AUTOFORM® 3D GRAPHICS SOFTWARE

AN OPERATION SUPPLEMENT MANUAL FOR THE

CINCINNATI AUTOFORM®
CNC FORMING CENTER

CINCINNATI

CINCINNATI INCORPORATED
CINCINNATI, OHIO 45211

EM-428 (N-11/96)

AUTOFORM 3D GRAPHICS SOFTWARE

INTRODUCTION

This manual supplement provides instruction on using the CINCINNATI 3D Graphics Programming Software. This software is available on the AUTOFORM Machine Control and as an offline package for a PC. Small differences in functionality between the control and offline software will be noted where they apply. A separate manual supplement for the 3D Offline Programming Software explains software installation.

The 3D Graphics Programming software adds the following new features:

- Ability to convert 3D wireframe and 2D flat pattern drawings from CAD systems supporting the DXF file format.
- Ability to design a 3D Part directly on the machine control or offline PC.
- Automatic fold/unfold of flat pattern and wireframe part.
- Bend Simulation and machine interference checking in 3D.
- Automatic creation of bending program.
- Ability to export Wireframe or Flat Pattern as a DXF file.
- Pointing device for selecting geometry and edit fields.
- Internal file storage is now double the capacity (192k)

The main advantage of this software is to eliminate the need to draw part geometry more than once. If a drawing has already been made on a CAD system, it is not necessary to re-enter the drawing again in another format. This software has the capability to read in CAD files using the industry standard DXF file format. Part Geometry in the DXF format may consist of either the flat unfolded blank (with or without bend lines) or a formed wireframe representation.

It is also possible to design a part from scratch using the 3D Graphics software. For simple parts without side flanges, the Part Design 2D Menu may be used. For more advanced parts, the Part Design 3D Menu can be used.

Once a part has been completed either though a DXF conversion or the Part Design method, the tooling can be selected. The 3D software will then find all of the possible bending sequences which do not have interference. In the PART SIMULATION screen, the user can view these sequences in 3D and select the most efficient one.

Once the sequence is chosen, a bending program will automatically be written, completing all the necessary ram and gage positioning calculations. When the operator runs the part they will see a 3D view of each step of the forming process.

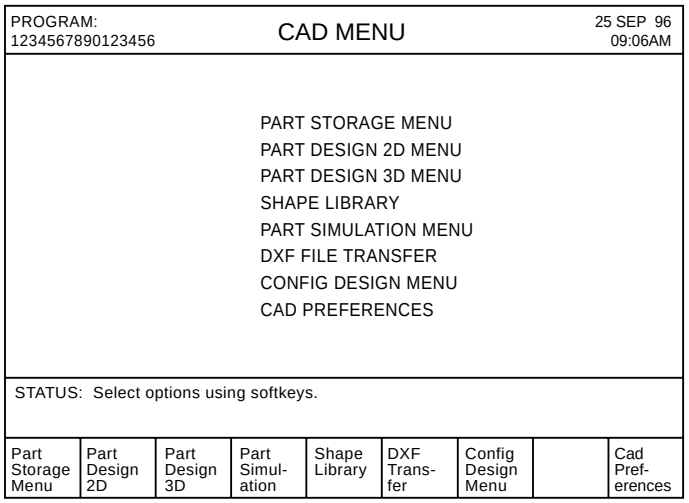
The organization of this manual is to first guide the user through the process of importing a DXF file and realizing a bend program ready to be run. The alternate method of designing a part from scratch using the Part Design 2D or 3D will follow.

POINTING DEVICE (MOUSE)

The new pressure sensitive pointing device located on the control allows the user to pick the field on a screen to edit by moving the on screen pointer arrow to the desired field and then pressing the left button on the pointing device. The pointer is moved up, down, left or right by applying pressure to the round button in the intended direction. Softkey functions may also be selected by positioning the pointing arrow anywhere in the softkey area and then pressing the left button. Pressing the button anywhere in the red Menu Title area is equivalent to pressing the Main Menu key on the front panel. Menu screens such as MAIN MENU or the CAD MENU allow selection of the item by picking the title of the menu using the pointer and button. This device will be referred to as the “Mouse”.

CAD MENU SELECTIONS

The CAD MENU screen will allow the user access to all functions corresponding to the design, simulation, storage and transfer of part drawings.



PART STORAGE MENU: Allow access to storage screen used for storing parts with the internal 3D format or existing 2D format to and from disk or internal memory.

PART DESIGN 2D MENU: Allows design of part cross section using 2D design techniques.

PART DESIGN 3D MENU: Allows design of 3D formed part model, or editing of formed model imported from DXF File Transfer.

SHAPE LIBRARY: Storage for frequently used common 2D part cross sections. Consult your AUTOFORM Operation and Maintenance Manual, EM-408, for more information.

PART SIMULATION MENU: Allows specification of simulation parameters, generation of all possible bend sequences, viewing and selection of bend sequence and automatic bend program creation.

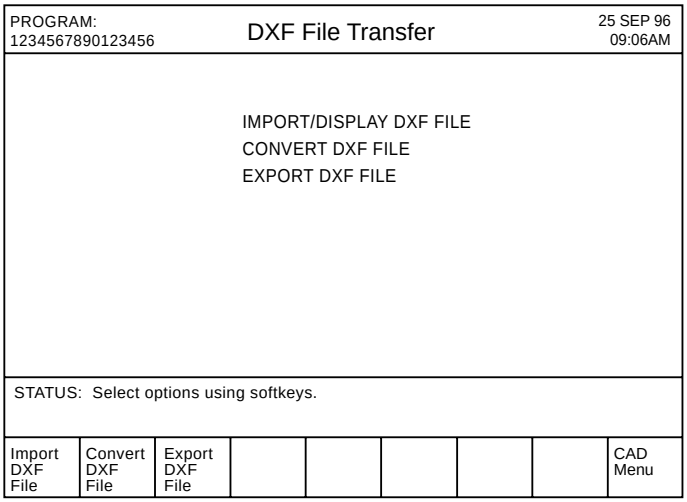
DXF FILE TRANSFER: Allows access to import/export CAD drawings using the DXF drawing format.

CONFIG DESIGN MENU: Allows specification of machine dimensions, options and material springback values.

CAD PREFERENCES: User defined preferences for screen colors, mouse selection, etc.

DXF FILE TRANSFER

The DXF FILE TRANSFER MENU screen allows the user access to all functions corresponding to the importing or exporting of a drawing and converting a drawing into the internal database of the control.



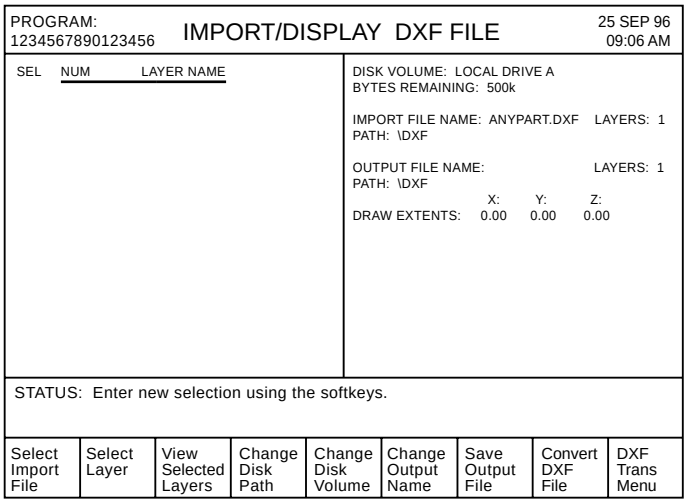
IMPORT/DISPLAY DXF FILE: Prompts user to specify the file to import, displays the file and prompts for required drawing layers to extract. Selected layers from file can be saved to another file specified by the user or user may transfer directly to Convert DXF File.

CONVERT DXF FILE: Converts drawing geometry into the internal 3D representation based on type of drawing selected.

EXPORT DXF FILE: Outputs a drawing from the internal representation into a standard DXF file for use with an outside CAD system or post-processing system.

IMPORT/DISPLAY DXF FILE

This function allows a user to import CAD geometry from other CAD applications using the DXF file format.



DRAWING LAYERS

A DXF file may have geometry separated into individual groups or layers. Typically layers are used to group text or dimension lines together. If the drawing contains multiple layers they will be listed on the screen. It is possible to un-select or turn off individual layers so the geometry will not appear when viewing the file.

DRAWING EXTENTS

The extents are the maximum X , Y and Z dimensions of the geometry in the DXF file. The dimensions of the part are in the units that it was drawn under. Drawings containing metric units should be imported with the Machine Configuration Unit selection set to metric. For a flat pattern drawing, the Z-dimension will be zero.

METRIC / ENGLISH UNITS

DXF files may be imported which have been drawn in either English or Metric units. For English, all linear units are assumed to be inches, while metric assumes millimeters. To import a metric drawing, set the Units

in Machine Configuration to metric, otherwise the drawing will be converted as inches. The Drawing Extents field on the screen can be used to determine if a drawing is in metric or English.

SOFTKEY FUNCTIONS

Select Import File: User selects the filename to be imported from the list of files shown. User may select file by sequence number, name or using the mouse.

Select Layers: Allows user to cursor select the desired layers to be included in the display view from those listed. Options include select all layers, select page and select single. Layers without titles will be shown as blank.

Select All	Select Page	Select Single						Exit

When pressed, the "Select Single" softkey activates either the cursor up and down keys or the mouse to allow individual layers to be un-selected.

View Selected Layers: Erases the screen and displays all the selected layers of the drawing on the screen. Displays drawing full screen at maximum scale possible.

Change Disk Path: Allows user to select which directory on the disk to obtain the imported file. Displays list of directories, allows user to access various levels of disk directory to determine path.

Change Disk Volume: Will only appear if there is more than one disk volume available. Allows user to choose where imported file is located.

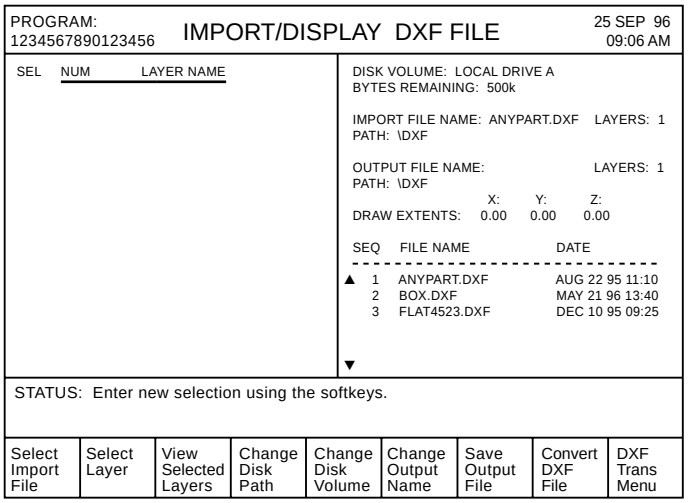
Change Output Name: User may save the imported file with the selected layers back to disk as another file. User enters desired name of file to be stored under.

Save Output File: Writes the output file to disk under Output Name.

Convert DXF File: Converts drawing into formed part model.

LOADING A DXF FILE

After pressing the "Select Import File" softkey, the screen will change as shown. The listing of files for the specified disk volume and path will be shown in the lower right area of the screen. Additional files may be viewed by scrolling using the up and down field keys or clicking on the up and down arrows with the mouse. The default path for DXF files is \DXF. This can be changed in the FILE PREFERENCES MENU. It is also possible to change the directory path to look for files by using the "Change Disk Path" softkey.



On the AUTOFORM Control, DXF files can only be loaded and stored directly to and from a floppy disk. They cannot be stored to the internal memory of the control, transferred using the communications or copied using the FILE TRANSFER screen. DXF filenames can only be up to 11 characters. They follow the same rules as normal DOS filenames using the 8 plus 3 character format.

Once a file has been selected, it will be loaded from disk. If it is a properly formatted DXF file, the screen will update to show a list of the layer names and the drawing extents. Note that some geometric entities in the file may not be supported. For a list of supported entities, please see *APPENDIX A*. Note that since the size of a DXF file can be large, it will not be possible to read some DXF files into memory. An error message **"Out of Memory, Complete Drawing cannot be loaded"** will be displayed. If this occurs, examine the original CAD file to see if it contains any non-part geometry or text which can be removed.

All layers will default to selected. Use the "Select Layer" softkey to turn off any unwanted layers.

RE-SAVING A DXF FILE

If desired, it is possible to save out the DXF file under a different name. Any layers or entities which have been un-selected will not be saved. This function is useful, for example, in splitting up a drawing which may contain multiple parts. To save a DXF file, first use the "Change Output Name" softkey to enter the new name for the file. Next press the "Save Output File" softkey to store the part. Note that all geometry in the part is stored in one layer when a part is re-saved.

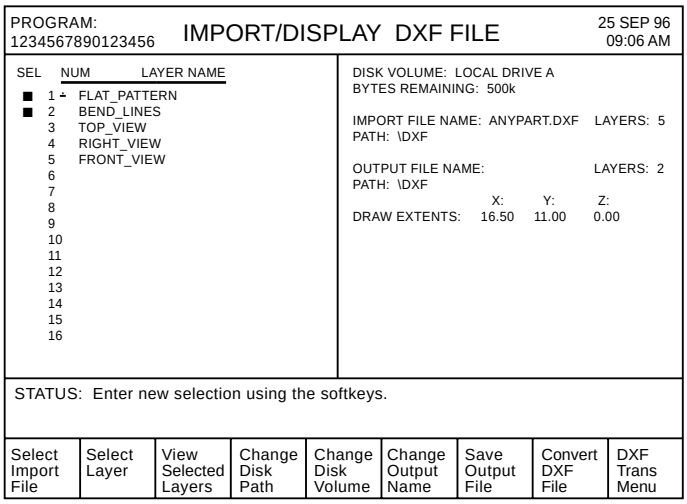
FLAT PATTERN GEOMETRY

The minimum requirements for flat pattern geometry is an outline of the un-bent flat blank. The drawing

must be true scale. It may contain internal features such as holes, slots, etc., which will be translated and remain with the part. It may contain predefined bend lines. These bend lines must be located at the "neutral" location of the bend, i.e. the flat blank should already include any bend allowance. The part drawing may contain text to help identify the bend line location and angle. Once bend lines have been defined or picked and the part converted into a formed model, all text will be discarded. If the DXF file contains any other views of the part, such as the isometric view or the different projected views, these must be removed before converting the part.

WIREFRAME GEOMETRY

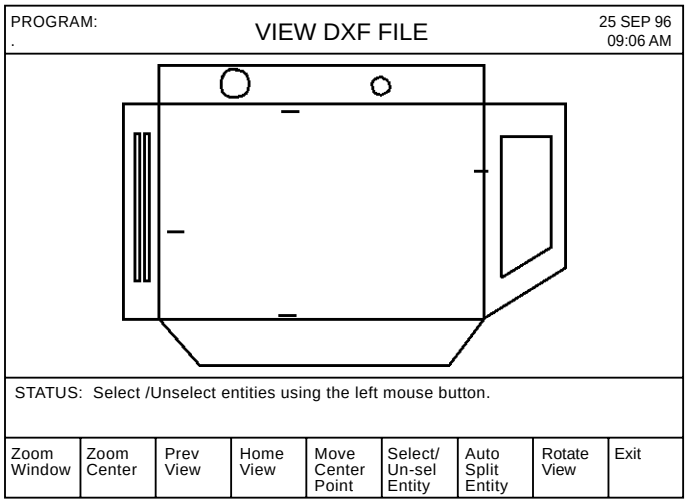
The minimum requirements for importing a 3D wireframe part is a formed outline of the part including bend lines. The wireframe is defined as the outside surface of the part. It cannot have thickness. Bends should not have explicit radius geometry defined, i.e no arcs. All dimensions for flanges will be assumed to have outside dimensioning. Like the flat pattern, flange geometry may contain internal features. Text is displayed but discarded once the part is translated. The part cannot have internal bends - these will be discarded.



For this example, the file ANYPART.DXF has been selected and loaded. It contains 5 layers. This particular drawing is a flat pattern. It also has the bend lines included on a separate layer. The drawing includes additional geometry for the different views of the formed part. These must be turned off.

VIEWING THE DXF FILE

The "View Selected Layers" softkey will display the DXF file. Any layers which have been un-selected will not be included when the file is displayed.



UN-SELECTING GEOMETRY

At this point it may still be necessary to remove individual geometric entities from the drawing which do not correspond to the actual part geometry. This can be done by using the mouse to click on the entity to be removed. The entity will then change color (default is from green to brown) indicating it is un-selected. Clicking on the same entity again will toggle it back to the selected state. There are a number of other functions provided to aid in removing entities and viewing the drawing.

SOFTKEY FUNCTIONS

Zoom Window: This softkey allows the user to use the mouse to define a rectangular window on the part to zoom up. Use the mouse to select the upper left corner of the window by pressing on the left mouse button. Move the mouse to define the lower right corner of the window and press the left mouse button.

Zoom Center: This softkey will zoom the drawing by a fixed amount about the center of the drawing area. Each time the softkey is pressed the drawing will be enlarged.

Prev View: This softkey will switch between the last view selected and the current view.

Home View: This softkey will change the view such that the entire part is displayed in the viewing window.

Move Center Point: This softkey will prompt the user to use the mouse to select a point on the drawing to use as the center view point. This function can be used to pan in any direction without changing the scale while viewing.

Select/Un-Sel Entity: Allows several choices in picking entities as shown:

Select All	Un-Sel All		Select Window	Un-Sel Window				Abort

Select/Un-Sel All: This softkey allows the user to select all entities of the drawing or un-select all entities.

Select/Un-Sel Window: This function is similar to the previous, however, the user first defines a window using the mouse. All entities contained within the window are either selected or un-selected. To pick a window, move the mouse to the upper left corner of the rectangular window to be drawn and press the mouse button. Move the mouse to define the lower right corner of the window and press the left mouse button.

Auto Split Entity: Some parts, in particular 3D wireframe drawings, require lines to be split before the part can be converted properly. Typically this is needed when a bend line used by several flanges is drawn as one continuous line. If you find the part has problems converting, use this function.

Rotate View: This allows the user to rotate the viewport in fixed increments of degrees. Increment is defined in Cad Preferences. Softkeys will appear to allow the rotation selection as shown.

Rotate CCW	Rotate CW	Rotate Up	Rotate Down					Done

Once the drawing appears to be correct the next step is to convert it into a formed 3D model. Exit out of the View DXF screen and Press the Convert DXF softkey.

CONVERT DXF FILE

The screen will be cleared and the drawing previously selected in the DXF IMPORT DRAWING screen will be displayed. The screen title will appear as either Convert 2D Flat Pattern or Convert 3D Wireframe.

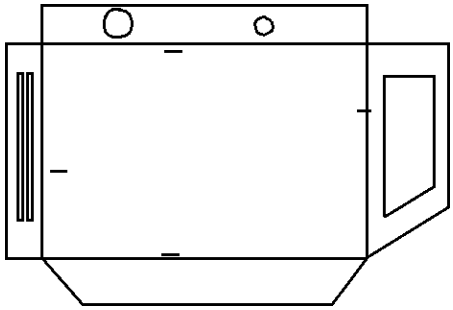
CONVERT 2D FLAT PATTERN

The flat may have bend lines already drawn or the user can then specify the location of bend lines and the bend angle for each flange. The software will draw the 3D formed part from this information.

PROGRAM:
.

CONVERT 2D FLAT PATTERN

25 SEP 96
09:21 AM



STATUS: Select a function using the softkeys.

Add Bend Line(s)	Delete Bend Line	Convert To 3D Part	Zoom View	Display Flat Pattern	Display Formed View	Measure Linear Distance	3d Part Design	DXF Menu

PICKING EXISTING BEND LINES

Add Bend Line(s): This softkey is used to create new bend lines on the flat part. If the part already has bend lines, then it is only necessary to pick each line and enter the bend angle. The status line will change to:

STATUS: Select a segment to use as a bend line.

The softkeys will change to show only an Add Bend Line option. At this point the user may select any existing line segment on the flat.

		Add Bend Line						Done

Use the mouse to pick a bend line on the part by pointing to it and clicking the left button. The line will now change color to indicate it has been picked.

The status line will change to prompt for the bend angle of the line.

STATUS: Enter the Bend Angle 90.00
MAX: 180.00 MIN: -180.00

Note that all angles are included angles from 0 to 180. Angles with a positive sign will be formed in the UP direction, angles with a negative sign will be formed in the DOWN direction (into the screen).

The user can continue to define more bend lines by picking the next bend line with the mouse. When all bend lines have been picked, press the “Done” softkey.

ADDING NEW BEND LINES

If the “Add Bend Line” softkey is pressed, the status line will change as shown.

STATUS: Select a segment to use as a reference.

To define a new bend line location the user first selects any line segment or previously defined bend

line of the flat part. This is done by picking the line with the mouse. The line segment is then redrawn as a dashed line and can be dragged in a parallel direction to the original line segment. As the mouse is moved, the line will move. A value will now appear on the screen labeled BEND OFFSET. As the mouse is moved, the offset value will show the distance from the original line segment the line is moved.

The Status line will change as shown.

STATUS: Drag reference segment to desired bend location.

The softkeys will change as shown.

Enter Offset Dim	Enter Offset Angle		Accept Bend Line					Abort

Pressing the left mouse button will fix the line at its current location. A bend line will be created at this location extending on either side until it intersects another line.

Enter Offset Dim: It is also possible to enter a fixed offset value directly from the keyboard by first pressing the "Enter Offset Dim" softkey.

Enter Offset Angle: This softkey will change the angle of the bend line by the entered amount. It will rotate the line about its center point. Positive angles rotate counterclockwise. The line can continue to be dragged or another offset dimension can be entered.

Accept Bend Line: Accepts the new bend line at its current location and prompts for a bend angle. Once it has been accepted, the only method to change it is to first delete an existing bend line and re-enter it.

Continue adding or picking bend lines until all bends have been specified.

Delete Bend Line: This softkey may be used to delete a previously entered bend line. Press the softkey and then identify the bend to be removed using the mouse.

Zoom View: This softkey allows the user to examine a portion of the drawing in more detail. It functions identically to the Zoom functions in the VIEW DXF MENU.

Measure Linear Distance: This softkey when pressed will prompt the user to select two points on the part using the mouse. The first point is the point to be measured from and will be indicated with an X when the left mouse button is pressed. Next move the mouse to the point to be measured to. If the left mouse button is pressed, the selected point is that which is closest to the mouse pointer. If the right mouse button is pressed, the selected point is the closest endpoint of a line to the mouse pointer. After selecting the points the linear distance and the actual X, Y, Z coordinates will be displayed on the second line of the status area.

Example:

STATUS: Select a Point to measure to:

Distance: 9.779 X: 1.047 Y: 9.125 Z: 3.358

More points to measure to can be selected by moving the mouse. To redefine the point being measured from, press the "Redo Base Point" softkey. Press the "Abort" softkey to stop measuring distances.

						Redo Base Point		Abort

CONVERTING THE PART

Convert To 3D Part: This softkey will translate the flat pattern into a 3D part model. First it will prompt to confirm that the existing 3D part in the PART DESIGN 3D MENU will be deleted.

STATUS: Current 3D Part will be DELETED! Continue with Conversion?

Next the material thickness of the part is requested. This will also be used as the default bend radius for all bends. The bend radius may be changed in the PART DESIGN 3D MENU if needed.

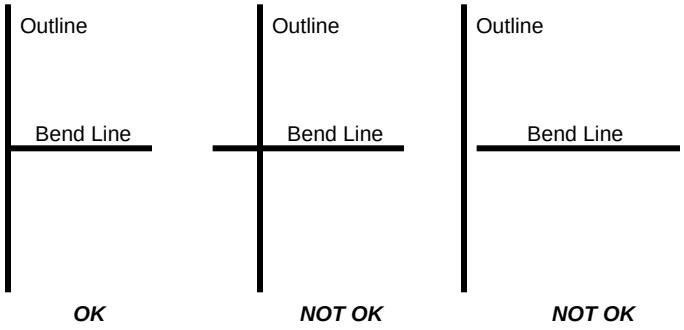
After entering the thickness the part is translated. If no error messages appear it has been successful. The bend information and geometry will be transferred into the internal database format for the active part which can be utilized further in the PART DESIGN 3D MENU.

If you get the error "No bend lines were specified on Part", it will be necessary to add bend line(s).

If there are error messages, the part geometry may be incomplete or bend lines may not have been defined. If the flange geometry on the part is missing lines, the drawing will show in a light blue color which line segment it had trouble identifying as part of a flange.

If Bend Lines are included in the drawing, they must follow these guidelines:

1. Each bend line must be one continuous line.
2. Bend lines cannot intersect.
3. Bend lines must begin and end exactly on the part outline.



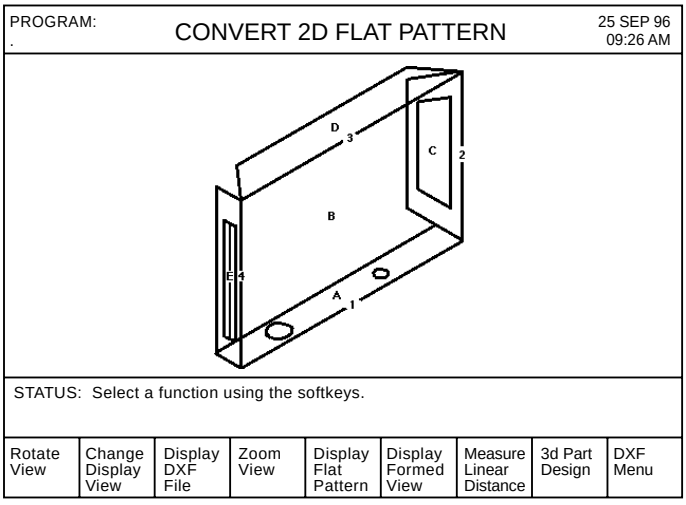
Here are some common problems to look for if a part does not convert:

- ✧ Bend lines extend outside the part outline
- ✧ Bend lines intersect each other.
- ✧ Lines that appear to connect are not really connected when zooming in real tight.
- ✧ Bend lines intersect internal features. (Use the Auto Split Entity function, and define bend line on both sides of the feature.)
- ✧ Part contains too many features. (Eliminate some until it converts.)

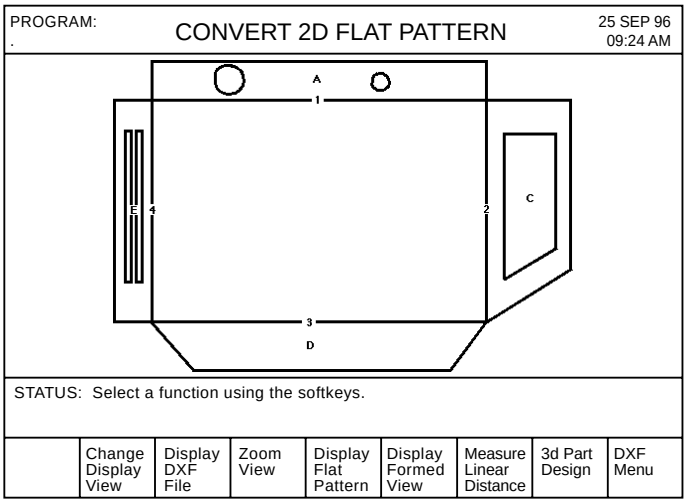
VIEWING THE FORMED OR FLAT MODEL

The two softkeys below may now be used to view either the flat or formed part. Bends will be labeled using numbers in the order in which they were picked. Flanges are labeled using letters.

Display Formed View: This softkey will erase the part from the screen and show it in its current state as a formed part (folded part). All bend lines defined up to this point will be shown as bends in the 3D Formed View.



Display Flat View: When pressed, this softkey will show the part in its unfolded state.

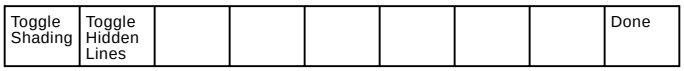


The softkeys will now change to allow more options on viewing the formed or flat model.

Display DXF File: This softkey will re-display the original DXF file so it may be compared to the translated part. If needed, bend lines can be changed and the part may be converted again.

VIEWING OPTIONS

There are several options on viewing the flat or formed model. Pressing the “Change Display View” softkey will bring up the following options.



Toggle Shading: This option will redraw the part using different shades of color to give the part the look of a real part. Shading colors are user selectable in the CAD PREFERENCES screen.

Toggle Hidden Lines: This option does not shade the part but instead removes lines which should be hidden from the users viewpoint.

It is best to rotate the part to the desired viewpoint before using these options due to the extra time required to calculate a shaded model.

Rotate View, Zoom View and Measure Linear Distance have the same options as before.

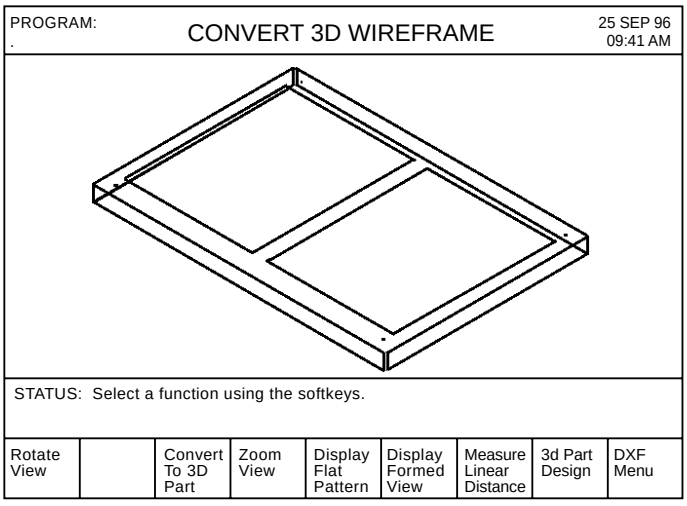
CONVERT 3D WIREFRAME

3D Part Design: This softkey will translate the 3D Wireframe part into a 3D part model. First it will prompt to confirm that the existing 3D part in the PART DESIGN 3D MENU will be deleted.

STATUS: Current 3D Part will be DELETED! Continue with Conversion?

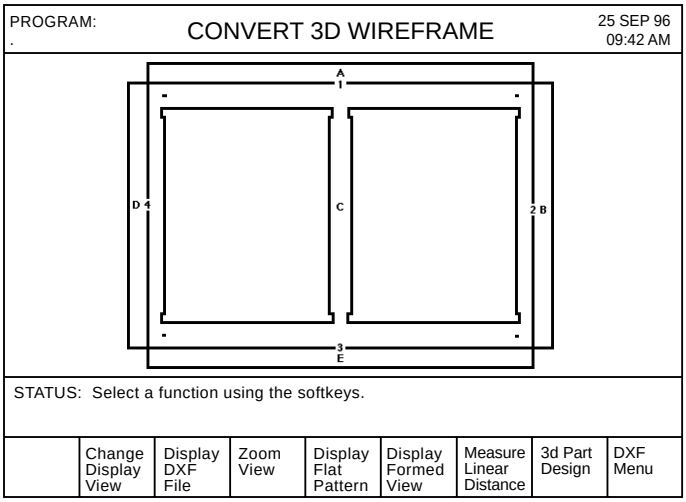
Unlike a Flat Pattern drawing, the 3D Wireframe already has the bend information within the geometry of the part. The only additional information required before converting it is the material thickness. This will also be used as the default bend radius for all bends. The bend radius may be changed in the PART DESIGN 3D MENU if needed.

After entering the thickness, the part is translated. It has been successful if no error messages appear. The bend information and geometry will be transferred into the internal database format for the active part which can be utilized further in the PART DESIGN 3D MENU.



If the part does not convert properly, check to see if there are missing segments on the flanges. To be recognized as a flange, there must be a continuous path from the start of the bend line to the end. Another common reason a part will not convert is that the part has thickness.

Viewing formed or flat patterns after converting works the same as described under the previous section.



Zoom View: This softkey will allow the user to change the viewing parameters in the same manner as the IMPORT/VIEW DXF screen.

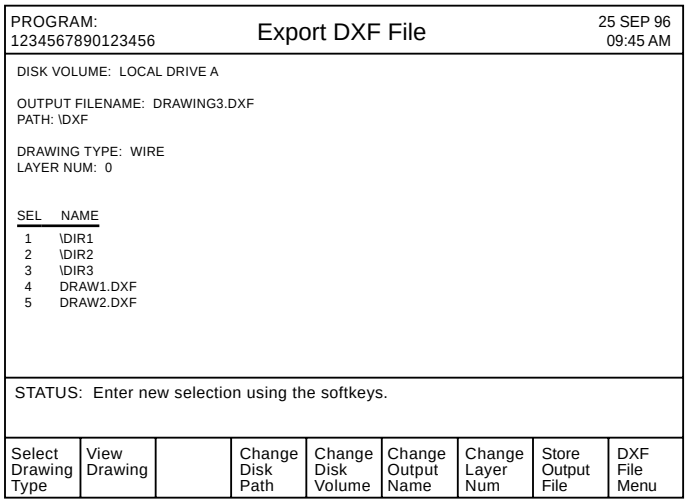
Display Flat Pattern: This will display the unfolded part. It is necessary to first convert the part before this function will work.

Display Formed Part: This will display the folded or formed part. Again, it is necessary to convert the part first.

3D Part Design: Pressing this softkey will switch to the PART DESIGN 3D MENU.

At this point the 3D part is ready to be simulated in the Part Simulation menu to test for all possible bending sequences. If the part requires some editing of flange dimensions or bend angles this may be done using the PART DESIGN 3D MENU.

EXPORT DXF FILE



Once a DXF file has been imported and gone though the conversion as just described, it may be exported back out as another DXF file. This feature may be used to import a 3D Wireframe formed part, have the software unfold it and then exported back out as a DXF file. The flat blank may then be utilized in some other software applications such as a nesting package or NC program generation for a laser or turret.

Select Drawing Type: This softkey will prompt for which type of drawing to be exported. Either the flat pattern or wireframe.

View Drawing: Displays the part in either the wire or flat to verify correctness before saving file.

Change Disk Path: Allows specifying which directory path the file will be saved under on the current disk volume.

Change Disk Volume: Specify on which disk volume file will be saved.

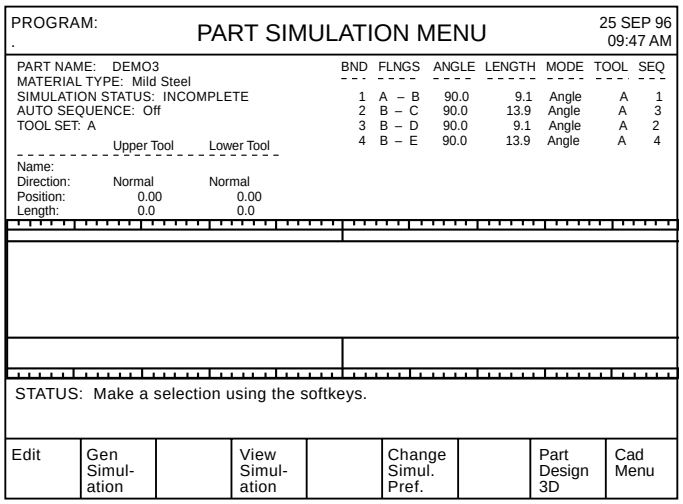
Change Output Name: Allows entry of user defined name to store the DXF file as.

Change Layer Num: User specified drawing layer in which all entities will be stored as.

Store Output File: This softkey will write the DXF file to the disk volume.

PART SIMULATION MENU

The PART SIMULATION MENU is shown below with the bend information completed from the example part.



The information shown on this screen has been transferred from the 3D part which was just imported and converted.

PART NAME: Filename which the part and the simulation are stored under. Defaults to the same name as the DXF file but may be changed in the PART DESIGN 3D screen. Maximum of 16 characters is allowed.

MATERIAL TYPE: Defaults to Mild Steel but is changeable in PART DESIGN 3D screen. Material Type is used to allow for the angle springback during bending. Springback values can be adjusted in the CAD CONFIGURATION MENU.

SIMULATION STATUS: This field provides an indication on the status of the current simulation. INCOMPLETE means that additional information must be entered before generating a simulation (typically the tool names). INVALID means that Generate Simulation has been attempted, however, no valid bending sequences were found. It can also mean that information on this screen has been changed since generating a valid simulation. VALID means that one or more bend sequences have been calculated without machine interference.

AUTO SEQUENCE: Default is OFF. In the OFF position, the Generate Simulation will only check the user specified bend sequence for interference. In the ON position, Generate Simulation will check for all possible combinations that the part can be made to determine valid bend sequences.

TOOL SET: This field indicates which tool set the Upper and Lower Tool information just below is referring to. Up to 26 die sets may be used for the same part. The letters A-Z are used to identify tool sets. The user must select at least one set of tools before attempting to generate possible bend sequences.

Name: This field allows a 16 character Upper and Lower tool name to be specified. The name may be entered by selecting the sequence number of the desired tool in the Tooling Library or by typing the complete name.

Direction: The orientation of the tool whether it is Normal as it appears in the Tool Library or Reversed.

Position: This allows the tool to be placed anywhere along the die area of the machine. The position entered should be the location of the center of the tool. The default position of 0.0 centers the tool in the middle of the machine. Looking at the front of the machine, the center is 0.0, Left of center is a negative position, Right of Center is a positive position value.

Length: This field defaults to the length of the tool as it is defined in the Tool Library. The length can be changed here to override the tool library value. This is normally done when sectionalized tooling is used.

The following softkeys will appear when entering the tool names.

Enter Tool Name	Clear Tool Name		Prev Tool Set	Next Tool Set		Tool Library		End Edit

"Next Tool Set" and "Previous Tool Set" will change the display to allow each of the 26 tools to be entered.

"Clear Tool Name" erases the tool names for the current tool set.

"Tool Library" conveniently switches screens directly to the Tool Library to allow selection of the desired tool. To return directly to the PART SIMULATION screen, press the "Last Menu" softkey.

The tool sets are graphically shown in the middle of the screen. The grey area represents the ram and the

filler block of the machine. It is true to scale, thus the open area is the maximum die space available for tools. The line in the center represents the center of the machine. The ruler on the top and bottom is dimensioned to also represent the entire length of the die area. The major dimension marks represent 1 foot increments and the smaller dimension marks represent 2 inch increments.

After defining the tool sets and pressing the "End Edit" softkey, all tools will be shown in the die area.

Tools appearing in green are positioned in the Normal direction, while those drawn in light blue have been Reversed.

MISMATCHED DIE DETECTION

If any tools have been selected which overlap each other, they will be highlighted in Red. An error message **"There is a conflict between one or more die locations"** will also appear. Adjust the position of the tools until they all fit within the die space.

If a tool set is drawn in orange, then an error message will also appear as follows **"Lowest Rev Pos cannot be achieved on all Tool Sets"**. This is an indication that the dies have been mismatched for the given part. It is not physically possible to bend all angles of the part without one dieset interfering with another.

BEND PARAMETERS

BND FLNGS ANGLE LENGTH MODE TOOL SEQ

The right side of the screen lists all of the bends for this part. Under the FLNGS heading, the two flanges which are formed by each bend are listed. The Bend ANGLE and Bend LENGTH are also listed. The Bend LENGTH is useful when selecting the correct tools for the part. None of these values can be changed in this screen. Use the PART DESIGN 3D MENU to make changes to the part.

Bend MODE can be changed. This selection is the actual ram reversal mode to be used when the bending program is created. The default is Angle Mode. The other choices available are Position, Tonnage, Absolute Position and the optional Adaptive reversal.

TOOL: This selection identified which bend uses which tool set. Enter the letter of the tool to be used (A-Z) for each bend. The default is to use Tool Set 'A' for all bends. If multiple tool sets are used, the part will move along the die to the location of the tool set.

Only 10 bends are listed at once. To scroll up or down to see additional bends, use the up and down field keys or use the mouse to click just above or below the first and last bend displayed.

BEND SEQUENCE OPTIONS

Bend SEQ can also be changed. This is the sequence to use for bending the part. It defaults to the same sequence that the bends were defined when the part was converted. They can be changed by entering the number from the keypad. There are several other options for the Bend Sequence.

The softkeys will change as follows:

Seq Not Specified	All Not Specified							End Edit

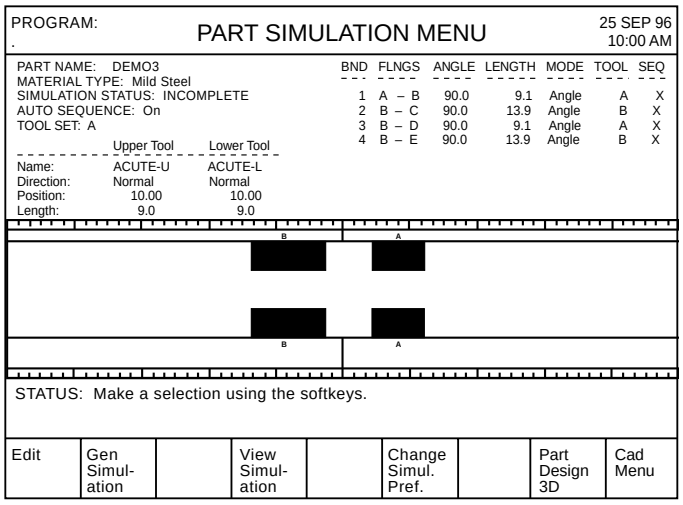
If only a fixed sequence is to be checked, then enter a number in the Bend SEQ field for every bend. To have the machine automatically check for all possible sequences, press the "All Not Specified" softkey. This will change the Bend SEQ number to an 'X' for all bends. Any bend sequence with an 'X' will be used as a wildcard and the machine will check for all combinations possible. It is also possible to pre-determine a partial bend sequence and leave others to be calculated. Use the "Seq Not Specified" softkey to set individual bends to an 'X'.

Note that the AUTO SEQUENCE field will automatically change to ON whenever an 'X' is selected for any bend.

IN-LINE BENDS

On some parts, several bends may fall in line. In other words, more than one flange can be bent in one ram stroke. These would normally be treated as individual bends. To combine in line bends into one step, set the Bend SEQ number for all of the bends to the same value. When a part is simulated, there will only be one bend step for each group of inline bends.

To edit any values on this screen, use the mouse to select the field or press the "Edit" softkey and use the field keys to select the desired field to change.
The following screen shows the PART SIMULATION screen after tooling has been specified and the bend sequence has been set for All Not Specified.



GENERATING BEND SEQUENCES

Gen Simulation: When pressed, this softkey will either verify the user's bend sequence if AUTO SEQUENCES is Off or attempt to generate up to 10 valid bend sequences automatically if AUTO SEQUENCE is On.

If AUTO SEQUENCE is On, the user is presented with a softkey prompting them if they would like to interactively generate bend sequences or have the software generate them without interaction.

Auto Seq Gen	Inter-Active Gen							Abort

Before attempting to generate a bend simulation, the software will verify that all tool sets specified for the flanges are valid. It will also check for incorrect values of material thickness, part radius, vee opening etc., and signal the appropriate error message.

The software will begin generating bend sequences. The status line will appear as shown below to show its progress. The "Stop" softkey, if pressed, will stop the process, keeping all sequences found up to the point of interruption.

STATUS: Generating Possible Bend Sequences. . .
Found X Searching

								Stop

The graphical gage will indicate the progress or percentage complete in searching for valid sequences. Note that if the user aborts the search before it is complete, they may continue the search at the point where it left off.

Once the user accepts a bend sequence for the purpose of creating a bending program, all other sequences found are cleared.

Re-Start Gen	Cont-inue Gen							Abort

Restart Gen: When pressed, this softkey will start over from the beginning looking for valid bend sequences. All sequences found up to this point are deleted.

Continue Gen: This softkey, when pressed, will continue to check for additional bend sequences from the point where the process was stopped. If 10 valid simulations have been found, the software will attempt to find up to 10 more sequences. If no other valid simulations exist, the software will keep the existing sequences. If it does find at least one additional sequence, the original 10 bend sequences will be erased. The status line will change as before while sequences are being generated.

Interactive Generation: This allows the user to step through the bend sequence generation, deciding interactively whether to accept a sequence or reject it. The software will begin the process the same as before. The user will be prompted whether or not this first bend is desirable. If it is, the user will be presented with the second bend. The user again has the choice of accepting or rejecting the bend. If a bend is rejected, the control will now begin checking other bend sequences, keeping those bends which have been accepted so far. If no valid sequences are found, the user will be prompted to continue starting one bend back or to accept the last valid sequence. A typical session might proceed as follows.

Accept Bend	Reject Bend							Abort

Calculating Bend Sequence...

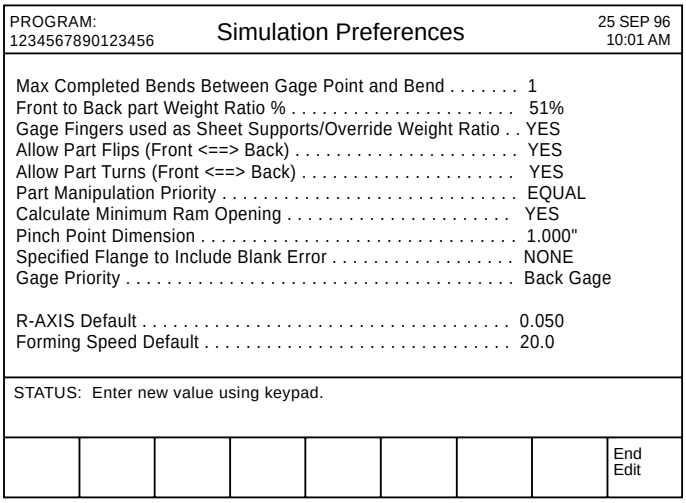
Bend 1 Accepted	3
Bend 2 Accepted	3 1
Bend 3 Rejected	3 1 2
Bend 3 Rejected	3 1 4
Bend 4 Accepted	3 1 5
Bend 4 Accepted	3 1 5 2
Bend 5 Accepted	3 1 5 2 4

Valid Sequence Found

This interactive sequence generation allows the user to throw out a large number of sequences by first previewing bends. Note that it is possible to revert all the way back to the first bend by continuing to make rejections. As the user rejects bends earlier in the forming sequence, more potential sequences are rejected.

SIMULATION PREFERENCES

The “Change Simulation Pref” softkey will display the following screen.



This screen is used to modify the various rules used in determining a valid bend sequence. This allows the user to optimize the bend sequence to their requirements. The softkeys will display the various choices for each field. The values stored in the screen get saved and recalled with a part. Some parts may require editing of these values before a valid simulation can be generated. It is always possible to override these preferences by manually sequencing and selecting the gage positions for the part.

Max Completed Bends Between Gage Point and Bend: This field controls how may pre-bent flanges are allowed between the current bend and the gage point. The default value of one provides the most accurate situation due to the error which builds-up after multiple bends. The user may also select the value of UNLIMITED in which case this check is not performed.

Front To Back Ratio part Weight Ratio %: This field controls the valid part length ratio allowed between the section of the part in front of the ram and behind. (FRONT/BACK = 99%, part is totally in front of ram). It is unsafe to have an unsupported sheet (i.e., length ratio < 50%) which could fall behind the machine. The default value for this field is 51% such that the part will be formed in a sequence which does not cause the part to fall. The range of values is 1-99.

Gage Fingers used as Sheet Supports/Override Weight Ratio: This feature uses stepped gage fingers as sheet supports to support the sheet in back of the ram. When this is selected and the user has chosen a stepped finger, the front-to-back weight ratio is ignored and the gage fingers are positioned such that the material rests on top. This only applies to a CNC backgage – not a front gage.

Allow Part Flips or Turns (Front <==> Back): These two selections control whether the part is allowed to be manipulated between bends. This can be used for heavy or awkward parts to ensure that any bend sequences generated will not result in the part being flipped or turned. The default value for both is YES, to allow flips and turns.

Part Manipulation Priority: This selection is used to rank the efficiency of the bend sequence in terms of which is a more time sensitive operation. After generating all of the bend sequences they are ranked by the number of flips and turns. The user selects which operation is less efficient. If neither, the default value of EQUAL is used.

Calculate Minimum Ram Opening: This selection controls whether the software calculates a minimum value for the Ram Opening when the actual program is written from the bend sequence. Otherwise the default value of max ram opening will be used.

Pinch Point Dimension: This selection controls the dimension of the offset from the front of the ram which is considered a safety pinch point. If the part should contact anywhere within the area, it will be considered as interference. The user may adjust this value accordingly. The default value is 1.000”.

Specified Flange to Include Blank Error: This field allows the user to select the flange of the part which will accept any blank error which may be present. The gaging and bend sequence will be determined such that this flange absorbs any blank error. The default value of NONE does not check for this condition. The user should enter the ID of the flange to accept the error. Enter the letter code for the flange using the keypad.

Gage Priority: This selection is only applicable if both a CNC Frontgage and CNC Backgage are installed on the machine. When calculating the correct gage position for a bend, there is the possibility that both the backgage and frontgage could be used. This selection controls which gage is used for this case.

R-AXIS Default: This selection is the value to be used as a relative offset for the R-Axis above the top of die. The R-Axis Gage Data value for each step is set to this value when a program is created. If the material thickness is less than this value, the R-Axis default value used is 0.010” less than the material thickness. The default value for this field is 0.050”.

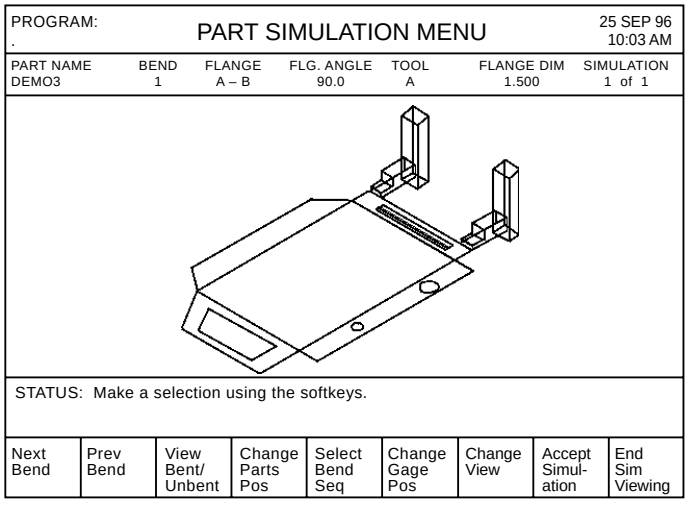
Forming Speed Default: This selections determines the forming speed which is set for each step in the RAM DATA screen when a program is created for a simulation.

VIEWING A SIMULATION

View Simulation: This softkey, when pressed, will change the screen to show a graphical display of the part and gage fingers. The current bend and gaging information is listed at the top of the screen. It shows the number of valid simulations found and the current number being viewed. To step through each bend, use the "Next Bend" and "Prev Bend" softkeys. To see the results of the current bend, bent or un-bent, press the "View Bent/Unbent" softkey to toggle back and forth.

The "Select Bend Seq" softkey can be used to look at all of the bend sequences which were found. If AUTO SEQUENCE is Off and the part has interference with the machine, it will be noted with a message for which object is in conflict. This message will appear in the Status Area. If more than one object is conflicting, only the first one will be displayed. The user will have to correct the situation and re-generate the simulation to find other sources of conflict.

The part will be drawn and identified using various colors. The outline of the part changes when it is flipped. The current bend is drawn in a different color than the other bends on the part.



Next Bend, Prev Bend: These softkeys will change the display to show the various bends of the selected sequence. Each time the display area will update with a new graphic showing the press brake, part, tooling and gage positions. The text information at the top of the screen will also update.

View Bent/ Unbent: When pressed, this softkey will change the view for the current bend. The graphic will show either the bend in its formed or pre-bent state. It will apply to whatever view is selected.

Change Parts Pos: This softkey allows the part to be re-positioned along the die to either the center of the die, left edge or right edge of the upper or lower dieset. The choices are shown below.

Left L Tool Edge	Center Lower Tool	Right L Tool Edge	Left U Tool Edge	Center Upper Tool	Right U Tool Edge			Abort

Select Bend Seq: When pressed, this softkey will change the softkeys as shown to allow the user to step through all of the valid simulations found. The top line of the display area lists the number of simulations found and which one is being viewed.

Next Seq	Prev Seq							Abort

Change Gage Pos: This softkey, when pressed, will present the following options for changing the gage location for the current bend.

Rotate Part		No Gaging	Toggle FR/BK Gage	Manual Adjust Gage	Toggle Step BK Finger			Abort

Rotate Part: This softkey, when pressed, will rotate the part 180 degrees about the center line of the current bend (front-to-back). The flange which is closest to the gage will be selected as the new gage point. The Flange Dimension is then updated on the top of the screen. Note that if the part now violates the weight ratio rule an error message will be displayed.

No Gaging: This softkey will select no gaging for the current bend. The backgage position is set to the maximum to move it out of the way.

Toggle FR/BK Gage: If both a CNC Front and Backgage are installed, the user may toggle which gage is used for the current bend. The graphic will update to show the part using the selected gage. This softkey will only appear if both gages are installed.

Manual Adjust Gage: In order to pick an alternate gaging location such as a segment of another flange, the user may manually select the gage surface. The user must first select which axis and gage finger which is to be moved using the softkeys.

Left X-Axis	Left R-Axis	Left Z-Axis	Right X-Axis	Right R-Axis	Right Z-Axis		Accept Moves	Cancel Moves

STATUS: Enter the dimension or use the field keys to increment.

The left and right field keys are used to change the position of the gage fingers. The finger will move in increments of 0.1" each time a field key is pressed. When moving the X-Axis, the right field key moves the gage backward, the left forward. When moving the R-Axis, the right field key moves the finger up, the left down. When moving the Z-Axis, the right field key moves the finger to the right, the left field key moves to the left. The gage position may also be entered using the keyboard if the exact dimension is known.

The screen will show the actual positions of the gages as they are manually changed. Note the heading at the top of the screen changes from Flange Dimension to Gage Dimension. It also shows the left and right gage positions independently. To accept the new location, press the "Accept Move" softkey. To cancel the move and revert back to the prior gage location, press the "Cancel Move" softkey.

When the X-Axis is selected to move an additional softkey is shown, "Auto Locate X:. This softkey will automatically move the X-Axis so it finds the closest edge of the part to gage from. If the R-Axis position has been changed so it no longer contacts the part, an error message **"Gage Finger does not contact part"** will be displayed.

Toggle Step BK Finger: This softkey, when pressed, will manually specify that a stepped gage finger is to be used. The material will default to laying on the gage finger against the step as a sheet support. All gage positions will be compensated accordingly. If the stepped finger is not currently being used, the following error message will be displayed **"Step Finger must be enabled in the Config Design Menu"**.

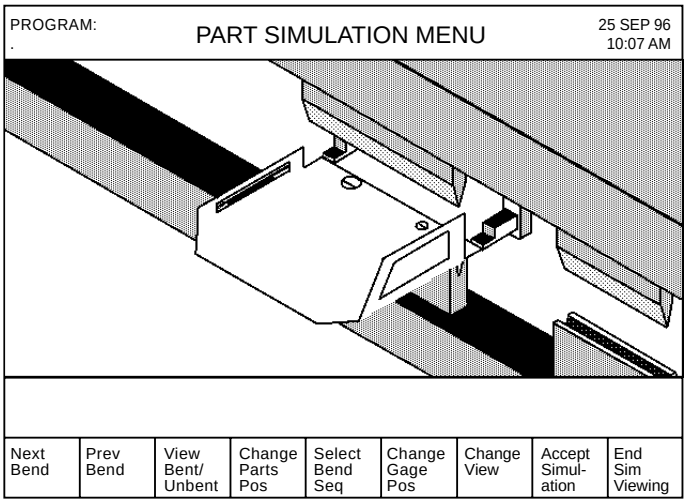
This completes the discussion of options for the "Change Gage Position" softkey.

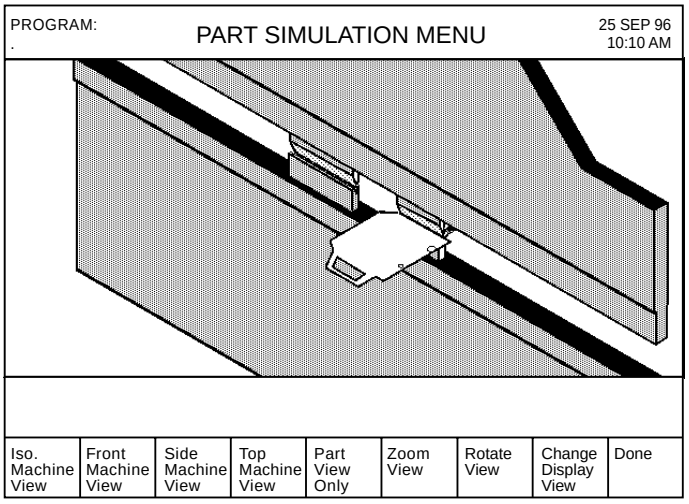
Change View: This softkey changes the graphic which is displayed. The options available are selected using the following softkeys.

Iso. Machine View	Front Machine View	Side Machine View	Top Machine View	Part View Only	Zoom View	Rotate View	Change Display View	Done

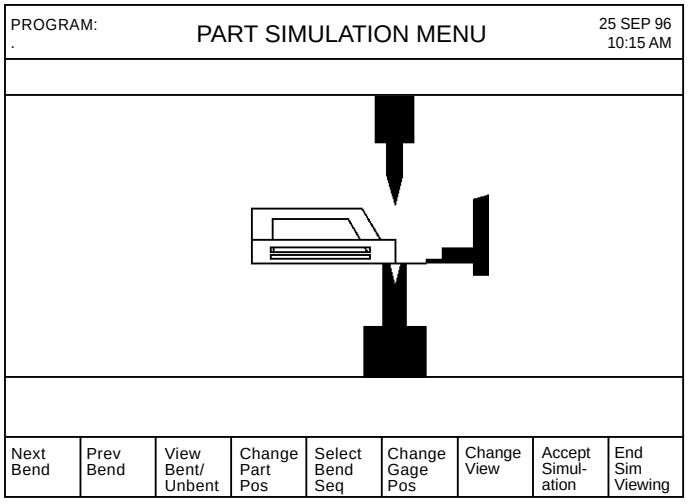
Iso. (Isometric) **View:** This option changes the display to show an isometric view of the press brake including the filler block, tooling, ram, part and gages.

Front Machine View: This option changes the display to show a view of the machine, part and tooling looking directly at the front of the machine.



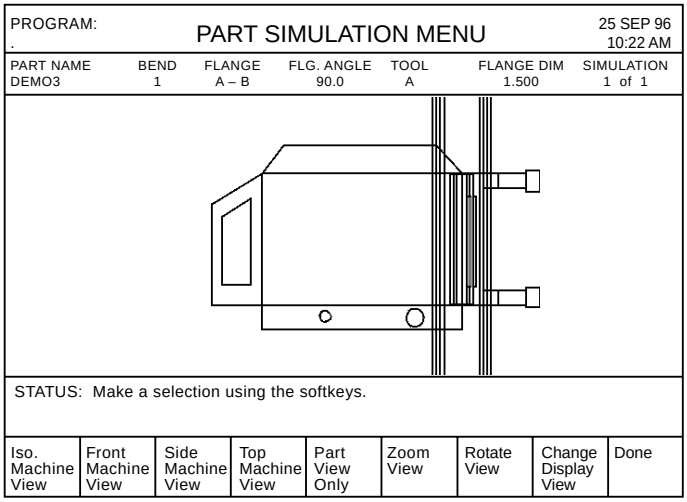


Side (Right) **Machine View:** This option changes the display to show a 2D view of the machine, part and tooling looking at the right view of the machine.



Top Machine View: This option changes the display to show a 2D view of the machine, part and tooling looking directly at the top of the machine. This view

is helpful in determining the position of the backgage along the Z-Axis.



Part View Only: This keeps the view the same except for only showing the part itself. The tooling and machine are not shown.

Zoom View: This softkey brings up the following choices for zooming in or out on the view.

Zoom Window	Zoom Center	Prev View	Home View	Move Center Point	Zoom Part	Zoom Out		Done

These are similar to all of the previous screens where the Zoom function is available.

Change Display View:

Toggle Shading	Toggle Hidden Lines							Done

Toggle Shading: This option will redraw the part using different shades of color to give the part the look of a real part. Shading colors are user selectable in the CAD PREFERENCES screen.

Toggle Hidden Lines: This option does not shade the part but instead removes lines which should be hidden from the users viewpoint.

It is best to rotate the part to the desired viewpoint before using these options due to the extra time required to calculate a shaded model.

CREATING A PROGRAM FROM THE SIMULATION

Accept Simulation: When pressed, this softkey will take the current simulation being viewed and automatically construct the bending program to make the part. All other simulations found will be deleted at this time.

The user is prompted to confirm that the existing program in active memory will be overwritten with the program for this simulation. If yes, the user is then prompted to enter a name for the program.

The software will now begin calculating the correct values to complete the bend program. Once complete, the program and the part simulation will be saved automatically to internal storage.

The following program values are calculated. All others not listed are set to default values.

JOB DATA

PART NAME: Assigned the name of the CAD part file used to create the program
MATERIAL THICKNESS
MATERIAL TYPE
K-FORMING FACTOR
UPPER AND LOWER TOOL SETS (26 possible)

RAM DATA

STEP
REVERSAL MODE - Angle, Adaptive, Position, Abs Position or Tonnage
REVERSAL VALUE - Angle or Position
BEND LENGTH
RAM OPENING
FORMING SPEED
TOOL

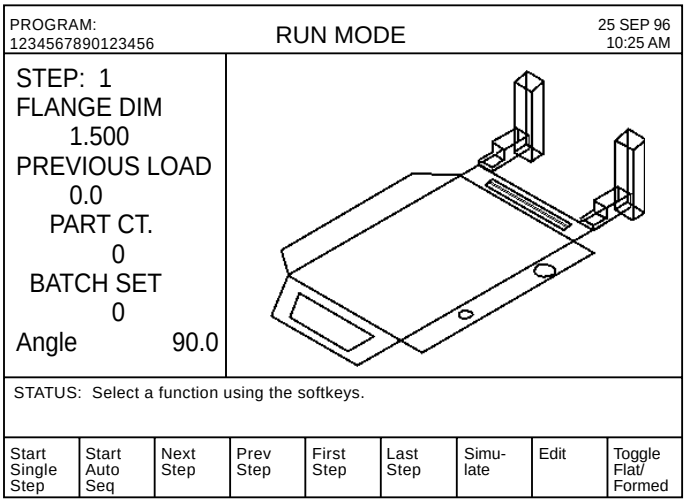
GAGE DATA

GAGE MODE - Based on position of gages from simulation, either standard or independent.
LEFT/RIGHT FLANGE:
GAGE ALLOWANCE
RETRACT DISTANCE
FINGER OFFSET - used if stepped finger is selected
RLEFT,R-RIGHT: If CNC backgage with programmable R installed
Z-LEFT, Z-RIGHT: If CNC backgage with programmable Z installed
FR-LEFT, FR-RIGHT: If CNC front gage installed

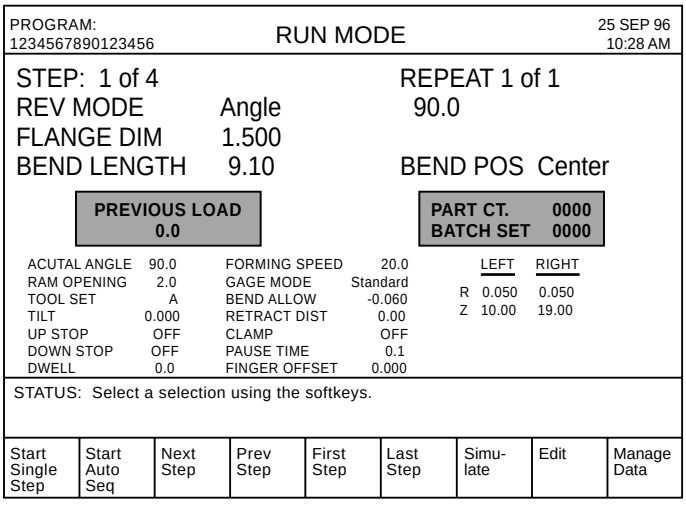
The program may be edited after it is created. Steps may be added manually for such operations as hemming or special gage manipulation. Note when editing a program that the graphic simulation may no longer correspond to the changes.

RUNNING THE PROGRAM

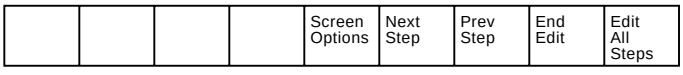
When running a program created from the Part Simulation, there are several options on how the Run Mode screen appears. Shown is the view referred to as the Normal, 3D Part View.



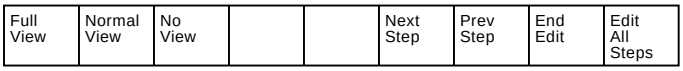
The screen will update automatically as the ram cycles through each step. Displaying the 3D part does not affect the operation of the machine. The "Next Step" and "Prev Step" softkeys will advance through the bend sequence updating the screen each time. Use the "Toggle Flat/Formed" softkey to view the current bend before and after. The "Edit" softkey when pressed will clear the screen and bring up the standard RUN screen allowing the user to edit any values as needed.



The softkeys will change as shown after pressing "Edit".



Pressing the "Screen Options" softkeys brings up the following choices for how large the view should be.

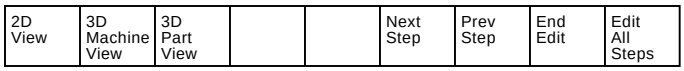


Full View: This view uses the entire screen to show the graphic. It does not show any step information.

Normal View: This view divides the screen to allow some step information to be displayed along with the part graphic. The step information cannot be edited at the same time the part graphic is shown.

No View: This is the standard text only RUN Screen without any part graphic.

The next three choices determine the appearance of the part graphic.



2D View: This view is the 2D side view of the machine and part. It is similar to the side view shown in the PART SIMULATION screen.

3D Machine View: This view is the 3D isometric view of the entire machine and part.

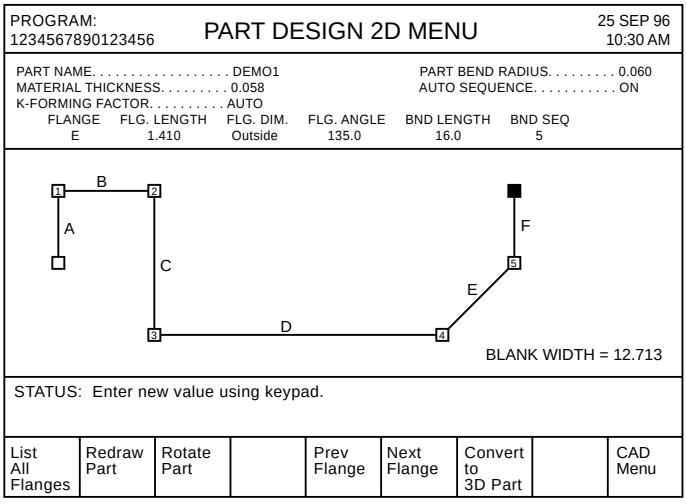
3D Part View: This view eliminates the machine to show only the part and gage fingers.

The view settings will remain until they are changed here or until a new part is loaded from Program Storage and the settings are changed at that time.

Note that if steps are added to the program manually, the RUN MODE screen will blank the part graphic for those steps.

PART DESIGN 2D MENU

This screen allows the user to design a formed part model by describing the 2D cross section. It can be used to design simple parts very quickly. This screen is carried over from the existing Graphics software with the addition of the Convert to 3D Part softkey.



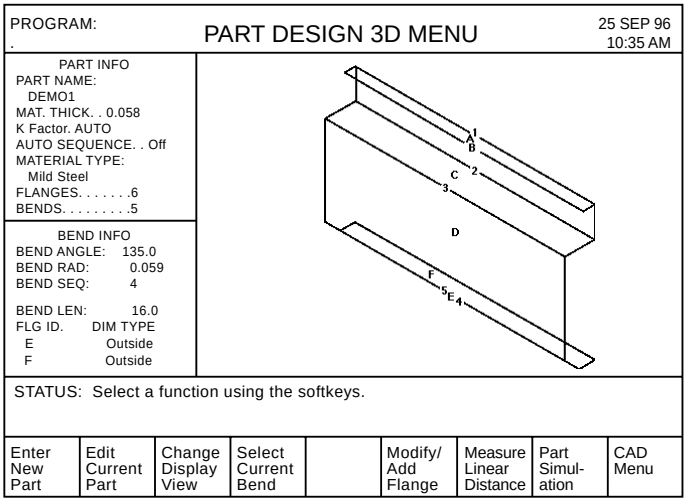
This screen accepts from the user the length, angle and dimension type of each flange of a part. The user also may specify the bending order, part bend radius and bend length of each flange.

As the user enters the flange information, the part cross section is drawn on the screen. The Blank width is calculated based on the ideal bend radius and k-factor optionally supplied. The part scale factor is calculated to allow the full part to be viewed on the screen.

The drawing is displayed in the viewing window. Bend sequence is identified for each bend and flanges are labeled with a single alphanumeric character. The current flange information is displayed at the top of the screen and can be edited. The blank width is displayed in the bottom right corner of the viewing area.

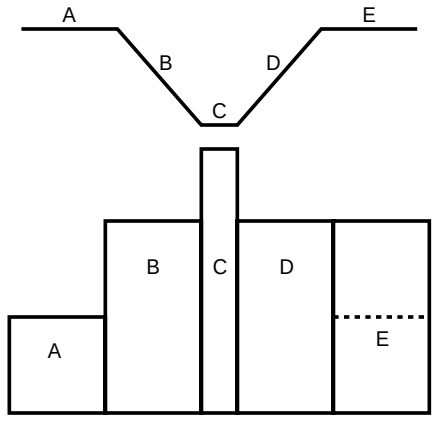
CONVERT TO 3D PART

When pressed, this softkey will extrude the 2D cross section using the flange length, bend length and angle, into a 3D part. All other information including Part Name, Material Type and Thickness are also transferred. Proceed to the 3D PART DESIGN screen to view the converted part.



This function makes it convenient to enter simple parts into the 3D PART DESIGN screen without having to describe the individual flanges using the detailed method of 3D.

Note that a 2D part when it is transferred into a 3D model will use the Bend Length field to determine the width of each flange. The transition between flanges which have different bend lengths may cause some confusion. The following example shows how a 2D cross section with different bend lengths will be converted. This example shows the cross section and the flat pattern.

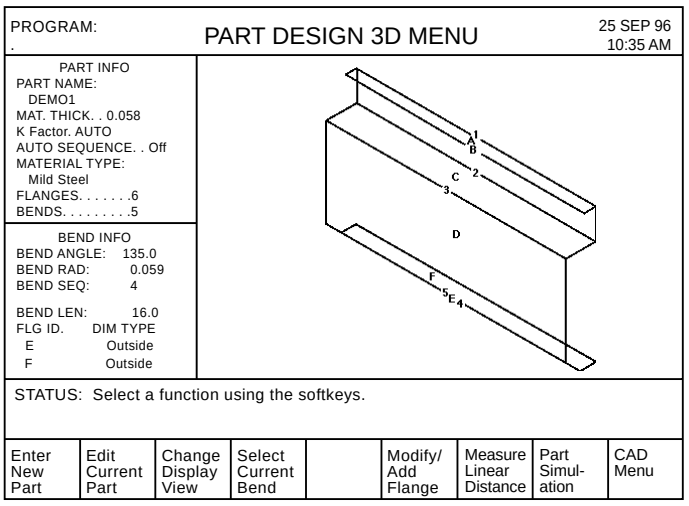


Note the last flange may have to be adjusted because its bend length is always defaulted to the previous flange. This is due to the fact that there is no associated bend length for the last flange.

At this point the part can now be simulated in the PART SIMULATION screen.

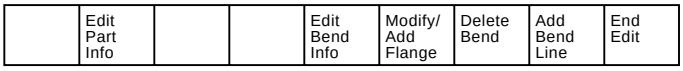
PART DESIGN 3D MENU

This screen allows the user to either edit a 3D part which has been converted from a DXF or design a formed part from scratch.



Enter New Part: This softkey, when pressed, will first prompt the user to confirm a new part should be created. If so, the current part is deleted and the drawing area will be erased. The user will then be prompted to enter the data in the Part Info section of the screen.

Edit Current Part: This softkey will present additional softkey choices for editing the Part Info and Bend Info items, modifying flange geometry and adding or deleting Bends.



Change Display View: This softkey will change the softkeys as shown in the following.

Display Formed Part	Display Flat Pattern	Toggle Hidden Lines	Toggle Shading	Zoom View	Rotate View	Toggle Labels		Done

Display Formed Part: This softkey switches the drawing view to the formed part model. The drawing will be displayed with all flanges entered up to this point in 3D. If no bends have been defined, the user will be instructed to do so with a status line error message.

Display Flat Pattern: This softkey switches the drawing view to the flat pattern view. Bends must be defined first.

Toggle Hidden Lines: When displaying the formed model, this softkey switches between displaying the model with hidden lines being shown.

Toggle Part Shading: When displaying either formed or flat, this softkey switches between displaying the model with shading for better realism.

Zoom / Rotate: This softkey allows the current part being shown to be rotated or zoomed to provide a better view. The same Zoom and Rotate functions are available in other screens.

Toggle Labels: This function controls whether the part is labeled with flange ID and bend sequence. Flanges are identified using alpha numeric labels while bends are labeled using numbers. These can be turned off and on using this softkey.

Select Current Bend: This softkey when pressed will allow the user to change the current bend noted in the Bend Info of the screen. The user may specify the bend by using the mouse to pick a bend line on either the formed or flat pattern drawing. They may also use the softkeys below to specify the bend. The current bend line will be highlighted on the drawing.

Next Bend	Prev Bend	First Bend	Last Bend					

Modify/Add Flange: This softkey when pressed will change the softkeys to allow the user to begin drawing a new flange or editing an existing one.

Create New Flange	Modify Current Flange	Select Current Flange	Delete Flange	Copy Flange	Flip Flange Vert	Flip Flange Horiz		Modify Part

Measure Linear Distance: This is the same measure distance function which is available on the CONVERT DXF screen. It can be used with either the formed or flat view of the part.

EDITING AN EXISTING PART

	Edit Part Info			Edit Bend Info	Mofify/ Add Flange	Delete Bend	Add Bend Line	End Edit

Edit Part Info: This softkey will highlight the PART NAME field listed under the PART INFO section of the screen. The mouse or field keys may then be used to edit the other fields including MAT. THICK., K Factor, AUTO SEQUENCE and MATERIAL TYPE. The FLANGES and BENDS fields cannot be edited directly.

Edit Bend Info: This softkey will highlight the first field in the BEND INFO area. The user may then edit any of the values listed. They will be applied to the current bend. Pressing the "End Edit" softkey will update the drawing to show the changes made. It is not possible to edit the BEND LEN field which is calculated. It is not possible to change which flanges are connected by this bend, other than their dimension type. To redefine the bend, first use the "Delete Bend" softkey.

Modify/Add Flange: This softkey will change the softkeys to allow the user to begin drawing a new flange or editing an existing one.

Flanges can be edited to adjust the outline or to add or remove internal features. Flanges can be copied and then rotated to create new ones. All of these options will be covered under the next section Creating a 3D Part.

Delete Bend: This softkey will delete the current bend as shown in the BEND INFO area. The flanges are not deleted, only the bend. The user will be prompted yes or no to confirm deletion of the bend. Upon confirming the decision, the bend will be deleted. If there are other bends connected to flanges for the current bend, an error message will appear **"Cannot delete Bend, delete attached ones first"**.

Add Bend Line: This softkey is used to add a new bend line to the part by specifying which flanges are connected to form a bend line. The screen will change to show a graphical list of all flanges entered so far in the area where the PART and BEND INFO is now shown. The user will be prompted to first select the flange to be connected. Next they are prompted to enter the flange to connect to. It will also be necessary to select a segment on each flange to designate it as the bend line. Once this is done the remaining information in the BEND INFO area may be entered using the keyboard. Bend lines may only be added if there are unused flanges available to be connected. If a flange is already

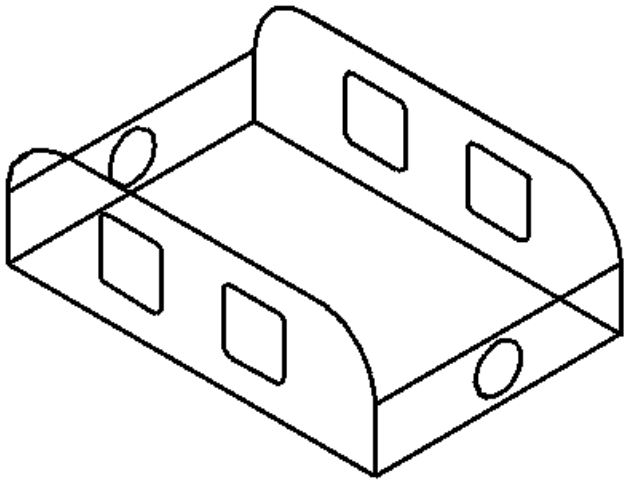
used, an error message **“Flange is already connected by bend”** will be displayed. If all flanges are already connected, the message **“All Flanges have been connected by bends”** will be displayed.

When defining a bend line, the user will also be prompted for whether the bend lines should connect at the start or end of the segment selected for each flange. Note that flanges are connected in the same orientation that they are drawn. It may be necessary to flip the flange to obtain proper orientation. This may be done in the Flange drawing screen using the “Flip Flange” softkey (Enter Bend Data: After selecting a bend line it is then necessary to enter the other bend parameters, such as the desired bend angle, bend sequence, dimension type and bend radius. The FLANGE INFO field will be highlighted upon pressing this softkey. The user may then use the softkeys and keypad to enter the information.

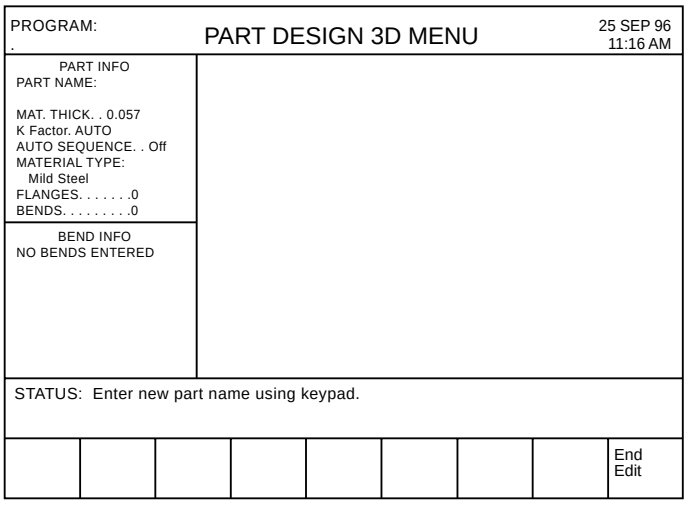
CREATING A 3D PART

The design process involves defining each flange of the part individually and then assembling them to complete the model. Flanges are drawn using 2D techniques to avoid the complexity of drawing in 3D. Specific features such as holes and slots can be inserted into a flange.

The following example will show how to create the 3D part shown below. The part consists of five flanges and four bends. Two of the flanges contains two internal slots. The other two contains one large internal hole.



Begin by pressing the “Enter New Part” softkey. Press the “YES” softkey to confirm that the existing part may be deleted.



COMPLETING THE PART INFORMATION

The screen will clear and the cursor will appear under the PART NAME field. The status line will change as shown requesting a name for the part.

STATUS: Enter new part using the keypad.

For this example, enter the name SIMPLE3D and press the ENTER key.

The field will move down to the MAT. THICK.field. For this example, leave the default value by pressing ENTER.

The next field will be K-Factor. The K-Factor is used to calculate bend allowance. It can be set to AUTO, in which case the value is calculated based on the material thickness and radius, or the K-Factor can be changed to FIXED and a value entered by the user. For this example leave it set to AUTO. For additional information on bend allowance calculation, see the *APPENDIX B*.

The next field is AUTO SEQUENCE. This is the same field which is discussed under the PART SIMULATION MENU which controls how bend sequences are determined. Leave it as the default value.

The last field under the PART INFO section is the MATERIAL TYPE. Select the softkey which corresponds to the material type. This is used to adjust for springback during bending. For this example leave it as ‘Mild Steel’.

Press the “End Edit” softkey.

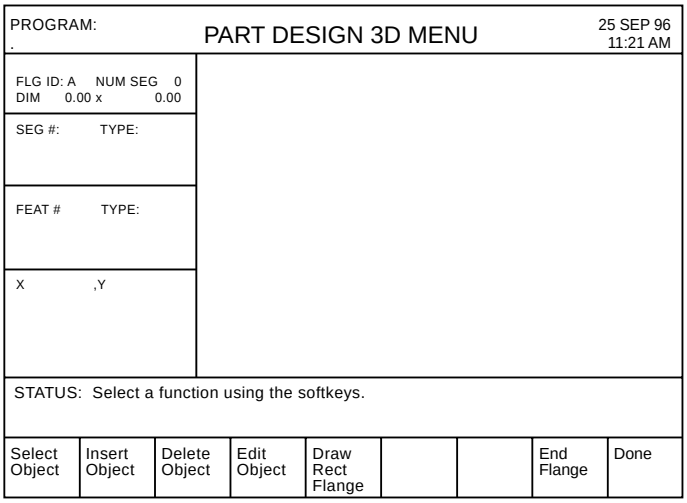
CREATING FLANGES

The next step is to draw out each of the flanges which make up the part. The process of drawing a flange involves constructing the outline of the flange from simple line and arc segments. Internal features such as rectangles, slots and holes can be specified.

To draw the first flange, press the “Modify/Add Flange” softkey. The softkeys will change as shown.

Create New Flange	Modify Current Flange	Select Current Flange	Delete Flange	Copy Flange	Flip Flange Vert	Flip Flange Horiz		Modify Part

Press the “Create New Flange” softkey. The screen softkeys will then change as shown. Note the FLANGE INFO section of the screen lists the FLG ID as ‘A’ and that it has no segments or dimensions yet.



The first flange to be drawn will be the simple rectangular base of the part. Since this is just a rectangle, the softkey labeled “Draw Rect Flange” may be used. This allows a flange to be created by simply entering the length and width of the rectangle.

Press the “Draw Rect Flange” softkey.

The status line will now prompt as shown.

STATUS: Enter the Length of the Flange using the keypad 0.000

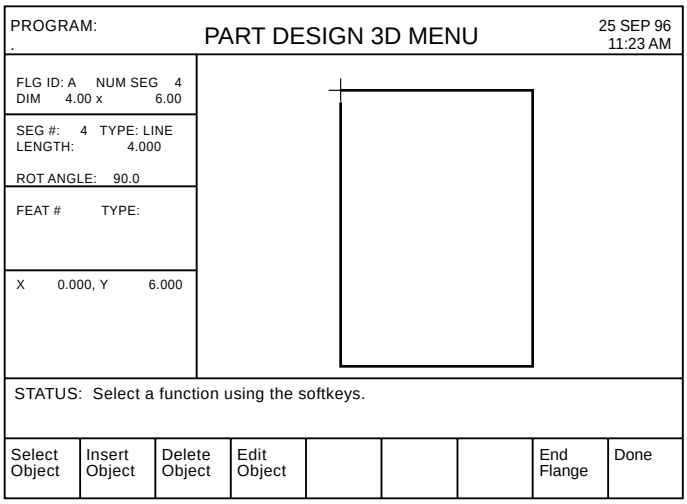
For this example, enter the value of 6.0 inches.

Next the status line will prompt.

STATUS: Enter the Width of the Flange using the keypad 0.000

For this example enter the value of 4.0 inches.

The screen will now show the rectangular flange which was just created



Note that the flange displayed actually consists of four separate line segments which were generated automatically from the length and width dimension. These segments could have also been created individually which will be illustrated on the next flange.

Press the “Select Object” softkey and then press the “Segment” softkey.

Select Object	Insert Object	Delete Object	Edit Object				End Flange	Done

Segment	Feature							Abort

Next Segment	Prev Segment	First Segment	Last Segment					Done

Press any of the above softkeys to step though each of the individual line segments of the part. Note the SEGMENT Info section of the screen changes to show the segment length and its rotation angle. The flange drawing will change to show the current segment in brown. The white cross hair + indicates the start of the line segment. The X, Y coordinates always show the relative x, y distance from the previous segment.

To complete this flange, first press the “Done” softkey and then the “End Flange” softkey.

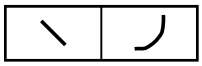
The next flange of the part will now be created. This flange contains arcs so it is not possible to use the simple “Draw Rect Flange” softkey.

CREATING A FLANGE USING SEGMENTS

Each of the four softkeys for selecting, inserting, deleting or editing an object will act on either a Segment or Feature of the flange. After pressing one of these softkeys, the option below will appear prompting the user to select which object should be acted upon.

Segment	Feature							

The following description of softkey functions assumes that the user has pressed the corresponding object function softkey (“Select Object”, “Insert Object”, “Delete Object” or “Edit Object”) and the object type softkey (“Segment” or “Feature”). Once the user has selected either the “Segment” or “Feature” softkeys, the display will change to show the choices available. For segments, the choices are LINE or ARC.



Select Segment: After pressing the “Select Object” softkey followed by the “Segment” softkey, both the up and down field keys will be activated. The user may then select a given segment which has already been drawn. The field keys will highlight each segment beginning with the first.

Insert Segment: Pressing the “Insert Object” softkey and then the “Segment” softkey will allow a new segment to be added to the flange. If this is a new flange, the first segment will be drawn. The new segment will be inserted after the current segment. Upon pressing this softkey, the user will be prompted to enter the type of segment to be drawn by using the softkeys which appear as shown below.

Line Segment	Arc Segment							Abort

The user is then prompted to enter the corresponding dimensions of the segment. This can be done with the softkeys and the numeric keypad.

For a line segment, the user must specify the length of the line and the angle to be drawn from. The angle notation is the same as that used in the 2D Part Design where the included angle is entered with a sign for direction. All angles are relative to the previous segment.

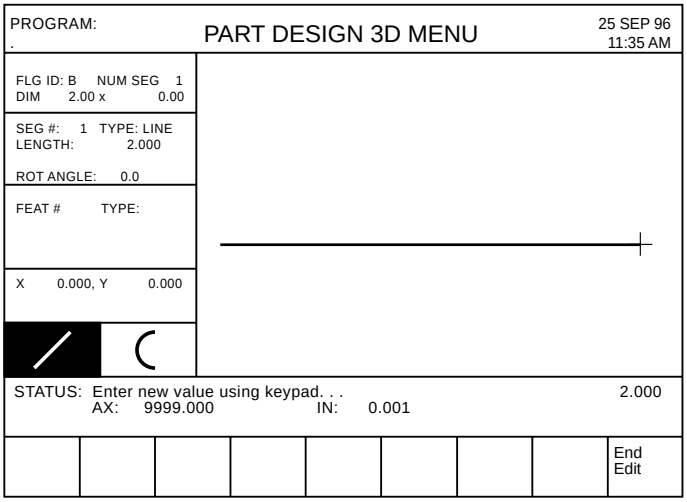
For an arc segment, a similar technique is used. In this case, it is necessary to first enter the direction angle, then the arc radius and arc angle.

Delete Segment: Pressing the “Delete Object” softkey followed by the “Segment” softkey, will delete the current segment or last segment in the case of a new flange. If a segment is deleted which is connected in between two others, the remaining segments are joined to fill the gap. The current segment is changed to the one before the deleted segment. It is not possible to delete a segment with attached features. Features must be deleted first.

Edit Segment: After pressing the “Edit Object” softkey followed by the “Segment” softkey, this softkey will highlight the data area corresponding to the current segment. The user may then enter in new values for the length and direction of the segment using the keypad.

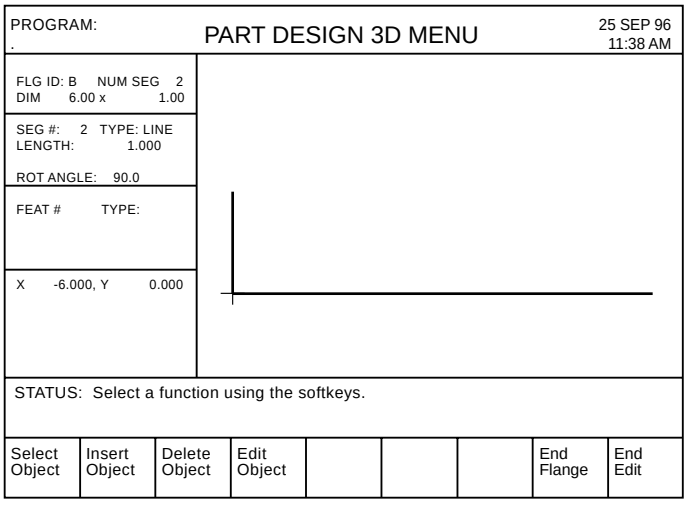
For this example, begin by pressing the “Create New Flange” softkey. Note the FLG ID has now become ‘B’. Press the “Insert Object” softkey followed by the “Segment” softkey. This first segment will be a line, so press the “Line Segment” softkey.

The screen will change to show a single line segment. The status line will change requesting a value to be entered. Note that the LENGTH field of the Segment Info area is highlighted. The default value is 2.000 inches.



For this example, the default value needs to be changed to 6.000 inches. It is now prompting to change the rotation angle of the line segment. Leave this at zero.

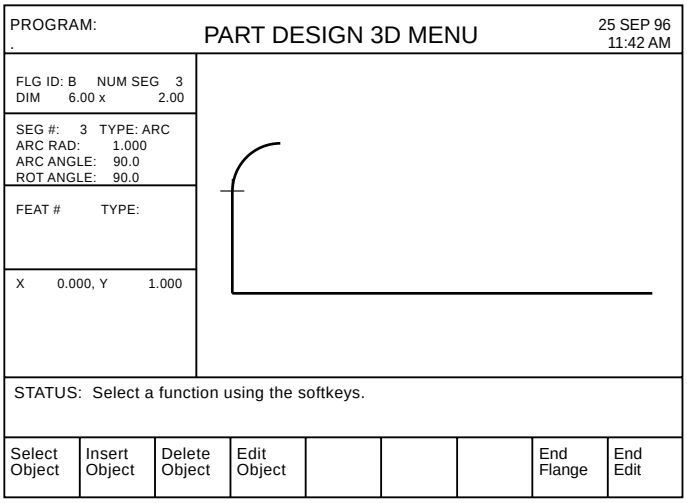
Press the “Insert Object” softkey and add another line segment. For the Length enter 1.000 and the rotation angle enter 90.0



Note how the rotation angle changes the direction of the segment. The angle is always relative to the previous segment.

Add another segment by pressing “Insert Object”. This time pick the Arc Segment instead of a line.

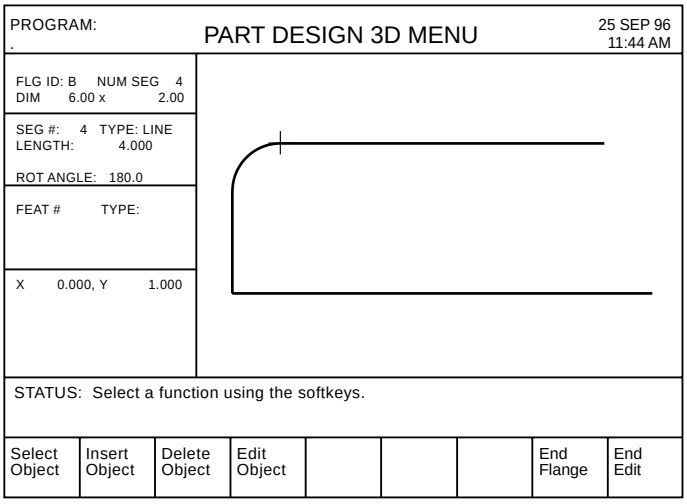
For the Arc Radius, enter 1.000. Leave the Arc Angle and Rotation Angle as default.



Note the arc has now been added to the flange outline.

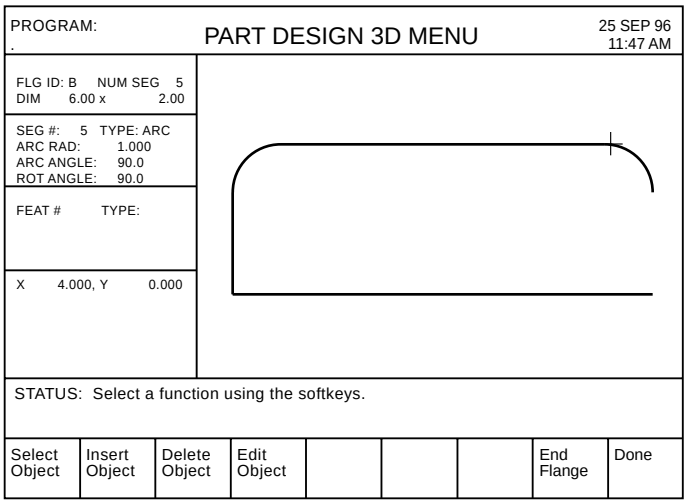
Next we will add another line segment as follows.

LENGTH 4.000
ROTATION ANGLE 180.0



Next we will add another arc Segment as follows:

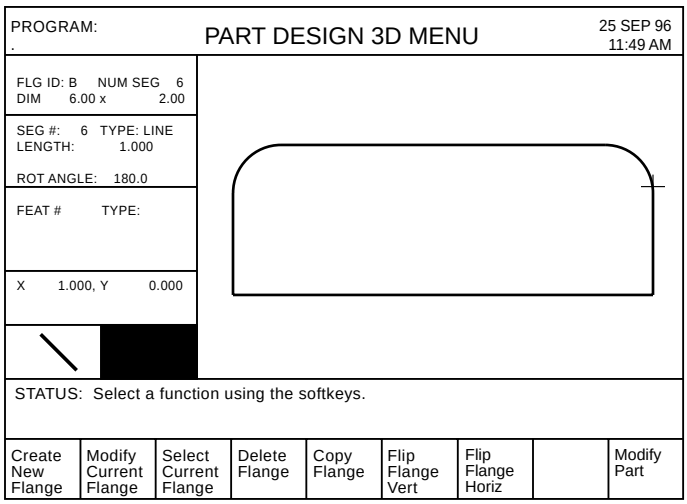
ARC RAD 1.000
ARC ANGLE 90.0
ROT ANGLE 90.0



Note that as segments are added the length and rotation angle always defaults to the same as the last segment of that type.

Now only one segment remains to complete the flange. The last segment will automatically be connected to the starting segment if the “End Flange” softkey is pressed.

Press the “End Flange” softkey.



Flip Flange: This softkey reverses the orientation of the flange. It is useful when flanges get connected as bends to ensure proper alignment. A flange may be flipped horizontally or vertically.

End Flange: This softkey, when pressed, will signal that the flange is complete. If the outline of the flange is not closed, then it will be closed automatically by inserting an additional line segment connecting the first and last segments. If the user has drawn an incomplete flange (i.e., only one segment or has segments which cross one another) they will be notified with an error message and the flange will not be accepted.

ADDING FEATURES TO A FLANGE

Features should be used primarily as an aid in orienting parts. Since the process of locating a feature takes several steps, features should be used on an as needed basis, rather than attempting to completely design all features of a production part.

Select Feature: Pressing the "Select Object" softkey followed by the "Feature" softkey, will activate the up and down field keys. The user may then select a given feature which has been drawn. The field keys will highlight each feature beginning with the first.

Insert Feature: Pressing the "Insert Object" softkey followed by the "Feature" softkey, will allow a new feature to be added to the flange. Note there must be at least one segment previously drawn before entering a feature. Upon pressing this softkey, the user will be prompted to enter the type of feature to be drawn. This may be done by using the mouse to pick either the Slot, Hole Bolt pattern or Freeform types or by using the softkeys.

Copy Feature: This softkey will appear when inserting a feature. It allows the user to quickly create several copies of the feature using a fixed offset from the feature just specified. The user will be prompted to enter the X, Y offset between the center of the features and the number of times to duplicate the feature.

Delete Feature: Pressing the "Delete Object" softkey followed by the "Feature" softkey will remove the current selected feature from the flange. The user will be prompted to verify this operation.

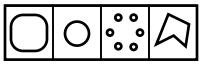
Edit Feature: Pressing the "Edit Object" softkey followed by the "Feature" softkey will highlight the data area corresponding to the current feature. The user may then enter in new values for the various feature parameters using the keypad.

Next, two slots will be added to this flange. Begin by pressing the "Modify Current Flange" softkey.

Use the "Select Object", "Segment" and "Next Segment" softkeys to highlight the lower line segment on the flange. Press the "Done" softkey. Now press the "Insert Object" softkey followed by "Feature".

Slot Feature	Hole Feature	Bolt Pattern	Free Form Feature					

The screen will also change to indicate the choices graphically as shown below.

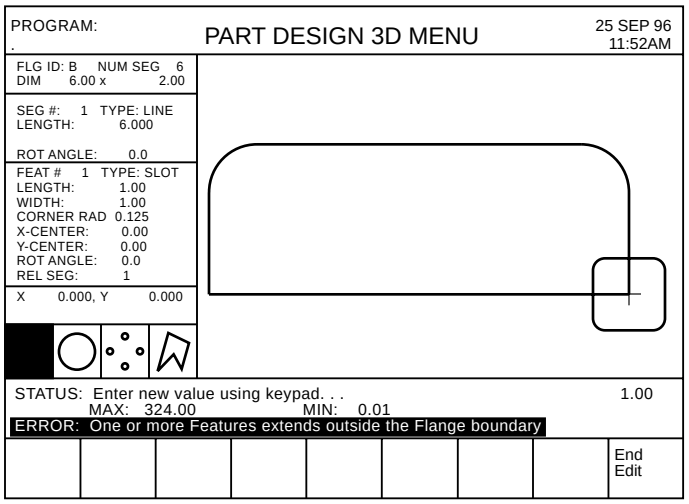


The user will then be prompted to enter the required dimensions of the feature. For a slot, the length, width and corner radius are required. A hole only requires the user to enter the diameter. A bolt pattern requires the number of holes, hole diameter, diameter of bolt circle. The user will be prompted for these values and input made using the keypad.

The user will then be prompted to enter the offset from the start point of the current segment to the center of the desired feature. Using the softkeys, the user will enter the X and Y offset dimension with the keypad. The user will then be prompted to enter an optional rotation angle for the feature. The rotation angle is a value from -180.00 to 180.00 degrees where 0 is the default value, positive rotation is counterclockwise. Note for a bolt pattern, the rotation angle is the angle from the X-axis to the first hole.

For this example press the "Slot Feature" softkey.

The screen will change to show a default slot drawn at the starting point of the lower line segment. An error will also appear stating that the feature extends outside the flange boundary. This is normal and will disappear once the location of the slot is entered.

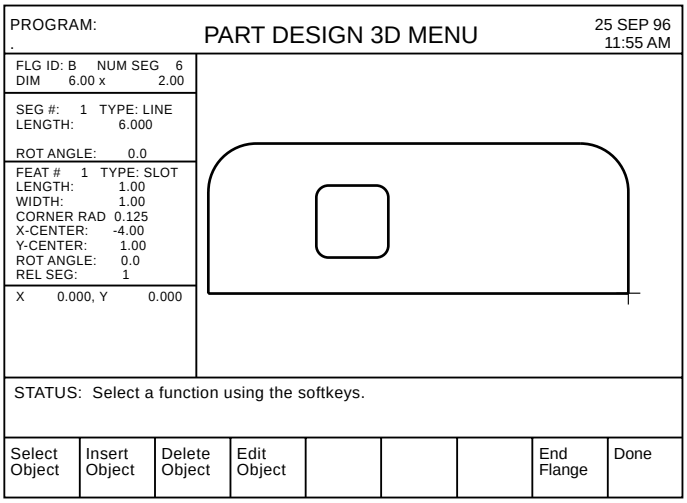


Under the Feature Info section are all of the values affecting the size and location of the feature. For this example leave the Length, Width and Corner Radius as default.

For the X-Center dimension, enter a value of -4.000. This dimension is always relative to the starting point of the current segment indicated by the cross hair cursor. Negative values are to the left, positive values to the right.

For the Y-Center dimension enter a value of 1.000. This dimension is always relative to the starting point of the current segment indicated by the cross hair cursor. Negative values are down left, positive values are up.

Leave the Rotation Angle at the default. Pressing the "End Edit" softkey, the screen should appear as follows:



Note the Feature Info section now contains all of the dimensions which were entered. The feature just added will be highlighted in brown on the flange graphic.

There is no fixed limit on the number of segments and features. There is, however, a limit on the memory available to contain the part. A part which has a large number of simple rectangular flanges will take less memory than a part which has a few very complex flanges with many segments and features on each flange. When an error message occurs stating that there is no more memory available for the active part while designing a part, try deleting features.

DUPLICATE FEATURE

The slot just added will now be duplicated and offset just to the right of itself. Press the "Select Object" and "Feature" softkeys. Now press the "Copy Feature" softkey. The status line will change as shown.

STATUS: Enter X-Offset for duplicate Feature 1.00

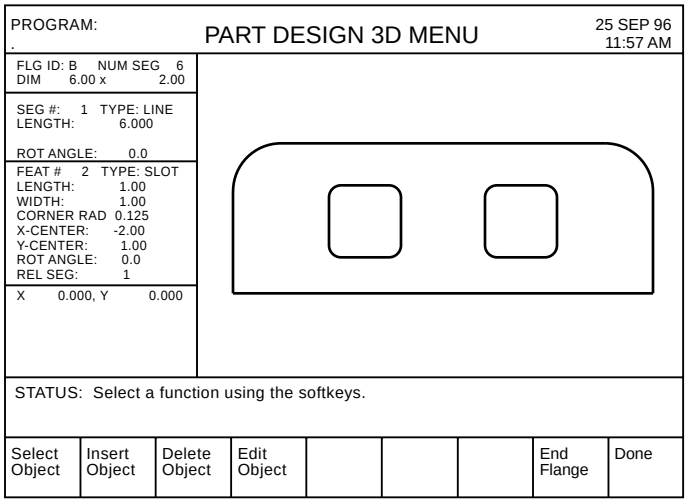
This is requesting the relative distance in the X-direction from the center of the current feature to position the new feature. Enter a value of 2.000.

STATUS: Enter Y-Offset for duplicate Feature 1.00

This is requesting the relative distance in the Y-direction from the center of the current feature to position the new feature. Enter a value of 0.000.

STATUS: Enter Number of Times to duplicate Feature 1

This is asking for the number of copies to be made of the feature. Each time a copy is made it will be offset from the previous feature by the same amount. Enter 1 to make 1 copy of the feature.



Note the flange now has two features and is complete. Press the "End Flange" softkey.

FreeForm Feature: This is a user defined feature made up of line and arc segments. The purpose of this feature is primarily to carry over any features from an imported DXF file, either flat or 3D. It can also be used to enter more complex features when designing the part as well.

In order to create a Freeform feature, the existing flange drawing mechanism is used. First create a new flange, only instead of an actual flange, this will be the new feature. When the Freeform feature type is selected, the user will be prompted to enter the flange ID which is to be converted to a feature. After entering the ID, the usual feature information (position, rotation) can be entered. The flange which was specified as a feature will be deleted. It is not possible to edit the individual segments of a Freeform feature once it has been inserted. The user may adjust the position of the feature or change its rotation as needed.

Note: *It is not possible to specify an existing flange which already has features. If this happens, an error message will be displayed* **"Cannot make Freeform feature from flange containing features"**.

COPYING FLANGES

To draw the third flange on the part, we will take advantage of the fact that they are identical. Use the "Copy Flange" softkey to make an exact copy of the previous flange. Note that the Flange Info screen has now updated to show FLG ID: 'C'.

Use the "Select Current Flange" softkey to step through all of the flanges created so far.

We will draw the last remaining flange once again using the "Draw Rect Flange" softkey.

Enter a length of 1.000 inches and a width of 4 inches. Next add the hole feature in the center of the flange.

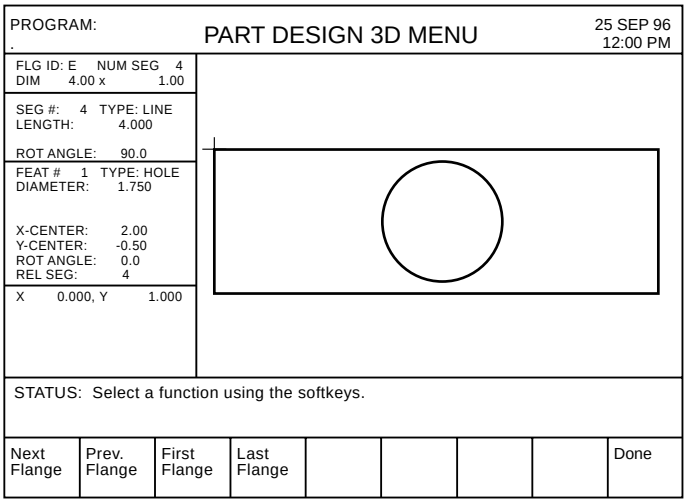
Press the “Insert Object” softkey followed by “Feature” softkey, then the “Hole Feature” softkey.

Next the diameter of the hole can be entered as 0.750.

Enter an X-center value of 2.000 and a Y-center value of -0.500

Press the “End Edit” softkey to view the flange.

Last, duplicate this flange using the “Copy Flange” softkey.



ADDING BENDS

Now that all flanges have been drawn to complete the part they must be connected by specifying bend lines. First press the “Modify Part” softkey to go back to the main set of softkeys.

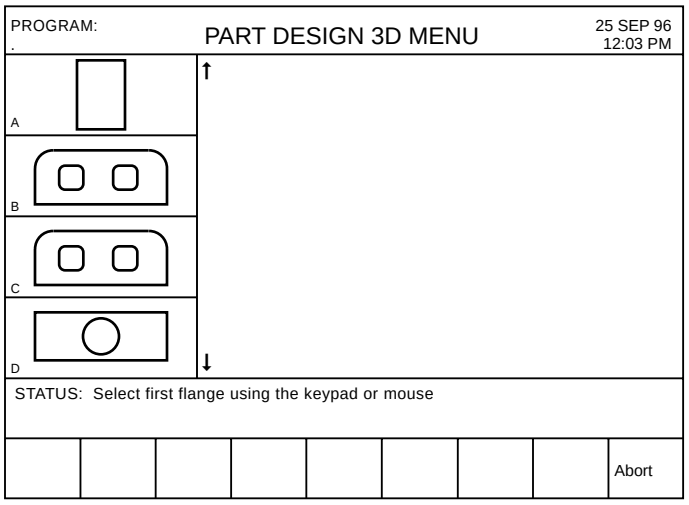
Enter New Part	Edit Current Part	Change Display View	Select Current Bend		Modify/ Add Flange	Measure Linear Dist	Part Simul- ation	CAD Menu

Press the “Edit Current Part” softkey

	Edit Part Info			Edit Bend Info	Mofify/ Add Flange	Delete Bend	Add Bend Line	End Edit

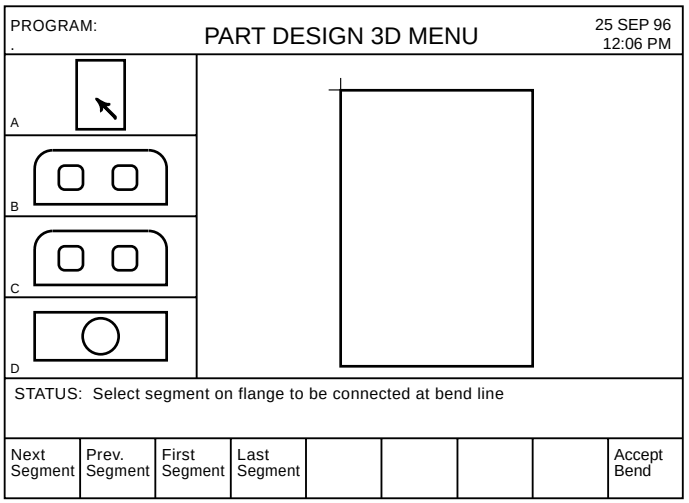
Now Press the “Add Bend Line” softkey.

The screen will change as shown to display a graphical list of the flanges on the left side of the screen. Only four flanges at a time are shown, use the up and down field keys or the mouse to click on the up and down scroll arrows to see additional flanges.



The status line is requesting that the first flange of the part to be attached should be selected from the list using the keypad or mouse. Any of the flanges can be picked as the starting flange. For this example the part will start with Flange ‘A’. Use the mouse to move the cursor anywhere inside the box surrounding Flange ‘A’ and click the left button.

Flange ‘A’ will now appear in the main drawing window. The top segment will be highlighted as the current segment.



The status line will now prompt to select the segment on the flange which will be the bend line. Use the “Next Segment”, “Prev Segment”, “First Segment” and “Last Segment” softkeys to move the highlighted segment to the desired segment.

For this example, press the “Next Segment” softkey to highlight the long vertical segment on the right side of the part. Press the “Accept Segment” softkey.

The status line will now change as follows:

STATUS: Select Alignment of Bend Connection using the softkeys.

Start Edge	End Edge							Abort

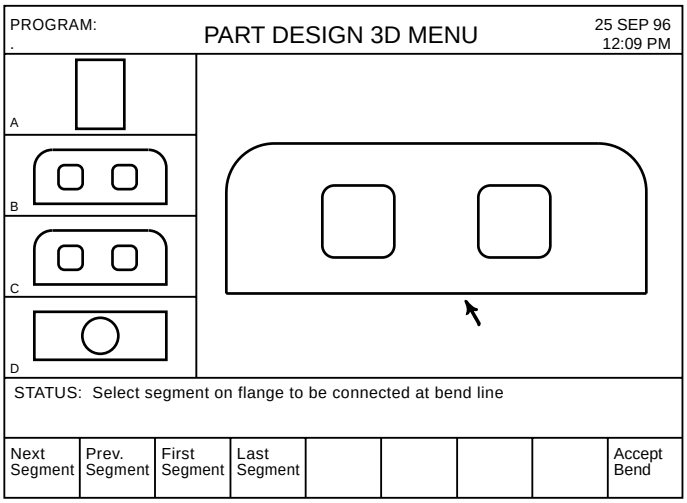
A bend can be aligned only at the start or end edge of a line segment. It is necessary to choose the alignment for both flanges. The start edge of the segment is always indicated by the white cross hair cursor. For this example select the "Start Edge". The status line will now change to:

STATUS: Select flange to attach using the keypad or mouse.

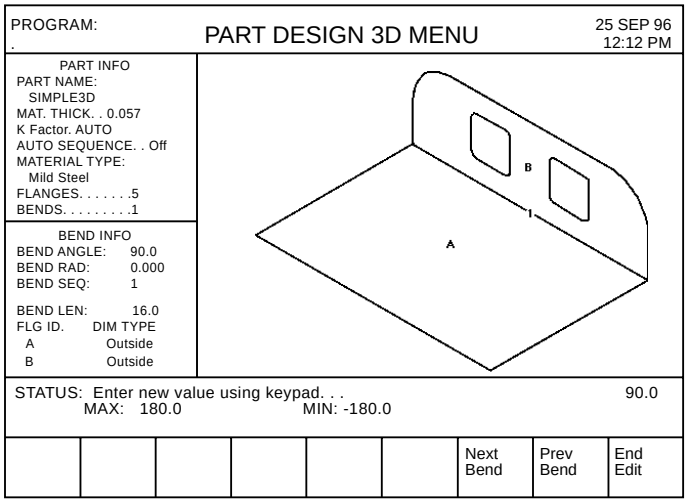
Select the flange to be attached in the same manner. For this example select Flange 'B'. The screen will change to show Flange 'B'.

It will now be necessary to pick the corresponding segment on Flange 'B' which will be attached to Flange 'A'.

Use the softkeys to select the bottom segment on Flange 'B' as shown:



Press the "Accept Segment" softkey. Select the alignment of the Bend connection as before, but this time pick the End Edge. The screen will now change to show the formed part so far with the one bend specified.



BEND INFO VALUES

The bend angle defaults to 90 degrees. The angle can be entered under the BEND INFO section of the screen. After selecting the angle, enter the value for the BEND RAD of the part. The BEND SEQ value can be changed as well, it will default to the order that the bends are defined.

Note the BEND LEN value which is already calculated and cannot be edited. The last two fields specify the dimension type of the flange. Each flange can have a different dimension type. This effects how the part will be unfolded. For example, an outside dimensioned flange must be adjusted or made smaller by the bend radius and the material thickness. See the *APPENDIX B* for additional discussion on Dimension Type and Bend Allowance.

For this example enter the following:

BEND ANGLE: 90.0
BEND RAD: 0.057
BEND SEQ: 1

DIM TYPE
A Inside
B Outside

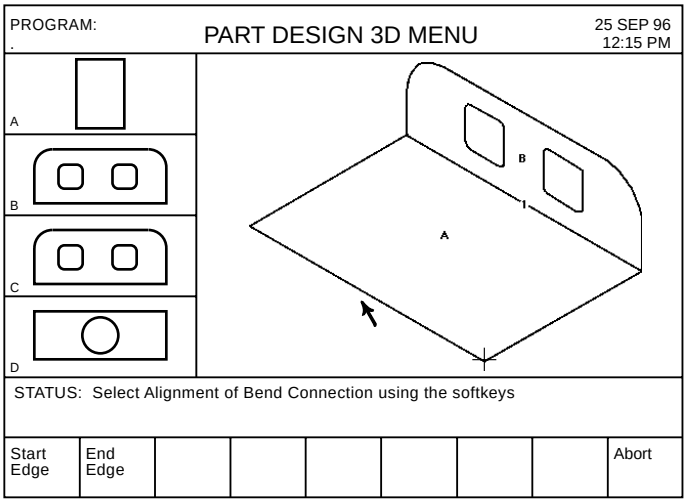
						Next Bend	Prev Bend	End Edit

This completes the information for the first bend of the part.

Use the "Next Bend" softkey to continue with the next bend.

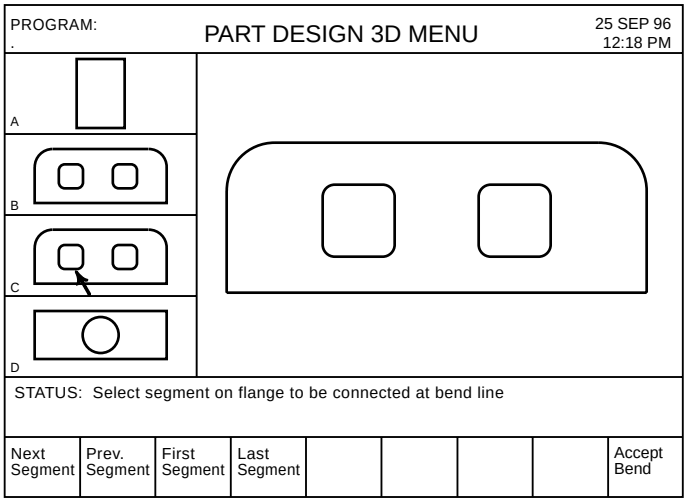
Once again the list of flanges appears on the left side of the screen. This time, however, the status line is prompting to select the bend line on the flange to be connected. Using the mouse, move the pointer to the

formed part and pick the long segment on Flange 'A' as the segment to be connected. The segment will be highlighted in a violet color as shown below.

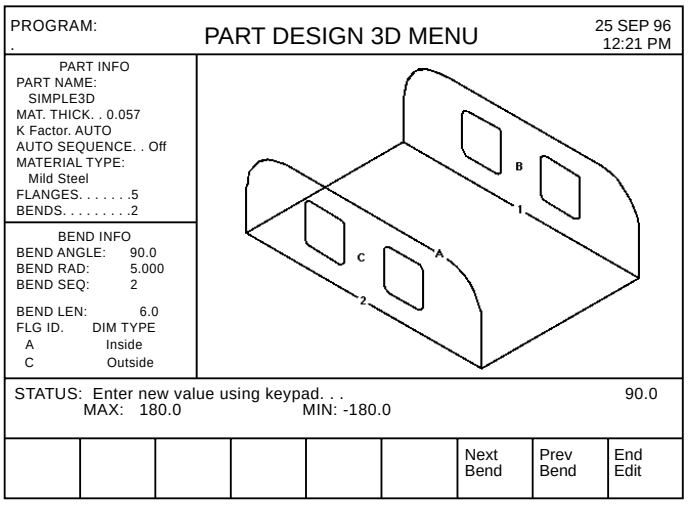


Select the Start Edge of the segment for the bend alignment. Now pick the flange to be attached from the list. For this example choose Flange 'C'.

The screen will change to show Flange 'C'. Use the "Segment" softkeys to pick the desired segment to connect as the bend line. For this example use the bottom segment.



Select the alignment for the bend connection as the End Edge. The part will now appear once again with the second bend complete.

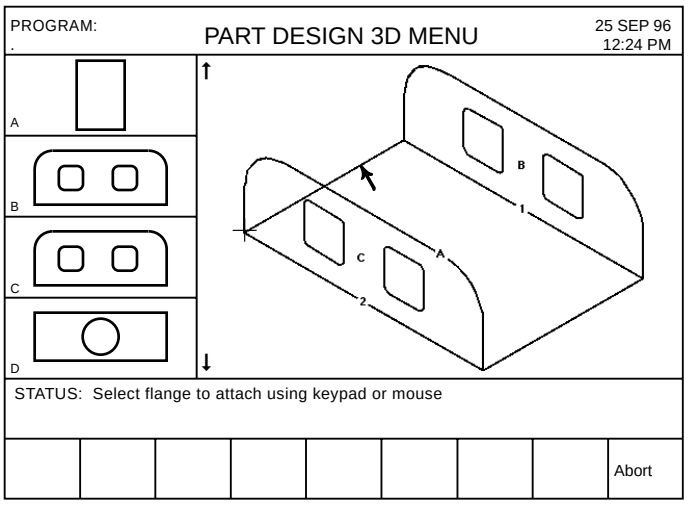


For the Bend information, the same values will be used because this is a symmetrical part. There is no need to change the values because they will default to the previous bend.

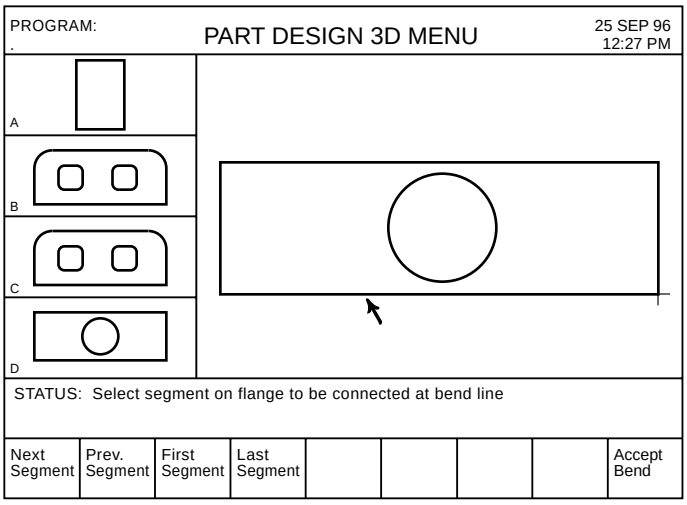
Note that at any time during this process the part may be viewed in the unfolded or flat view by pressing "End Edit", "Change Display View" and "Display Flat Pattern" softkeys.

Continuing with Adding Bend lines, press the "Add Bend Line" softkey and choose the top segment on Flange 'A' as the next segment to connect.

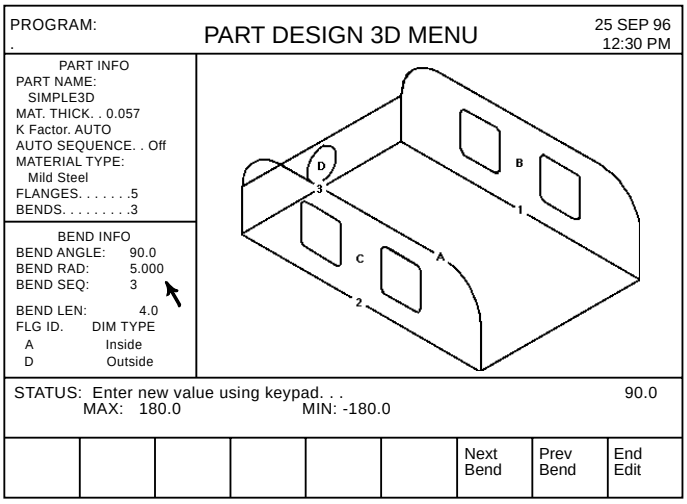
Pick the Start Edge for alignment.



Now Pick Flange 'D' from the list to be connected. Move the current segment to the bottom segment as shown.

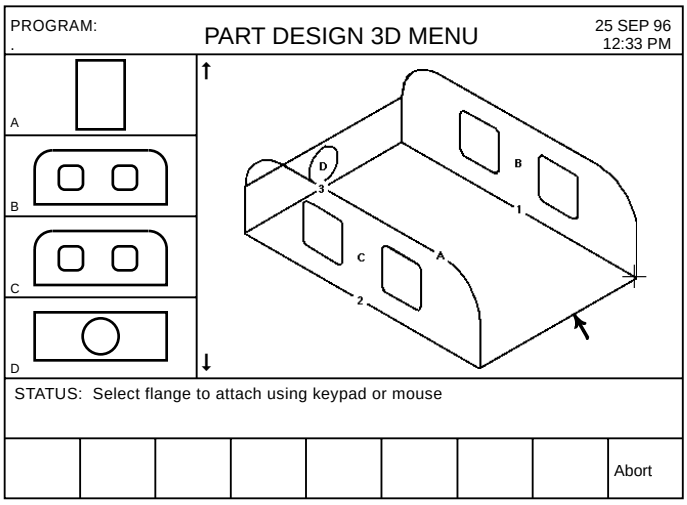


Press the “Accept Segment” softkey and then “End Edge” softkey. The part will now appear formed with the third bend.



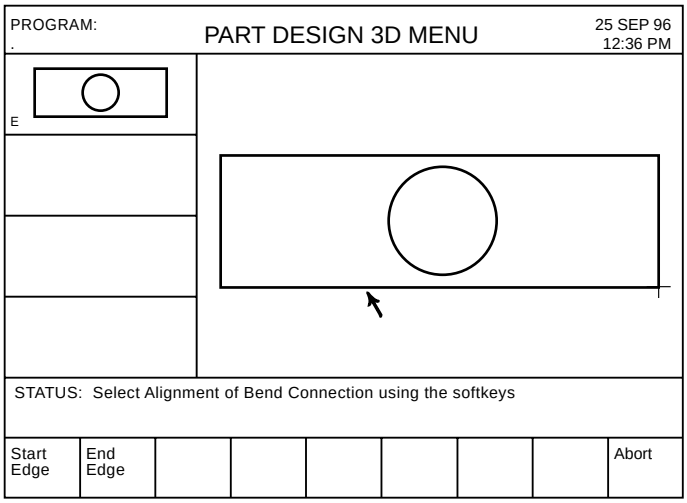
Again, for this bend, all of the BEND INFO can be left as default. Press the “Next Bend” softkey to add the last bend to the part.

Select the bottom segment on Flange ‘A’ and the End Edge as alignment. Note this is the opposite from what the previous flanges have been because the flange is now turned around on the opposite side of the part.



This time it will be necessary to scroll down through the flange list to locate Flange ‘E’, which is the last flange to be connected. Click the down arrow using the mouse and pick Flange ‘E’.

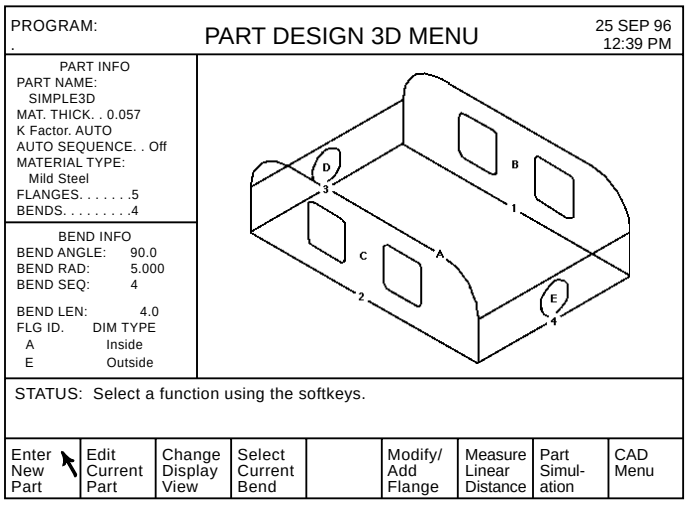
Flange ‘E” will now be drawn.



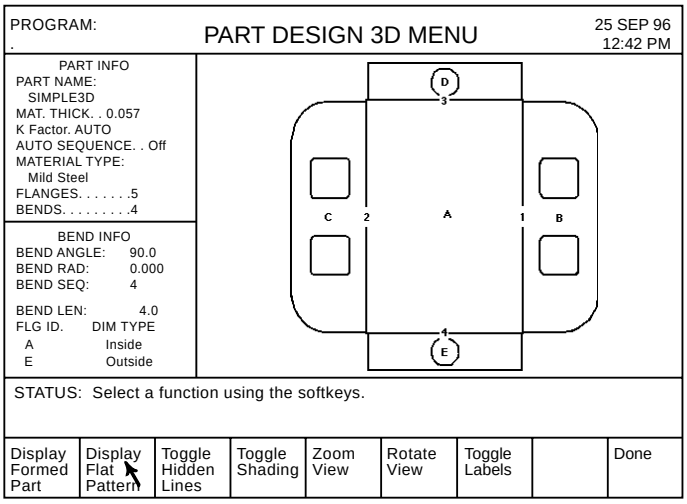
Use the softkeys to pick the bottom segment.

For the Alignment, choose the Start Edge. The BEND INFO is the same as the previous one.

This completes the part. It should now appear as shown. Press the “End Edit” softkey.



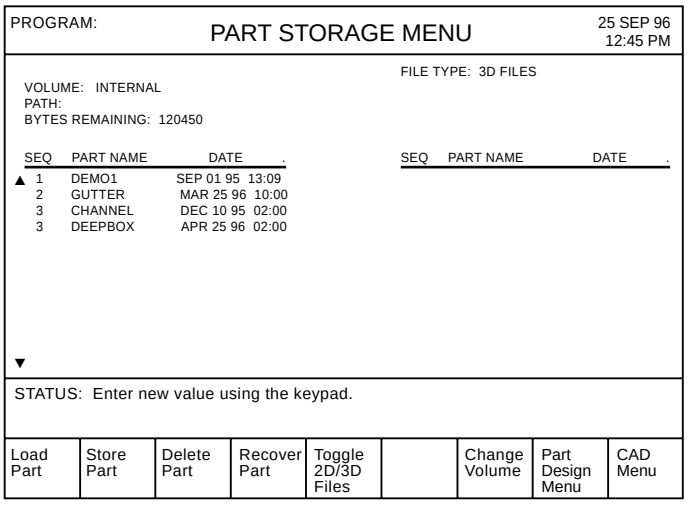
Note the PART INFO section of the screen shows that there are 5 Flanges and 4 Bends. Use the "Change Display View" softkey, then "Display Flat Pattern" to view the unfolded part. It should appear as shown.



This completes the procedure for designing a 3D part.

PART STORAGE MENU

The PART STORAGE MENU is used to load or store 2D and 3D CAD Part files.



SOFTKEY FUNCTIONS

The "Load Part" and "Store Part" softkeys correspond to either the 2D or 3D part according to the state of the "Toggle 2D/3D Files" softkey.

FILE TYPES

There is now a new file type with the 3D Graphics software called a 3D CAD Part. There are two types of CAD files referred to as 2D or 3D. The STORAGE screen will display a listing of both types of files. The FILE TYPE field at the top of the screen indicates the type. The "Toggle 2D/3D Files" softkey will alternate between the two filetypes.

When a 2D part is loaded from the storage volume, it will be transferred to the active memory and displayed in the PART DESIGN 2D MENU. When a 3D part is loaded from the storage volume, it will be transferred to the active memory and displayed in 3D PART DESIGN. The average size of most 3D CAD files will be 3-4k.

A part may be selected for loading by entering the sequence number, name, or by picking the part name from the directory using the mouse. More directory pages may be viewed by pressing the up and down field keys or by using the mouse to point and click on the up and down arrows on the left side of the screen.

After loading a part, the control will change screens automatically to the corresponding PART DESIGN screen.

Store Part: This softkey will store the current part (2D or 3D) to the volume specified. If a file already exists, the user will be prompted to confirm if it should be overwritten.

Delete Part: This softkey allows deletion of a part. A part may be deleted by selecting the name, sequence number or using the mouse to pick the

name from the directory listing. The user will be prompted to confirm the deletion with a YES response.

Recover Part: This softkey is only valid if the volume is set to INTERNAL. It will allow the user to undelete a part if the control has not been powered-off since the part was first deleted.

Change Volume: This softkey selects between internal memory storage and the local floppy drive. On the local floppy drive, 2D and 3D files are stored in different directories. The default directories are \CAD and \CAD3D.

Part Design Menu: This softkey will switch screens to either the PART DESIGN 2D or 3D screen.

Continuing with the example for the Part Design, the SIMPLE3D part may now be stored in Part Storage. Press the "Store Part" softkey. The directory will now be updated to include the new file.

INTERNAL STORAGE CAPACITY

Included with the 3D Graphics option is an increase from 96k to 192k of internal file storage. This can be used for all file types including programs and tools.

PROGRAM STORAGE

LOADING BEND PROGRAM AND CAD PART

The PROGRAM STORAGE screen is used to load and save bending programs to storage. When a program is loaded which was originally created using the Part Simulation, an additional prompt will be displayed.

STATUS: Do You wish to load the Matching CAD PART with Program?

This gives the user the opportunity to automatically load the CAD Part so the RUN MODE screen will show the graphical bend sequence. If the graphical display is not needed, the user may press "No" at this time.

If the YES response is made, an additional question will be asked on the type of Run Mode view to be used. See the discussion on the RUN MODE screen for the difference in the views.

Use the softkeys to select either the "Full Screen View" or the "Normal View" and then the view type which is either the 2D View, 3D Machine View or 3D Part View.

The CAD file will then be loaded. Pressing the "Run Mode" softkey will change screens to the RUN MODE and immediately display the part graphic.

AUTOMATIC LOADING / STORING FROM REMOTE PC USING COMMUNICATIONS

The PROGRAM STORAGE screen now allows a program to be loaded directly from the Remote Volume using the Kermit communications protocol to transfer the file. If the program was created using CAD, the corresponding 3D CAD Part will be loaded automatically. It is no longer necessary to request the Program and CAD files separately using the FILE TRANSFER screen.

The other advantage this offers is the ability to keep the master copy of all programs on the remote PC. When the operator needs to run a specific part, he requests it and it is loaded into the control quickly without having to change screens. Program files can then be deleted from the control at the end of the day or week.

Use the "Change Volume" softkey to set the Volume to Remote.

When "Load Program" is pressed, the status line will prompt for a filename in the usual manner.

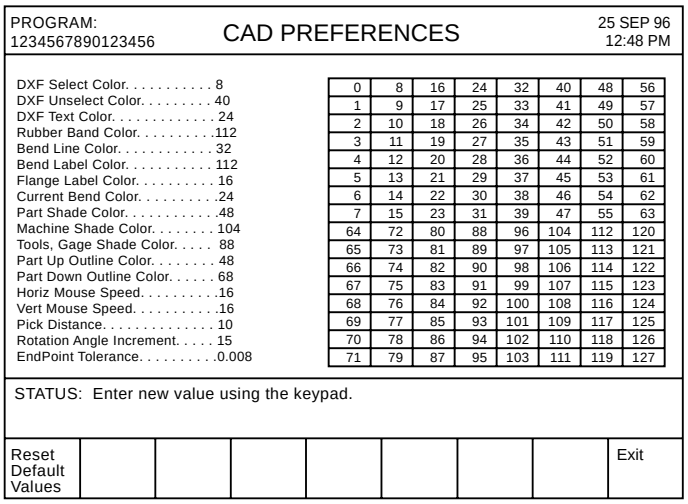
After pressing ENTER, the Machine Control will make a request to the remote computer attached to the machine through the communications interface. It will search the remote computer for a program with the requested name. If it finds the file, it will then proceed to transfer it automatically to the control and store it in internal memory. It then looks at the program file to determine if an associated CAD Part is included. If so, the CAD Part is searched for on the Remote Volume and it is also transferred to the Machine Control.

After the program and CAD Part are transferred, they are loaded into active memory and the screen will change automatically to the RUN MODE. The part may now be run.

When the "Store Program" softkey is pressed, the control will save the program currently in active memory to internal memory and the remote computer.

CAD PREFERENCES

This screen contains user selectable settings for the colors, mouse speed and other tolerances used in the IMPORT/DISPLAY DXF, CONVERT DXF PART DESIGN 3D and PART SIMULATION screens. Pressing the "Reset Default Values" will change all values to their original default settings.



COLOR SELECTION

For fields which require a color selection, use the mouse to pick the color value or enter the number from the table using the keypad. Certain fields, such as Part Shade Color, require the color selected to be the first one in the color shade group, i.e 0,8,16 ...112,120. A shade group consists of 8 shades of the color varying from Light to Dark. All colors must be in the range of 0 -127.

DXF Select Color: Color used in IMPORT/DISPLAY DXF and CONVERT screens to indicate line has been selected. Defaults to GREEN.

DXF UnSelect Color: Color used in IMPORT/DISPLAY DXF and CONVERT screens to indicate line has been un-selected. Defaults to BROWN.

DXF Text Color: Color used to display text in a DXF File. Defaults to RED.

Rubber Band Color: Color used when dragging a window for Zoom or Selection. Defaults to YELLOW.

Bend Line Color: Color used to identify Bend Lines on the Part. Defaults to MAGENTA.

Bend Label Color: Color used to draw text for Bend Labels. Defaults to YELLOW.

Flange Label Color: Color used to draw text for Flange Labels. Defaults to LIGHT BLUE.

Current Bend Color: Color used to identify current bend in PART DESIGN 3D and PART SIMULATION. Defaults to RED.

Part Shade Color: Shade group color when part is drawn. Defaults to GRAY.

Machine Shade Color: Shade group color when ram, bed and other machine components are drawn. Defaults to BEIGE.

Tools, Gage Shade Color: Shade group color for tooling and gage finger. Defaults to LIGHT RED.

Part Up Outline Color: Color used to draw the part outline when bend angles are positive. Defaults to WHITE.

Part Down Outline Color: Color used to draw the part outline when bend angels are negative. Defaults to BLUE.

Horiz Mouse Speed: Values which control the gain for the Left to Right movement of the mouse. Default value is 16. Range is slow to fast (0-999).

Vert Mouse Speed: Values which control the gain for the Up and Down movement of the mouse. Default value is 16. Range is slow to fast (0-999).

Pick Distance: Value representing the number of screen pixels away from an object the mouse cursor must be in order to select the object. Default value is 10. Range is 0 -99.

Rotation Angle Increment: Angle increment used when specifying the View rotation. Each time the rotate “CW” or “CCW” softkey is pressed, the part will be rotated by this increment. Default value is 15. Range 0-90.

End Point Tolerance: Value in inches used when converting a DXF file to determine how close end points have to be before they are considered connected. Adjust this value larger if drawings fail to convert because of unconnected lines in the original file. Adjust this value smaller if drawing scale is very small and some lines are incorrectly taken as connected. Default value is 0.008. Range is 0.00 to 0.100 inches.

APPENDIX A – DXF COMPATIBILITY

This section describes which features of the DXF format are supported. DXF is a proprietary format written by AutoDesk for exchanging CAD data between application programs and AutoCad®. The DXF format supports drawing entities from AutoCad®. Many other third parties have adopted the DXF format. It is important to note that many features from different CAD vendors will not always be supported in the DXF format. Complex entities such as splines and surfaces tend to cause problems between different systems using the DXF format. The design goal of the DXF translator for the AUTOFORM Graphics software is to utilize the most basic features of the DXF

format. This will ensure compatibility with most CAD systems using the DXF format. It may be necessary for the user to make changes to a drawing prior to exporting it to the AUTOFORM Control to convert unsupported drawing entities into more basic ones which are supported.

1. The following ENTITY types are supported:

   LINE, LINE3D - Lines defined with start and end coordinates either 2D or 3D
   CIRCLE - Defined with x, y, z center and radius
   ARC - Defined with center point, radius and start, end direction angle.
   TRACE, SOLID - A solid filled region defined by either 3 or 4 x, y, z points.

   INSERT - An insert is a macro-like call to a previously defined block of entities. All inserts will be exploded into their individual entities when the file is translated.

   POLYLINES - 2D or 3D Polylines consist of a connected sequence of line and arc segments treated as a single entity. Each point of the polyline is defined as a VERTEX. These can be connected by straight line segments or spline curves. A polyline may be closed or open.

   Note that polylines with only straight line segments will be supported.

   VERTEX - individual entities defining each point of a polyline entity.

   Note the following entities which are excluded. All DIMENSION type entities are excluded.

   ATTRIB - special entities which contain user defined text fields for database type functionality.
   SHAPE - collection of lines, arcs and circles used to construct special symbols.
   POINT - single x, y, z coordinate.

   Poly Face Meshes - Connected sequence of 3D faces defining a flat or approximated curved surface. These will not be supported.

2. DXF Files should be ASCII formatted, not binary. Exported DXF files will always be ASCII formatted DXF files as output.

3. Other entity properties use the following translation scheme for entities and layers.

   Entity Layer - Multiple layers are supported when reading DXF files. The user may extract and display individual layers from a DXF file. When writing DXF files, all entities will be placed in the same layer. The user may specify the desired layer. A maximum of 255 layers may be imported. The Layer names will be displayed on the import DXF screen.

   LineType - All dashed line types will be imported and displayed while converting the DXF File. Once converted all lines are displayed as continuous solid lines.

   Color - Entity colors are ignored. All geometry will be displayed in the colors specified in the CAD PREFERENCES for DXF Select and Unselect color. Text will be displayed using the DXF Text Color.

   Entity Handles - Unique identifiers associated with an entity. These will will not be used. When writing files all entity handles are set to NULL.

4. The following TABLES Entries, STYLE, VIEW, USER COORD SYSTEM and VIEWPORT, will not be used.

APPENDIX B

When using the PART SIMULATION screen on the AUTOFORM to automatically generate part programs, the GAGE ALLOW value is calculated automatically when a program is generated.

The operator enters the Part Bend Radius on the PART DESIGN MENU as well as each flange dimension type. The K-Factor field is also included on the PART DESIGN MENU to allow the option of entering the desired K-factor by the operator.

EXPLANATION OF FORMULAS USED

One formula used is the equation to calculate the Bend Allowance (BA) or arc length of the bend.

$$BA = [(K \times t) + r)] \times [2\pi \times (180 - ANG) / 360]$$

where:

t = Material Thickness
r = Inside Bend Radius
ANG = Bend Angle
K = % location of neutral axis of material = Neutral Axis / t
BA is the arc length of the bend.

This formula has been proven over the years to provide the best approximation of the length of the arc. However, it is dependent on how well the location of the neutral axis can be predicted. Currently we are using the following factors for the K value.

K-FORMING FACTOR CONSTANT

The K used in the equation is dependent on the selection made in PART DESIGN for the K-Forming Factor. If the AUTO selection is made, K will be as follows using the step values shown:

K = 0.40 for 14 Ga. or thicker
0.33 for thinner than 14 Ga.
0.50 when Bend Radius (r) is greater than or equal to 3 times (t)

If the FIXED selection is made, K will be equal to the value entered by the operator.

The definition of K-factor is the location of the neutral bending axis of the material. If the K-factor is 0.500, the neutral axis is located at the center of the material. A typical value will be in the range 0.30 - 0.50. However, the operator may enter a value in the range of 0.10 - 0.90.

The Bend Allowance formula has been found to provide a good approximation. However, it is dependent on the location of the neutral bending axis. The ability to change this factor has been included to allow the operator to approximate this value more accurately if they are knowledgeable about their material.

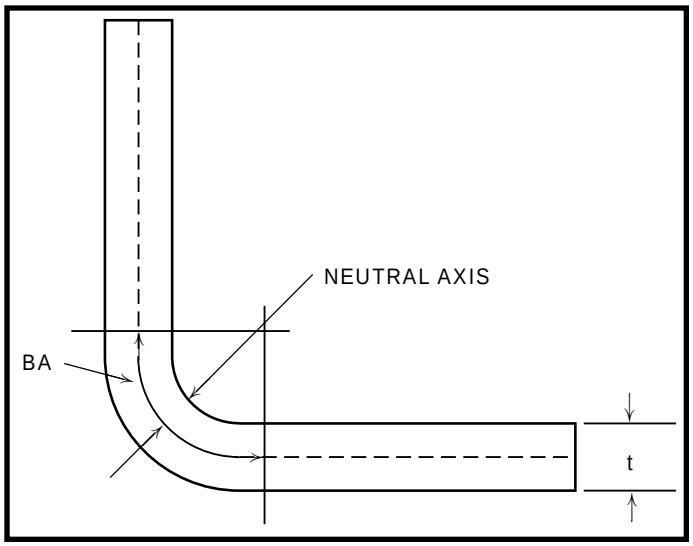


FIGURE 1 – Neutral bending axis of material

The PART DESIGN MENU will display these softkeys when editing Flange Dimension Type.

Inside	Outside	Radius	Vertex	Neutral				

Select the type of dimensioning used for the Flange Dimension. Note the diagrams shown below which indicate the proper way of defining the flange.

Specifying a NEUTRAL dimension type results in no bend allowance being calculated. This assumes the blank already includes it. The neutral dimension is used whenever a flat blank is imported as a DXF file.

The following shows a diagram of a single 90° bend indicating three methods of specifying a flange dimension (Outside or Vertex, Inside, Radius). For the case of a 90° angle all of the dimensions can be converted by the following formulas.

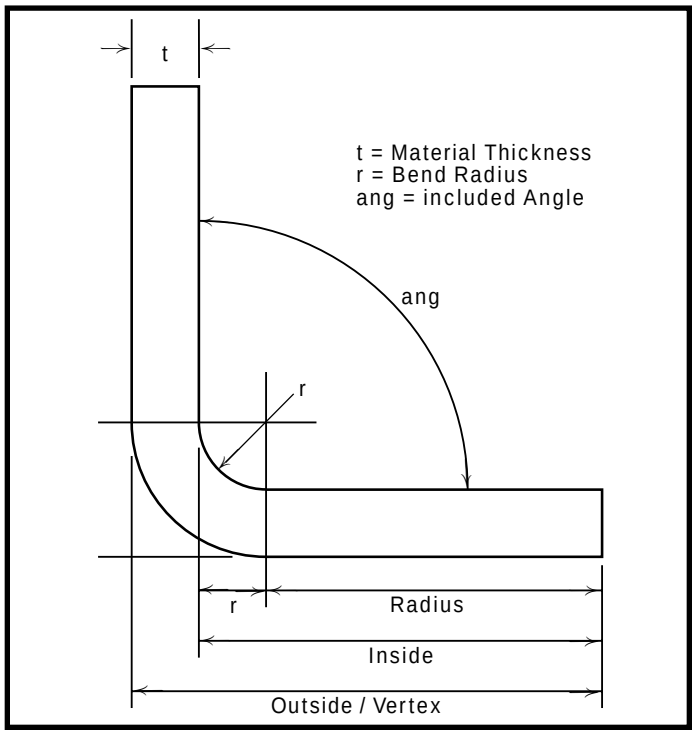


FIGURE 2 – Three methods of specifying a flange

Note: *For 90° angles Outside and Vertex dimensions are equal.*

Radius Dimension = Inside Dimension - r
Radius Dimension = Outside Dimension - (r + t)

Using the equations above the final Gage Allowance is calculated as shown below.

GA (Outside) = 1/2 BA - (r + t)
GA (Inside) = 1/2 BA - r
GA (Radius) = 1/2 BA

Typically when and Outside or Vertex dimension has been specified, the GA value will be negative. When a Radius dimension has been specified, the GA will be positive. For an Inside dimension, it can be either positive or negative.

It is important when specifying the type of dimension that it matches those indicated in the above drawing. The above formulas apply only to angles equal to 90°. Shown in Figure 3 is a diagram of an acute angle where different formulas apply.

Radius Dimension = Inside Dimension - r
Radius Dimension = Outside Dimension - (r + t)
Radius Dimension = Vertex Dimension - TAN
((180 - ANG)/2) * (r + t)

Using the equations above, the final Gage Allowance is calculated as shown:

GA (Outside) = 1/2 BA - (r + t)
GA (Vertex) = 1/2 BA - TAN ((180 - ANG)/2) * (r + t)
GA (Inside) = 1/2 BA - r
GA (Radius) = 1/2 BA

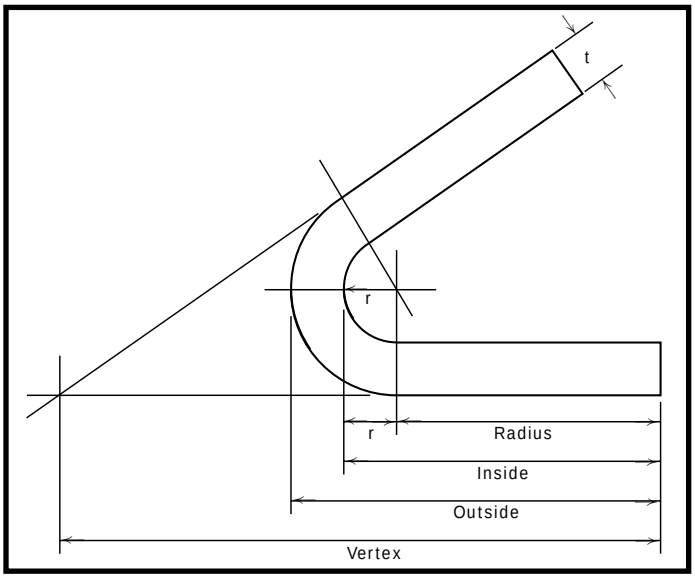


FIGURE 3 – Acute angle with different formulas

Typically when an Outside or Vertex dimension has been specified, the GA value will be negative. When a Radius dimension has been specified, the GA will be positive. For an Inside dimension, it can be either positive or negative.

It is important when specifying the type of dimension that it matches those indicated in the above drawing. When the included angle is greater than 90° the dimensions shown above become much harder to specify as shown in Figure 4 for an obtuse angle.

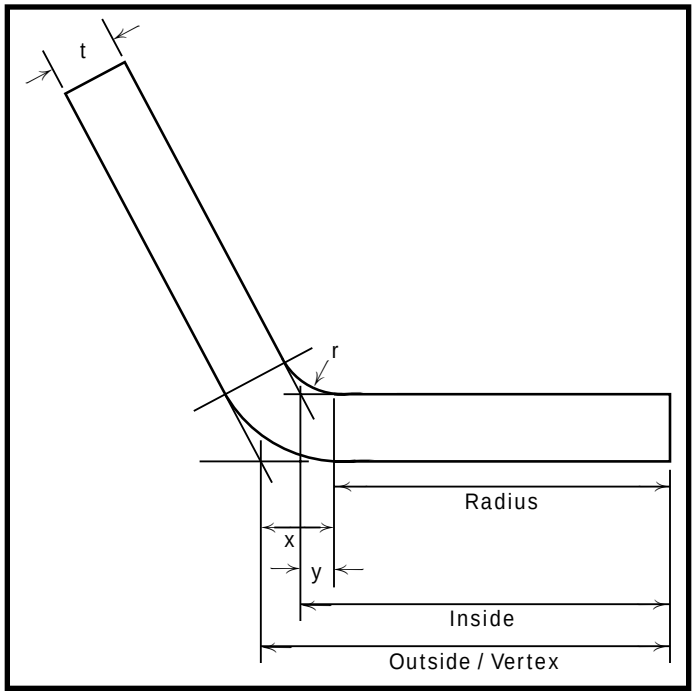


FIGURE 4 – Obtuse angle

Note that the dimensions labeled "x" and "y" are measured from the radius to the theoretical intersection point of the two flanges and is no longer equal to the bend radius. Thus, the required formulas are shown below for angles greater than 90°.

The Gage Position, to achieve the desired flange, is found by adding half of the BA value to the Radius Dimension. Therefore, the following formulas define the Gage Allowance (GA) value shown in the GAGE DATA MENU on the AUTOFORM for the dimension types.

GA (Outside) = 1/2 BA - [TAN ((180 - ang)/2) * (r + t)]
GA (Inside) = 1/2 BA - [TAN ((180 - ang)/2) * r]
GA (Radius) = 1/2 BA

CINCINNATI

CINCINNATI INCORPORATED