

- Create a copy of the part and remove small details like corners, fillet radii, chamfers in the part copy. Design this part copy.
- Use ‘Rod’ prim in place of free flow ‘Pancake’ prim.
- For hollow parts, guide the program on location of the punch out by making the bore slightly smaller at that location. Note that the punch out bore has to be the smallest bore.

• Fasteners

- Use reusable templates.
- Save commonly used prim dimensions for Hex, Lobe etc. in Prim tables. In ‘Create’ dialog box of Nonsymmetric prims, there is a provision to save the entered dimensions in a table for reuse at another time.
- Small chamfer at part end (point) sometimes creates problems because NAGFORM does not allow forming this feature if the reduction is not within open extrusion limits. Make a copy of the part and change the ‘End Point’ dimension to get the designs on the part copy.
- For Parts that have hex with taper flange, use PFG prim and not POE hex prim.
- Use ‘Design Helper’ to reduce number of operations.

• Family of Parts

- Use Design templates

• Nonferrous Cold formed Components

- Need to change Default Rules in the ‘Rule’ file as current default values are written for steels. If the nonferrous material is much softer than steels, increase allowable limits on extrusion ratios to allow forming of thinner parts etc. Backward extrusion rules also need to be changed to allow deeper backward extrusion and forming of thinner tubular sections.

• Shafts

- To begin with, create a copy of the part and remove some of the small details like radii, chamfers etc. With too many small details, number of possible solutions can become excessive. Also you may get more number of operations to form the part with many small details
- In Auto Design, allow more than desired number of operations. Our logic does not combine more than two processes. You may need to combine operations yourself in manual design.

• Hollow Parts

- Use subtracted primitives to create hollow parts.
- If you do not get any design, create a copy and remove small details such as end chamfers, corner and fillet radii.
- Our logic places punch-out at part ends or in its middle. If you want punch-out at a particular location, make the bore minimum at that location.
- If the part has a long bore, divide the single long bore into two short bores, one very slightly smaller than the other. The program can then form these in two separate operations.
- Make sure you set the length/dia. ratio of punch-out correctly otherwise you would not get any design. Design Helper would inform you if your rules are not set correctly.

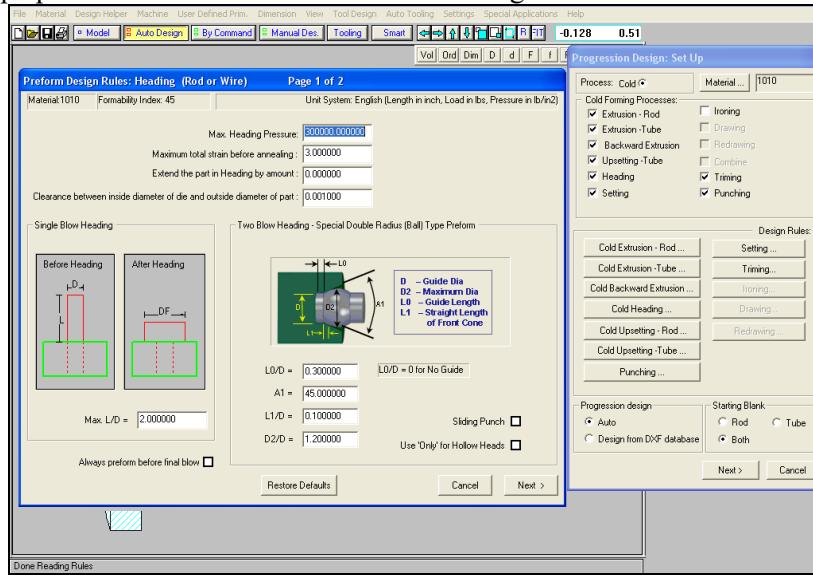
12.3 NAGFORM 1.3 Enhancements

In the latest version of NAGFORM, a key emphasis has been placed on the ‘User Friendliness of the program’ and ‘Improvements in the Sequence Design’. Listed below are the main enhancements in Version 1.3 of NAGFORM.

- 1. Auto Design Enhancements:**
 - a. In ‘Auto Design’ of Forging Sequence, a ‘Ball’ type preform has been added for ‘Heading’ part shapes that generally use curved preform instead of a ‘Cone’ type preform.
 - b. The program automatically searches for similar parts and sequence designs in all existing project and template files. Thus, before designing, the user knows the similar parts that have been designed in the past and the sequence designs that are available.
 - c. The user now has the ability to flag an auto-design as ‘Preferred Sequence Design’ or ‘Undesirable sequence design’. Auto Design of similar parts can now be done using ‘User’ preferences of sequence designs in addition to those given by the ‘Auto Design’. Those marked ‘Undesirable’ are omitted from the Auto Design.
- 2. Functionality Enhancements: In ‘Model’ and ‘Auto Design’ sections of NAGFORM,** ‘Floating Menus’ using Right Click of mouse have been added to allow quicker changes in part Model and access to some of the Auto Design functions.
 - a. In ‘Model’, actions on part model such as ‘Modify’, ‘Copy’, ‘Delete’, ‘Explode’ and ‘Save File’ can be performed by right clicking on the Primitive/Union and selecting the option from the floating menu.
 - b. In ‘Auto Design’, floating menu allows ease in selecting ‘User preferences’, ‘Save as Template’, ‘Show Process Info’, ‘Save as Preferred Design’ and ‘Save File’.
- 3. Graphical Input for the interactive ‘Design by Command’:** ‘Design by Command’ procedure allows the user to create personalized forging sequences. A ‘Graphical Input and Help’ has been added in Version 1.3 to allow for a quick and easy creation of forging sequence / design template using Design by Command. For ‘Standard’ product lines of Fasteners and formed parts like Hex bolts and Spark Plugs, the User can easily create reusable Sequence Designs Templates using ‘Design by Command’ Graphical procedure.
- 4. Surface Area of Part:** NAGFORM now has the capability to calculate the surface area of any part including the model and preforms.
- 5. Other Improvements made are:**
 - a. In ‘Model’, the buttons representing primitives have ‘Pictures’ in place of ‘Text’ for ease in selecting a primitive type.
 - b. In tooling assembly of ‘Auto Tooling’, dimensioning of ‘Radius’, ‘Angle’ and ‘Grinding Symbol’ has been added.
 - c. Material database of NAGFORM has been updated to include all materials that are in the NAGSIM.2D program.

12.3.1A Auto Design Enhancements: ‘Ball’ Type Heading Preform

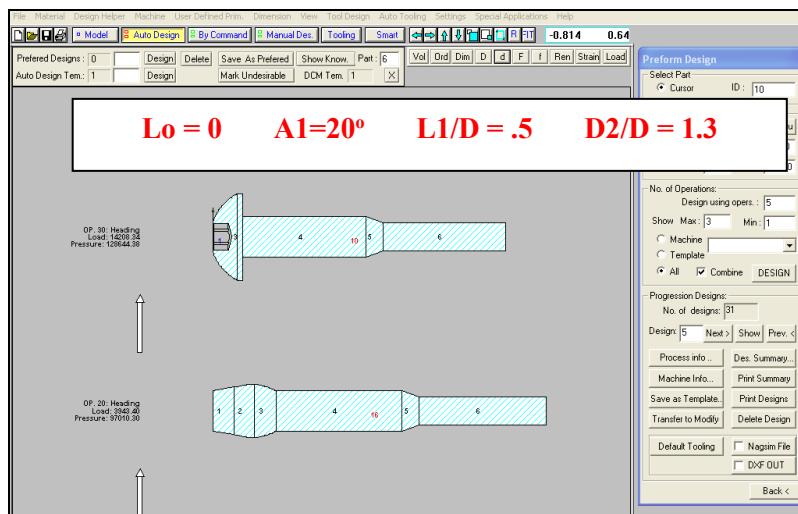
In ‘Auto Design’ of Forging Sequence, a ‘Ball’ type preform has been added for ‘Heading’ part shapes that generally use curved preform instead of a ‘Cone’ type preform. As shown, the dimensions of this Ball Type preform have been added In the ‘Heading Rules’.



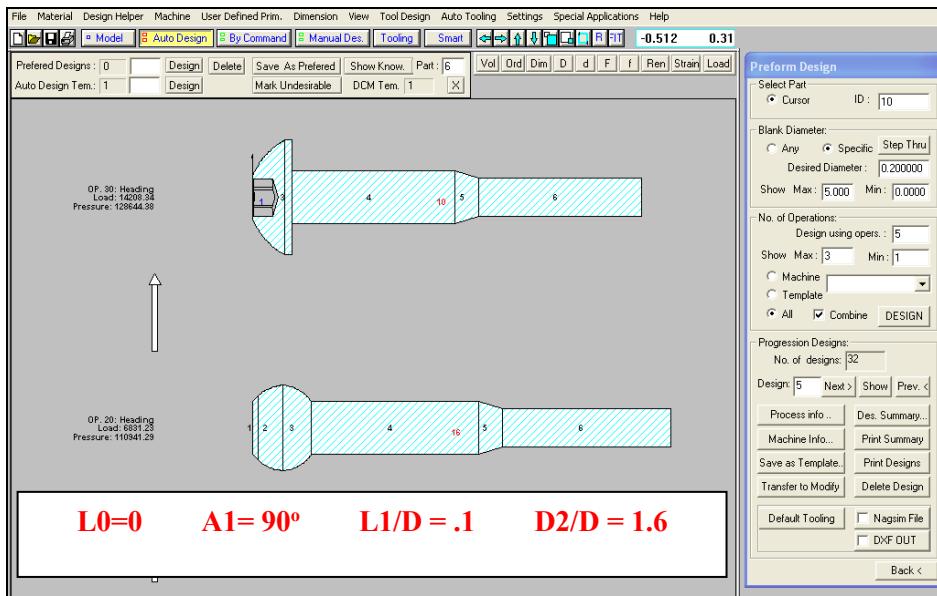
The ‘Ball’ preform has a straight guide portion of Length ‘L0’, a taper length ‘L1’ with Angle ‘A1’ and a ball (two ‘Corner’ primitives) with maximum diameter represented by D2. All the dimensions are given as ratio of these dimensions to the guide diameter ‘D’.

When no guide is needed, L0/D can be set zero. By changing the Angle A1, L1/D and D2/D dimensions, various shapes of the ‘Ball’ preform can be obtained as shown in example below.

To use this type of preform for only those parts that have a hole in the head (Such as Pan-head screws), select ‘Use Only for Hollow Heads’. If this is not selected, the program would give ‘Ball’ type preform for both solid and semi-hollow parts. The heading rule of maximum ‘L/D’ ratio for ‘Cone’ Type preform also applies to this preform. For greater L/D ratio, if a sliding punch is intended, check ‘Sliding Punch’ box.



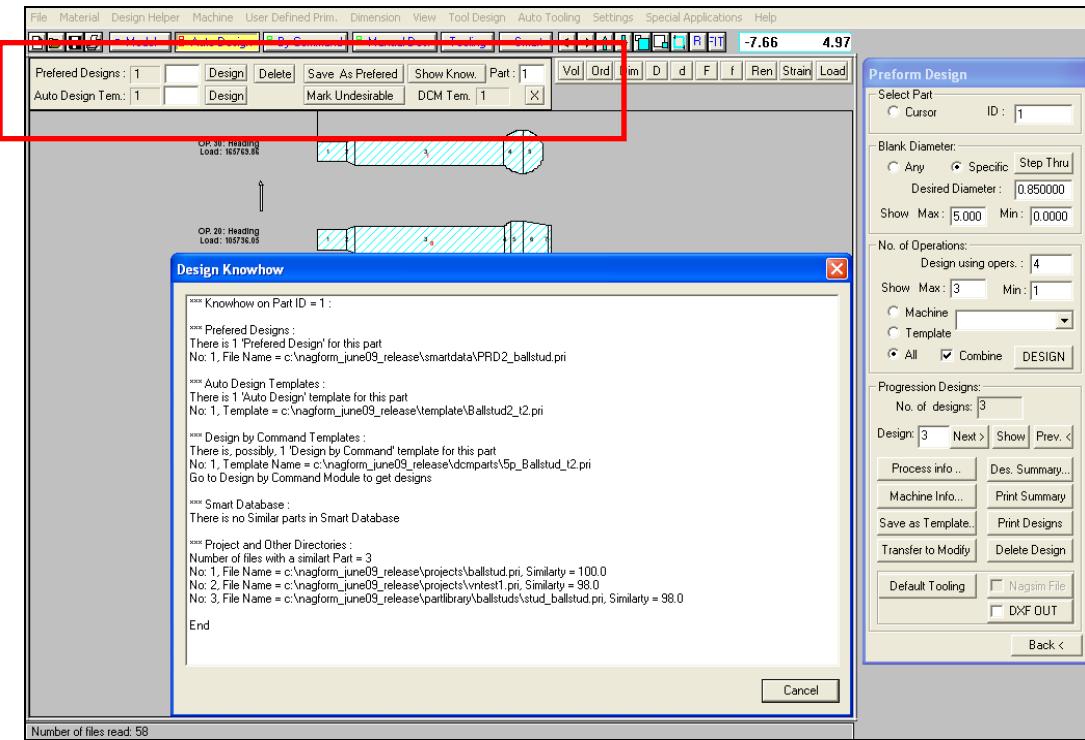
EXAMPLE USING BALL TYPE PREFORM



SAME EXAMPLE WITH DIFFERENT PREFORM DIMENSION SETTINGS

12.3.1B - Auto Design Enhancements: Automatic Search of Similar Parts and Sequence Designs

In this version, when the User goes to ‘Auto Design’ for obtaining alternative sequence designs of a part, the program automatically searches for similar parts and sequence designs in all existing project and template files. This includes search for ‘Design by Command’ session files and similar parts in ‘Smart’ database. Thus before designing a sequence of a part, the User would know about the similar parts that have been designed in the past and the sequence designs that are available. The results of search done by NAGFORM program are displayed in a ‘User Preference / Knowledge’ dialog box that opens at the top of the Graphics area as seen below.



The dialog box shows number of ‘Auto’ designs, ‘Preferred’ designs (discussed later) and DCM designs available for this part. To get further details of the search, click on ‘Show Know.’ button. The Design Know how dialog box would appear as shown above. This gives the location of the files that have similar parts and sequence designs. It includes the search in ‘Smart’ database.

12.3.1C. Auto Design Enhancements: User Defined ‘Preferred’ and ‘Undesirable’ Sequence Designs

In ‘Auto Design’, NAGFORM program gives all possible sequence designs that satisfy the Forming Rules and are within the parameters of wire size and number of operations set by the User. The sequence designs are not directly influenced by the liking or disliking of the User.

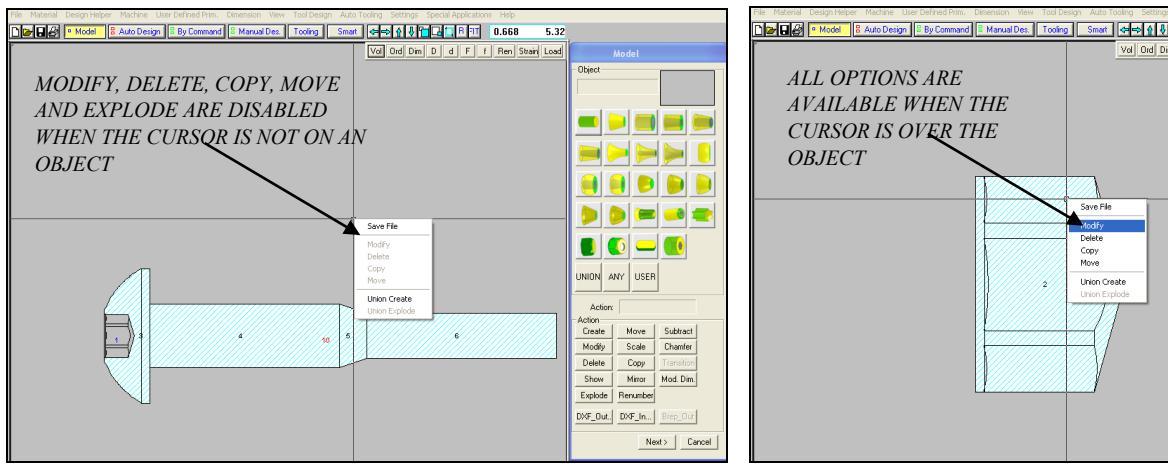
In this version of NAGFORM, ‘User Preferences’ regarding sequence designs for a particular part have been added. During ‘Auto Design’, the User can save any design as a ‘Preferred Sequence Design’ or as an ‘Undesirable Sequence Design’. Any number of designs can be saved as Preferred or as Undesirable. During Auto Design, the program omits the ‘Undesirable’ sequence designs from the alternate designs.

To save any sequence design as ‘Preferred’ design, display that design in the graphics window and then press the button ‘Save As Preferred’. The sequence design is saved as ‘Preferred Design’ in ‘SmartData’ directory of NAGFORM program. There is no need to assign any name to this sequence design as NAGFORM automatically assigns a name based on the existing preferred designs in the ‘SmartData’ directory. When a design is saved as ‘preferred’, the number of preferred designs in ‘User Preferences / Knowledge’ dialog box is updated.

To delete a preferred design, enter the design number in ‘User Preferences/ Knowledge’ dialog box and click ‘Delete’. Use similar procedure to mark any Sequence Design as undesirable.

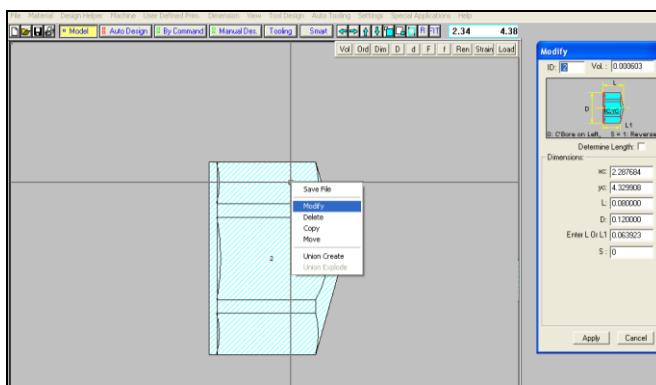
12.3.2A. Floating Menu in 'Model'

In the Model screen, a ‘Floating Menu’ has been added to allow quicker changes in part Model. The menu appears only when the user is in the ‘Model’ section. The menu shown below appears when the user Right-Clicks’ the mouse.

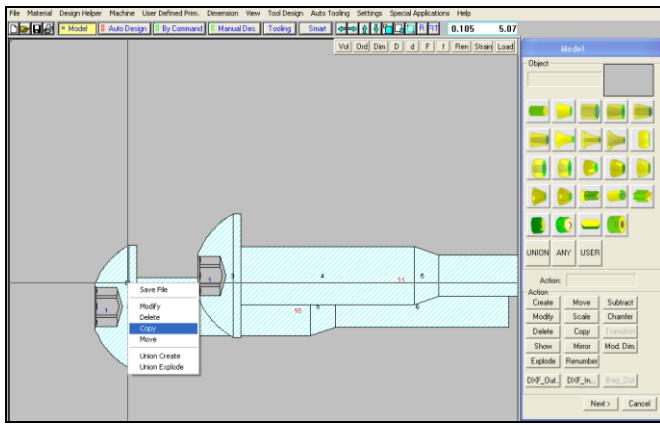


The various selections in this menu include:

- Save File – Select to save the NAGFORM File.
- Modify – Select to modify the primitive / union. This selection is only enabled when the user has the mouse pointed to a primitive / union. Once selected, the dialog box for modifying the Union / Primitive will appear as seen below. *** Please note that the ‘Modify’ action can also be selected by double clicking on the Union/ Primitive ***



- Delete – Select to delete a primitive / union. This selection is only enabled when the user has the mouse pointed to a primitive / union.
- Copy – Select to copy a primitive / union. When selected, a copy of Union or Primitive is created and placed next to the part (As seen in the next Figure). This selection is only enabled when the user has the mouse pointed to a primitive / union.

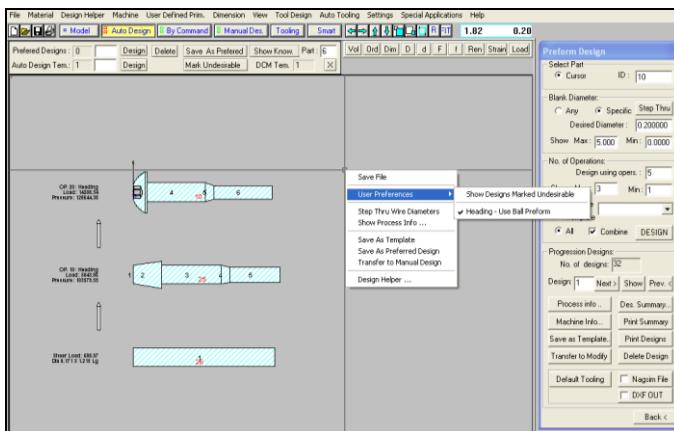


- Move – Select to move a primitive / union. Once selected, the user can then move the part by moving the cursor. This selection is only enabled when the user has the mouse pointed to a primitive / union.
- Union Create – When selected, the ‘Union Create’ window appears.
- Union Explode – Select to explode an existing Union. This selection is only enabled when the user has the mouse pointed to a union.

12.3.2B. Floating Menu in ‘Auto Design’

In the ‘Auto Design’ section, a floating menu has been added for ease in performing certain tasks of the Design Module.

- ‘Save File’ saves the project file.
- ‘User Preferences’ allows a User to select to show or omit ‘Undesirable’ designs. The user can also include or omit preform designs with ‘Ball’ type preforms.
- ‘Step Thru Wire Diameters’ starts the designs with different wire diameter.
- “Show Process Info” opens the dialog box that shows further information such as area changes, strain etc in the forming operations.
- “Save as Template” is for saving any design as an Auto Design Template.
- ‘Save as Preferred Design’ saves the design being displayed as a preferred design.
- ‘Transfer to Manual Design’ transfer the design being displayed to Manual Design for making changes manually.
- ‘Design Helper’ opens the Design Helper dialog box for help in getting a design. Design helper should be used when the Auto Design is unable to give a design.



12.3.3 ‘Design by Command’ Graphical Input and Help

‘Design by Command (DCM)’ procedure in NAGFORM allows the User to create his/her own forging sequence design manually. Sequence designs created using DCM have the following advantages over designs created without it:

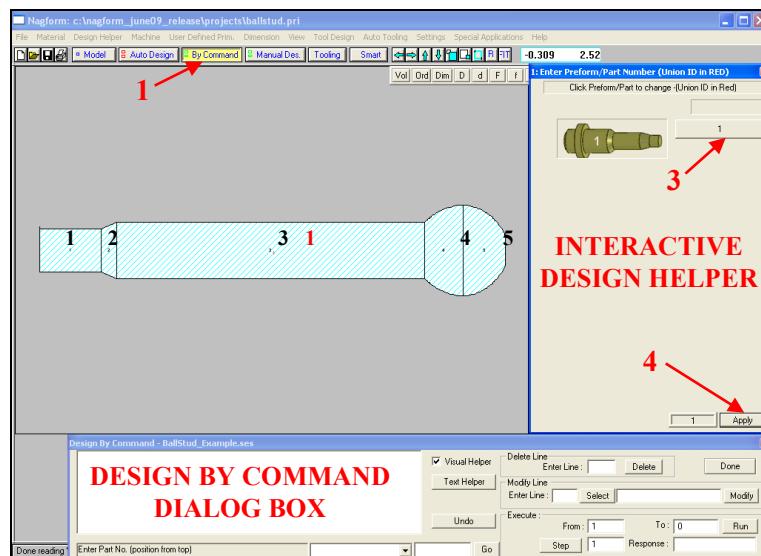
- Dimensions of the forging preforms are calculated automatically with proper distribution of volumes.
- Forging Loads are calculated automatically.
- Forging Design is checked against ‘Forming Rules’
- Sequence Design created through DCM are reusable templates that can be used for designing ‘Similar’ parts within couple of minutes.

‘Design By Command’ is ideal for automating and standardizing Forging Sequence designs of Product lines such as Fasteners (Standard and Special) as well as cold formed parts such as spark plugs.

DCM procedure involves developing the sequence design interactively through a series of questions and answer session in NAGFORM. In the previous version, inputs from a User were in form of an abbreviated ‘Text’ format in response to multiple choice questions. This required the User to be thoroughly familiar with the multiple choices. For a new User, it was difficult to use DCM without spending some time learning through examples. To make it simpler, additional ‘Graphical Input and Help’ windows have been included in the DCM procedure of creating a Sequence Design. This is explained below with an example.

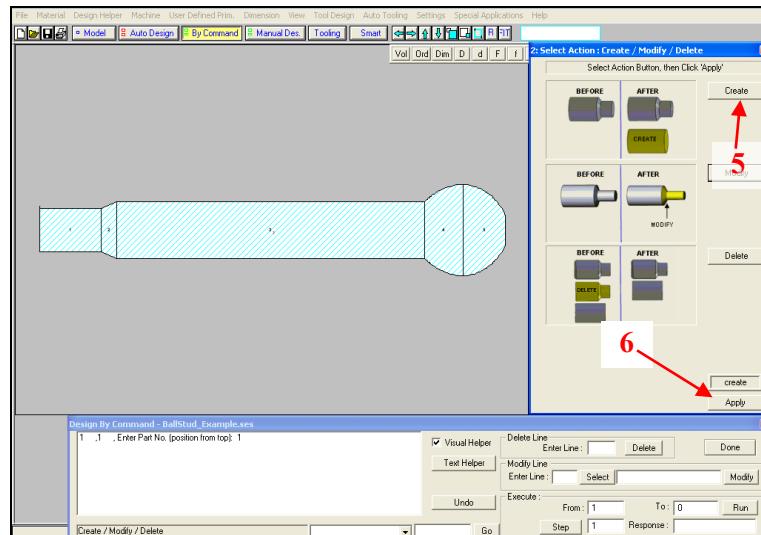
Example – This example depicts a simple Heading Process of Primitives 4, 5 in two Stations.

1. The Design by Command section is initiated by clicking on the ‘By Command’ button.
2. Along with the Standard Design by Command dialog box, an Interactive Design Helper Screen will appear.
3. Click on Union ID Number (in this case 1)
4. Click on ‘Apply’ to proceed to the next window.

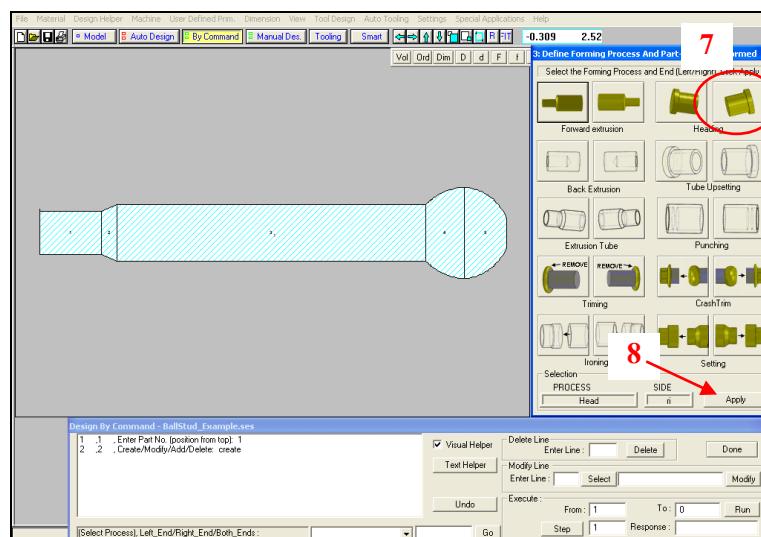


Example (Continued)

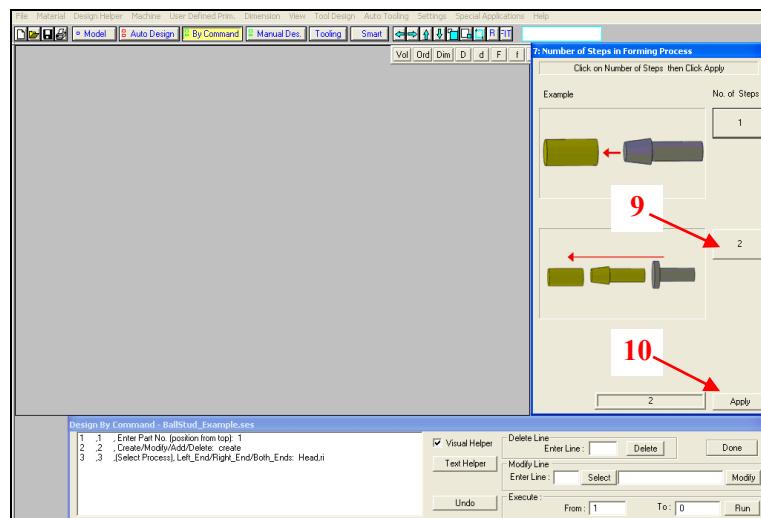
5. Click on ‘Create’ to create the next forming station (working backwards in Sequence Design).
6. Click on ‘Apply’ to proceed to the next window.



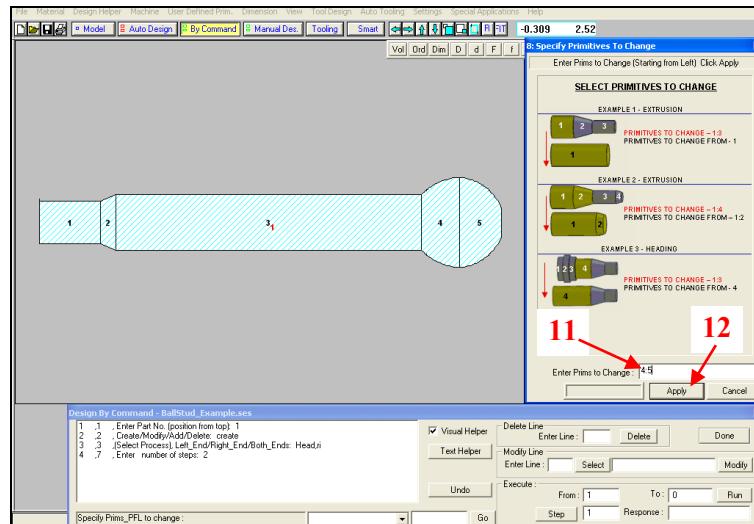
7. Select Type of Operation – In this case “Heading on Right Side”
8. Click on ‘Apply’ to proceed to the next window.



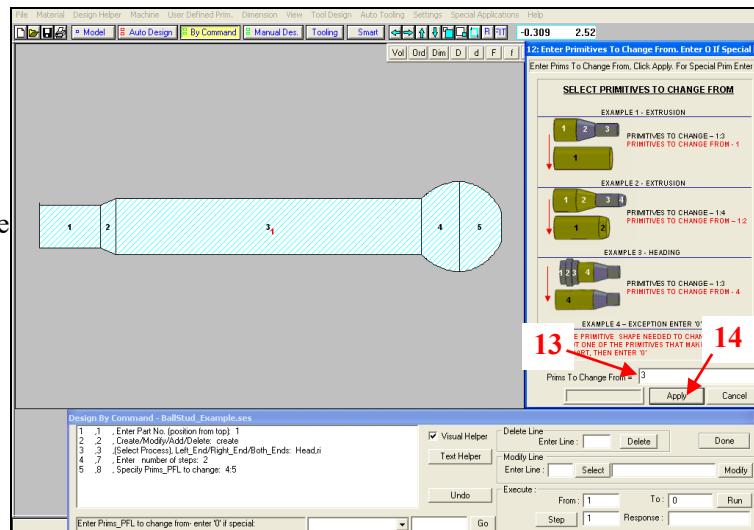
9. Select ‘Number Of Steps For the Heading Process’ in this case ‘2’.
10. Click on ‘Apply’ to proceed to the next window.



11. Enter Primitives to Change. In this case Primitives 4,5 are being headed. Enter '4:5'
12. Click on 'Apply' to proceed to the next window.



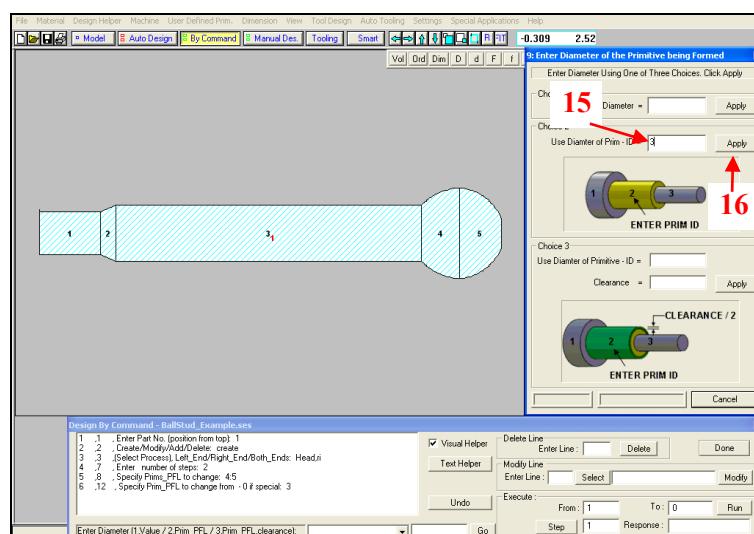
13. Enter Primitive to Change From. In this case Primitives 4,5 are being headed from Primitive number 3. Enter '3'
14. Click on 'Apply' to proceed to the next window.



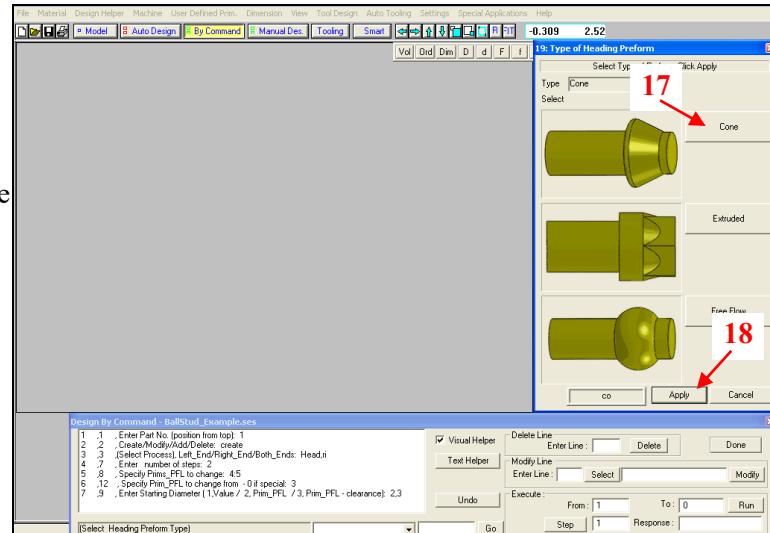
15. Select Diameter Options
 - a. Exact Diameter
 - b. Same Diameter as an existing Primitive.
 - c. Same Diameter as an existing Primitive with an additional clearance.

In this case, Option 2 will be used. Enter '3' for using diameter of Primitive #3.

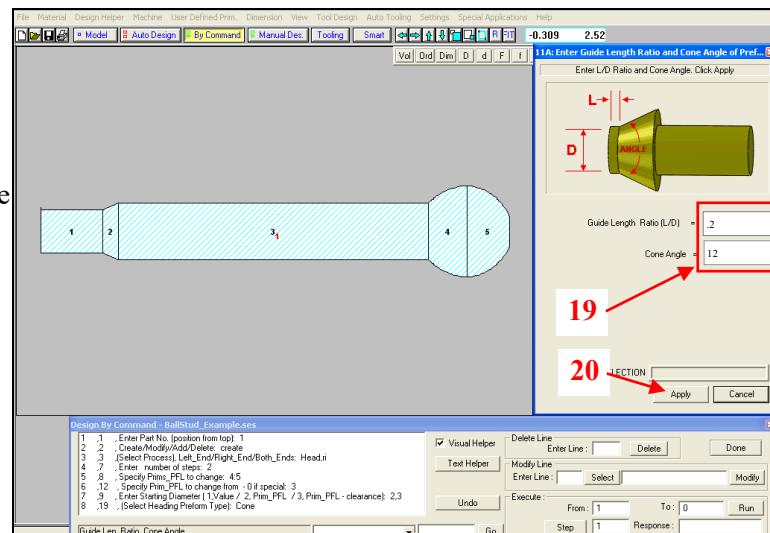
16. Click on 'Apply' to proceed to the next window.



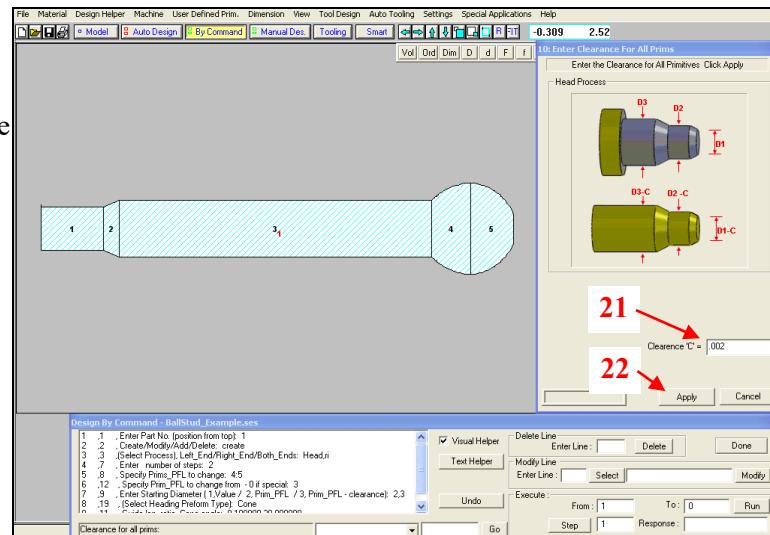
17. Select Preform Type by clicking on the button. In this case, select Cone.
18. Click on ‘Apply’ to proceed to the next window.



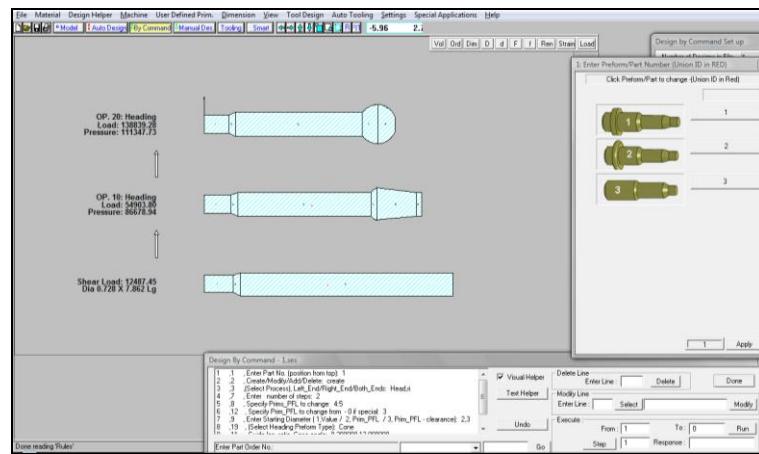
19. Enter details of the ‘Cone’ Preform. Enter L/D Ratio (.2), and Cone Angle (12).
20. Click on ‘Apply’ to proceed to the next window.



21. Enter Clearance Amount (.002)
22. Click on ‘Apply’ to proceed to the next window.

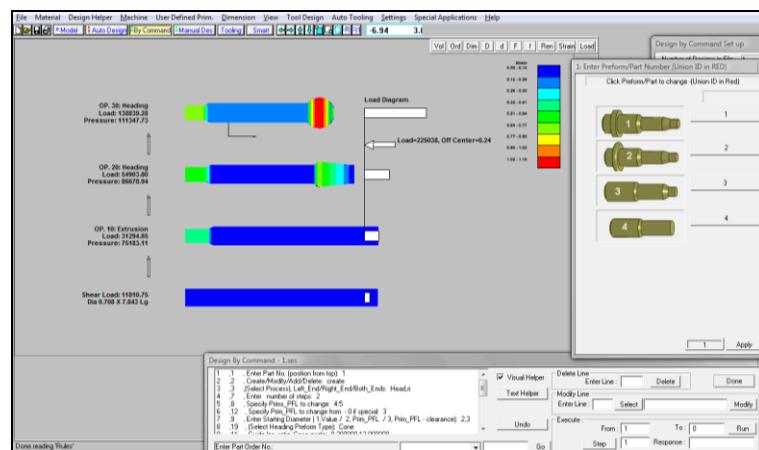


VIEW RESULTS (2- STATION HEADING)



23. Continue in a similar way to create the next station (working backwards in sequence design)

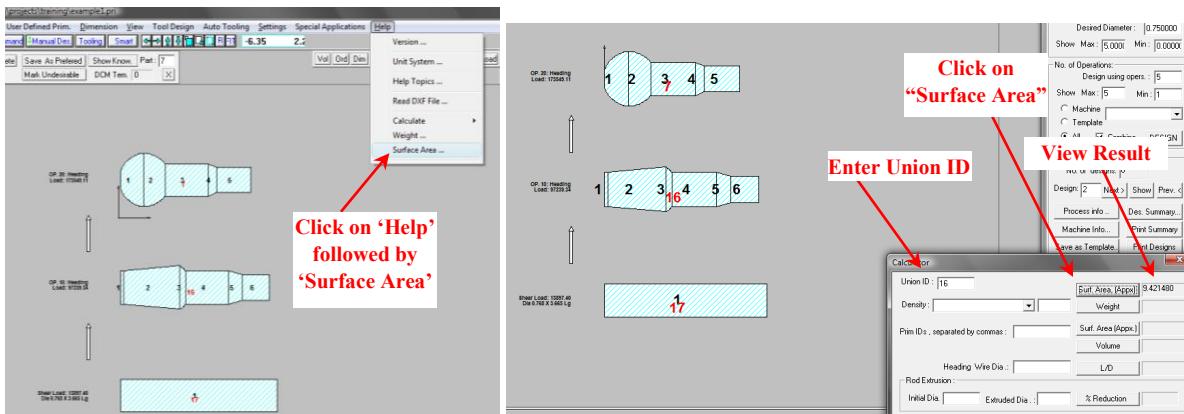
FINAL RESULT - LOAD DISTRIBUTION - STRAIN ON THE PART



12.3.4. Surface Area Calculations

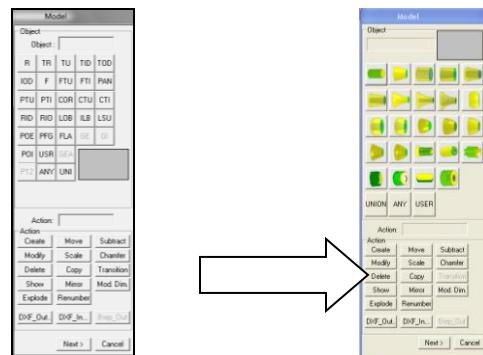
NAGFORM now has the capability to calculate the surface area of any part including the model and preforms. Following are the steps needed in order to calculate the Surface Area:

- Click on 'Help' followed by 'Surface Area'
- Enter Union ID (in this case '16') and click on 'Surface Area (approx)' Button. The program will display the Surface Area on the Right.



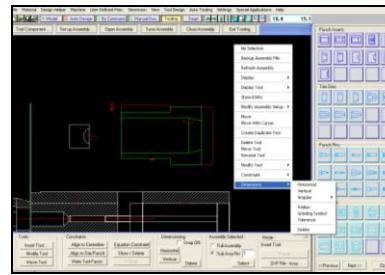
12.3.5. Other Improvements

5A. In the ‘Model’ section of NAGFORM, the buttons representing primitives have ‘Pictures’ in place of ‘Text’ for ease in selecting a primitive type.



5B. In Tooling Assembly of ‘Auto Tooling’ module, dimensioning of ‘Radius’, ‘Angle’ and ‘Grinding Symbol’ has been added

5C. Material database of NAGFORM has been updated to include all materials that



12.3.6. Integration With NAGSIM.2D / 3D

In NAGFORM, a simulation file for NAGSIM.2D/3D is automatically created for any selected part progression generated in Auto Design. For NAGSIM.2D, a single file is created for the whole progression whereas for NAGSIM.3D, one file is created for each selected station of the progression.

For NAGSIM.2D, the ‘Part Progression’ and the ‘Default Tooling’ of NAGFORM program is used to create the geometry of the part and tools in the simulation file. Thus a complete NAGSIM.2D simulation file for up to six stations progression can be created. To simulate, the User has to open this file in NAGSIM.2D, mesh the parts and begin simulating. It takes only couple of minutes to go from NAGFORM’s design concept to NAGSIM.2D simulation.

For NAGSIM.3D, NAGFORM creates the simulation file without the part and tool models. The User can either import the tool drawings from a 3D CAD models or use the NAGFORM-SolidWorks interface, to create 3D models of the default tools in SolidWorks.

