



FastCAM Software Series

FastSHAPES[®] Reference Manual

The World Standard in Profiling Software

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Conventions used in this Manual

Throughout this manual information that is important to your understanding of FastSHAPES® is highlighted in certain ways. The items that are emphasized in this way include:

- menu items
- keys
- mouse buttons
- screen messages
- important notes.

Representing keys

When you need to type a specific key, or click a particular button, the instruction is shown as:

ENTER

This means press the Enter button.

Representing lists of instructions

Lists of step by step instruction have been detailed with the use of numbered lists. To complete the task you simply follow the instructions in sequential order. For example:

1. Run the FastSHAPES® program by “double clicking” the FastSHAPES icon on your Windows desktop.
2. From the FastSHAPES® library, “double click” on the CONE icon.
3. Select the JOB data form at the top, by clicking on it.

4. Click in the “Job Number” field and type in “1”, and then press the **TAB** key.

Representing important notes

A writing symbol is used to draw your attention to important notes in the text.

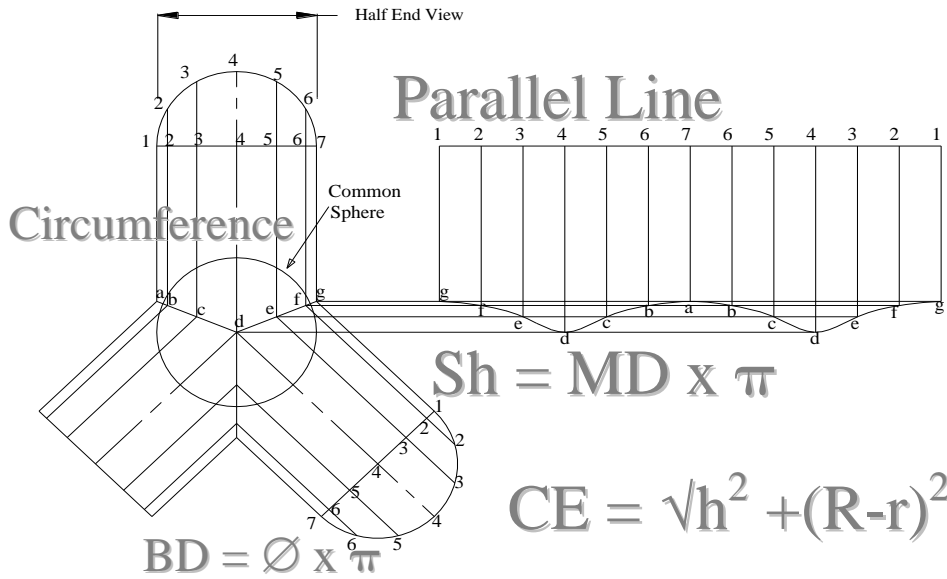
For example:



NOTE: The appearance of this screen will vary depending on the amount of FastSHAPES licensed.

CHAPTER

1



FastSHAPE'S Introduction

FastSHAPES® is a unique, True Thick Plate Development software suite, designed for the steel fabrication and associated industries. Its main feature is that it is extremely simple to operate, which completely eliminates the manual marking out of patterns and templates.

FastSHAPES® is for TRUE thick plate development **NOT** just simple sheetmetal patterns. It outputs, full manufacturing details for all shape developments:

- bend lines,
- bend angle,
- longitudinal seam offsets,
- plus “green” allowance for forming.

The available output file formats are:

- DXF FastSHAPES aim to supplement rather than replace CAD programs.
- CAM Patterns can be edited using the sister product FastCAM
- NC Requires FastCAM which supports all popular Numeric Controllers for pattern cutting.
- PRN Printable data tables for manual lofting.

Graphical output includes, full 3D viewing of developed shapes with the ability to view even individual components within a construction.

Several FastSHAPES® products can be used in conjunction with other Windows products such as Excel® requiring spreadsheet data entry.

FastSHAPES® will create all common developments including;

Transitions:

Rectangle to Round including: Oval, Frogmouth and circular transitions with multi-height variable strokes, off-set seams and green allowances.

Lobsterbacks:

Including plain, tapered and transforming gored bends with automatic nesting of shapes and back to back “stitch” cutting if required.

Cones:

Including ultra low (tank bottoms) and ultra high angles (flagpoles).

CONE allows for multi-strake variable thickness with mean, outside or inside developments.

Right and Offset cones are superbly catered for including plate thickness and optimized seam locations. The cone suite can also produce all manner of conical truncations.

Pipe branches:

All types including “Set-On and Set-In” developments and offset branches. P Branch also produces outside wrap templates for cutting standard pipe to form Tee’s. The structural pipe branch program also includes the compound weld preparation for heavy wall pipe intersection. This data can be used for output to complex beveling type automated pipe cutters.

Power transmission:

FastSHAPES® includes a full range of sprockets, gears and rack and pinions for mining and heavy industry applications.

Features that distinguish FastSHAPES are

- support for all the output formats listed above, NC in particular
- ability to handle **thick plate**, hence FastSHAPES is ***not*** just the traditional sheet metal pattern-maker.
- linkable with FastCAM for automatic nesting and pathing of parts, and transmission of full plate nests to production equipment via DNC connection when available
- ability to handle bevel cutting of thick plate via FastCAM NC output.

Bevel cutting support is NOT offered in the Standard Edition, being very dependent on equipment installed by the user. A number of bevel cutting systems have been successfully implemented in Special Editions of FastSHAPES / FastCAM, and enquiry’s are welcomed.

The Standard Edition DOES contain output in some area that facilitates bevel cutting using secondary processes after the programmed perpendicular cut has been made.

The principal area of application of FastSHAPES is medium (1/8” to 3/8”, or 3 to 10mm) to heavy (1 / 2” or 12mm and over) plate metals manufacturing in which the main jointing technology is welding. Applicability to light sheet metal manufacturing is incidental, and available in some FastSHAPES by coincidence rather than by design. (The medium / heavy classification and demarcation is possibly subjective, depending on specific applications. It is used here as given for general convenience and reference.)

Getting Started

Your Security Key

Your Security key (FastLOC) is your company's software protection device. Your *FastSHAPE* software programs cannot be accessed without this device being fitted. Your Security key must be plugged into a Parallel port on the rear of your PC. If your system hardware also includes a printer, it is necessary to place your Security key into LPT1 and attach your printer cable to the open side of the key.



If a printer is attached to your Security key, the Printer MUST be on-line to make the key operational.

If you already have another Security key attached to your computer, then simply attach the FastSHAPES key to the end of this one.

Running the *FastSHAPES* Program

1. Double-Click the *FastSHAPES* Icon on the Windows Desktop, with your mouse, **OR**
2. Press the Start button, located on the Windows Taskbar; proceed to the Programs Menu, and then across to the *FastSHAPES* Menu, where you select the *FastSHAPES* program Icon.



The FastSHAPES program will need to be installed from the CD ROM before it can be started.

Language Selection

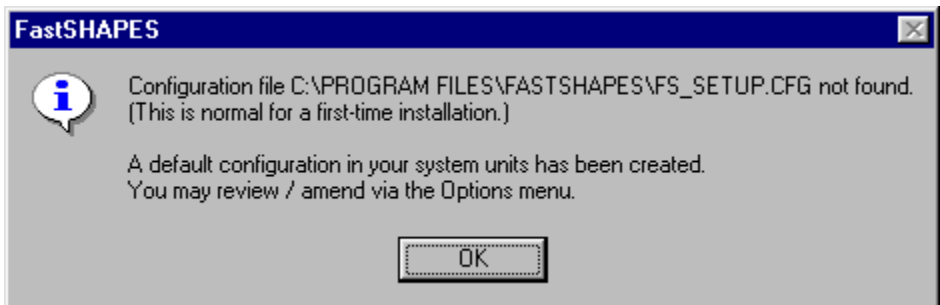
FastSHAPES is a Multi-lingual program. It currently has the ability to run in the following languages:

- English
- Chinese &
- Polish

Simply select the desired language from the Language menu found at the top of the program

Opening Screen

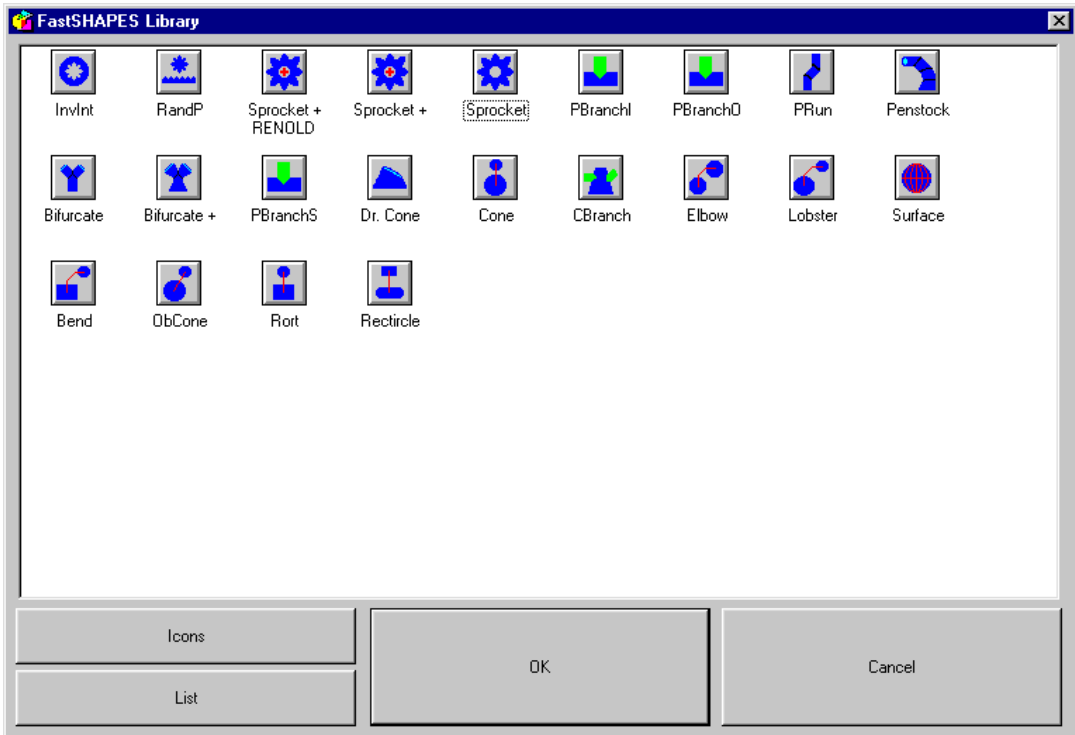
The initial screen of FastSHAPES will present the user with the following message:



This message highlights the fact that the user needs to set the configuration for the software. These parameters include everything, from: Screen colors to what Output is needed. Please refer to the **Configuration** section for more information.

Selecting a FastSHAPE

To select a FastSHAPE, simply "double click" on the desired shape icon, in the FastSHAPES Library screen.



NOTE: The appearance of this screen will vary depending on the amount of FastSHAPES licensed. Some may appear ghosted (shadowed/grayed-out), which means you currently do not have a license to run these particular Shapes. Contact your Reseller to purchase additional individual Shapes.

For a more detailed view of the Shape Library screen, simply hit the LIST button to see a detailed explanation of each particular Shape, which lists:

- Shape Name

- Shape Class, e.g. – Right Circular conic Surfaces
- Development Method, e.g. – Radial Line
- Shape Description

Once a FastSHAPE has been chosen the main screen for that Shape should appear.

Each Shape's main screen is split up into 4 "tabbed" areas. Those being:

- Shape Description of the particular FastSHAPE.
- Job Descriptive area for relevant Job details.
- Data Input screen for all relevant Shape Data.
- Parts ListGenerated list of all segments, including Area, Mass, Perimeter...etc.

Inputting Data

Once a FastSHAPE has been chosen, the user can begin inputting data, via the Job and Data tab at the top of the FastSHAPE form.

The data on the Job tab, is not mandatory, but does aid in the referencing to any retrieved data at a later date.



NOTE: The Job/Drawing & Assembly number will also come up as a default option in the File Name on the Output form.

The data on the Data tab, is where all the Shape dimensions are input.

The mandatory fields will fields will be highlighted in a "yellow" color.

“Red” fields indicate that the data entered cannot be correctly interpreted e.g. Letters input where a number is required.

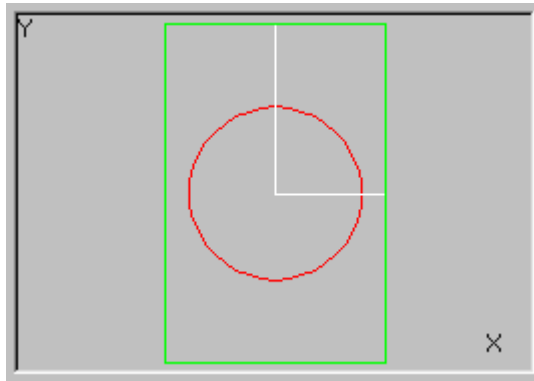


*NOTE: Refer to the Section on **Units** for details on how to “enter” different types of data, e.g. Inch, fractional...etc.*

To move from field to field you can either use the TAB key or use the mouse to click into each field.

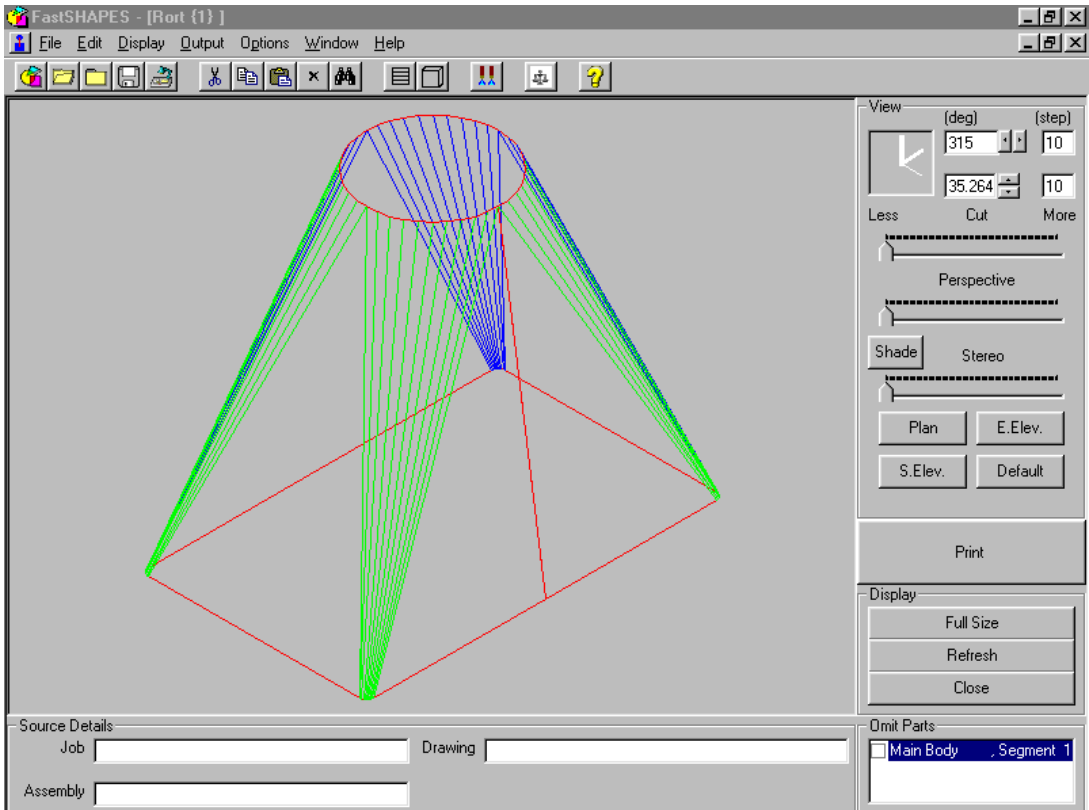
As you begin to input data, you may notice the Display Window begin to show a PLAN View of the Shape.

(Example of the Display Window)



Generating the Shape

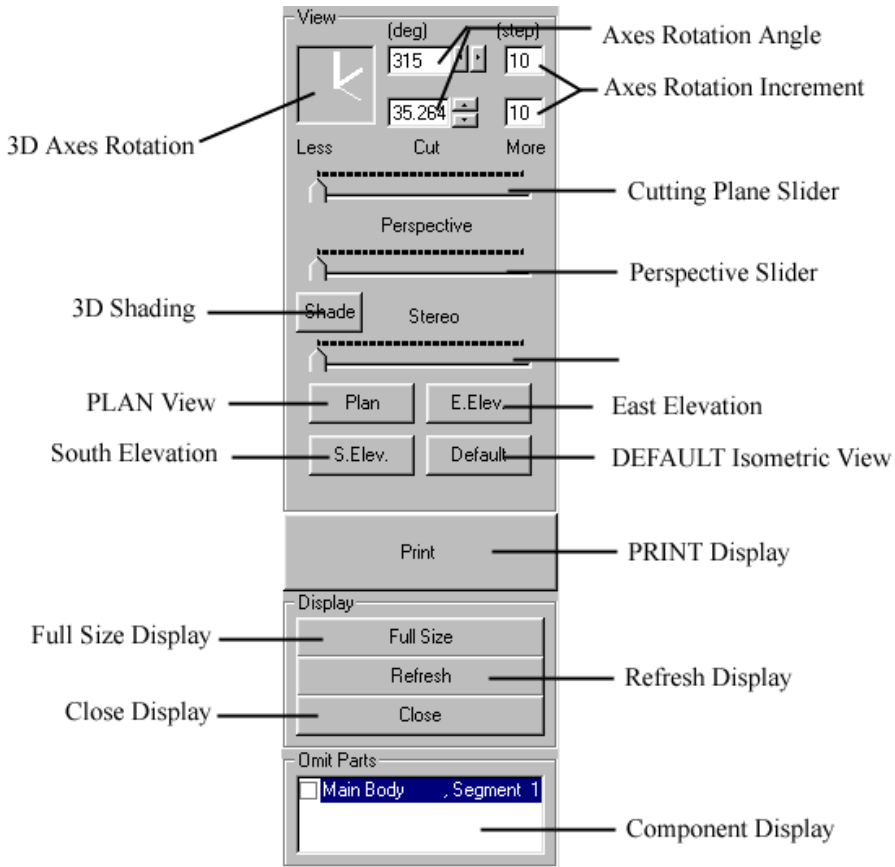
To generate the Shape, (once all the appropriate data has been input) simply "Left click" in the display window, and you will be taken to the Shape Display window. An isometric Wire-frame display of the Shape should now be visible. (*see below*)



While on this display form, you can do many things, such as:

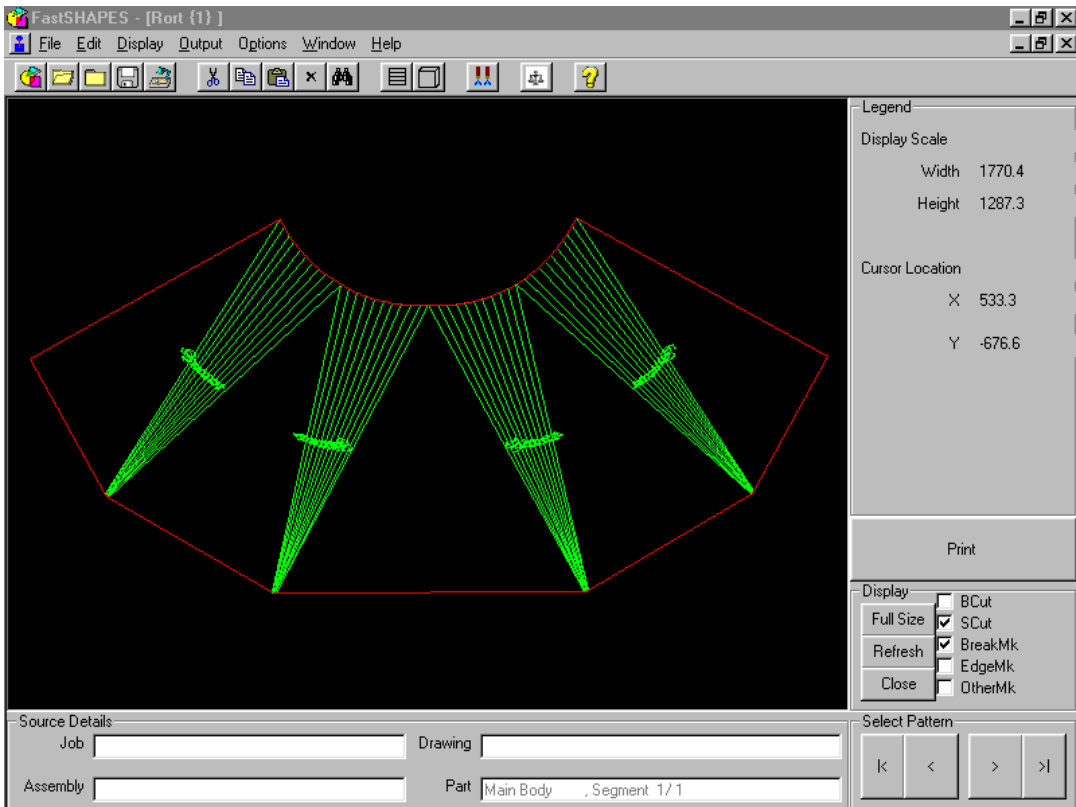
- Rotating the displayed shape,
- Viewing the shape from different views,
- 3D Shade the shape,
- Zoom in to an area of the shape,
- Slice into the shape,
- View separate components of the shape.....etc

See the descriptive list below, for details on each Display operation:



If you "Right click" on the display window (on the DATA tab), you will see the pattern(s). You could also "Right click" on the CLOSE button (on the DISPLAY form), to toggle between the Wire-frame view and the Pattern view. (*See Below*)

Pattern Display:



Output

There are several options of Output from FastSHAPES, as listed in the Introduction.

To Output, you **MUST** first develop the Shape (as detailed in Generating the Shape), and then either select the OUTPUT menu item or the OUTPUT icon on the Toolbar. Once selected you will be presented with the following form:

Output

Job Number : AS472-R

Drawing Number : AS472

Assembly Number : 6SQ Quantity Required : 1

Make .FSD file (Data) for Assembly
 Make .DXF file (3D-CAD) for Assembly

1 Main Body	, Segment 1/ 2
2 Main Body	, Segment 2/ 2

Output Selected Pattern(s) as

.CAM FastCAM file(s)
 .NC file(s) via FastCAM now
 .DXF (2D-CAD) file(s)
 .PRN tabulation file(s) Print now

to Drive ... c: to Directory ... C:\PROGRAM FILES\FastSHAPES>Data

to File AS472-R Sq to Round_nnn.ext

Ask before Overwrite

Output Now Cancel



NOTE: The appearance of this form may change slightly depending on your input data & configuration.

You can check/uncheck any items as necessary.

It is important however to always save the "FSD" file if you wish to retrieve the information at a later date.



NOTE: If the “.NC files via FastCAM now” is checked, then the FastCAM program will execute and automatically path the generated part files. The FastCAM program will close after pathing is complete.

You can also select/de-select any development parts as necessary, simply by clicking on the parts you desire to keep. By holding the CONTROL key down while clicking, you can select multiple parts.

Type the name of the file in the "to file..." box and then select the correct directory & drive for Output.

Click on the "Output Now" button and the file(s) will be saved.



NOTE : If a printed tabulation is required for manual mark-out then simply select the "PRN" file and "PRINT NOW".

FastSHAPES also has an AutoFileName facility, which is switchable in the Configuration options.

The AutoFileName format is as follows:

FS_AAA_001.*

- **FS** represents **FastSHAPES**,
- **AAA** is the incrementing portion of the filename, e.g. AAB, AAC....etc
- **001** is the number of the pattern which has been output, e.g. 5 patterns, 5 numbers 001, 002, 003, 004 & 005.

Methods of Development

Each FastSHAPE uses one of the following methods of development. Unless specifically noted to the contrary for a particular FastSHAPE, *only* the one method is used.

1. Direct

Used when the assembly or part is specified at source in flat (2-dimensional) form. Typically used for such mechanical parts as sprockets and gears.

Accuracy or exactness of the pattern depends on the ability or otherwise to replicate elements of the specified shape. This issue is covered specifically in the reference to each applicable FastSHAPE.

2. Radial Line

Used when a 3-dimensional part or assembly; is unequivocally defined by right-circular cones.

The right-circular cone is a surface of revolution, which can be thought of as the path swept by a straight line being rotated through space around an axis. The straight line is said to be a surface generator, and forms the basis of the development method. To the extent that all generators in a conical surface can be extended to a single point at the apex of the cone, they ‘radiate’ from the apex and give rise to the name of the method.

For the sake of brevity in general discussion, conic surface generators are referred to as ‘**breaklines**’ as defined for triangulated surfaces. Strictly speaking they are not actual breaklines, although the distinction is of more academic than practical interest.

Right-circular cones also have a concise and precise mathematical definition, hence their intersections with like elements and with planar surfaces can be solved in a precise manner. The locations, directions and

lengths of the surface generators can be calculated directly and exactly in both the 3 dimensional assembly and its 2 dimensional development. The method is hence described as being ‘exact’

It is possible for error to be introduced if the radius of curvature of the cone is less than about 5 thickness'. This range is not a seriously practical one for manufacture and use of right circular cones and is not specifically considered in the radial line method. Error from this source is far more significant in assemblies of the type that can practically be developed only by use of triangulation.

The 3-dimensional DXF files, which can be output define the assembly on the surface on which the 3-dimensional setting out data are given. This is usually the interior surface of the assembly. If in doubt and the validity of the 3-dimensional DXF data is critical, a second run of the data with zero thickness for all parts will put the issue beyond any doubt.

3. Triangulation

Used when a 3-dimensional part or assembly cannot be defined unequivocally by right-circular cones.

This method is inherently approximate. The degree of ‘exactness’ (or lack thereof) is normally dependent on:

- (1) the location of setting out points relative to the material thickness and
- (2) their spacing around joints in the assembly.

‘Setting out points’ are normally located at mid-thickness of the material. It is well known that this is incorrect in the case of tightly pressed bends having a radius of curvature less than about 5 thickness'.

For example:

The nominal developed length of a 90 degree bend of internal radius R , based on mid thickness, is $1.57*(R+t/2)$, or about $2.4t$ if $R=t$.

In practice, for a bend with $R=t$, the developed length is more nearly $2t$, and the neutral axis in the bend is at about $1/3$ of the thickness, not at mid thickness. This is about the tightest bend that can practically be made in the most ductile of materials.



Minimum bend radii recommended in material standards and specification range from 1 to about 3 or more thickness', depending on ductility and press direction relative to plate rolling direction.

Boilermakers have traditionally made allowance for this effect by rule-of-thumb methods such as ‘developed length = sum of internal sizes’.

To see how this works; imagine a rectangular frame made up by pressing the four corners tightly to an internal radius $R=t$, (t is the thickness); and the straight portions of the sides are A and B .

By the midplane calculation, the developed length would be:

- $2*(A+B) + 2*\pi*(t+t/2) = 2A+2B+9.4t.$

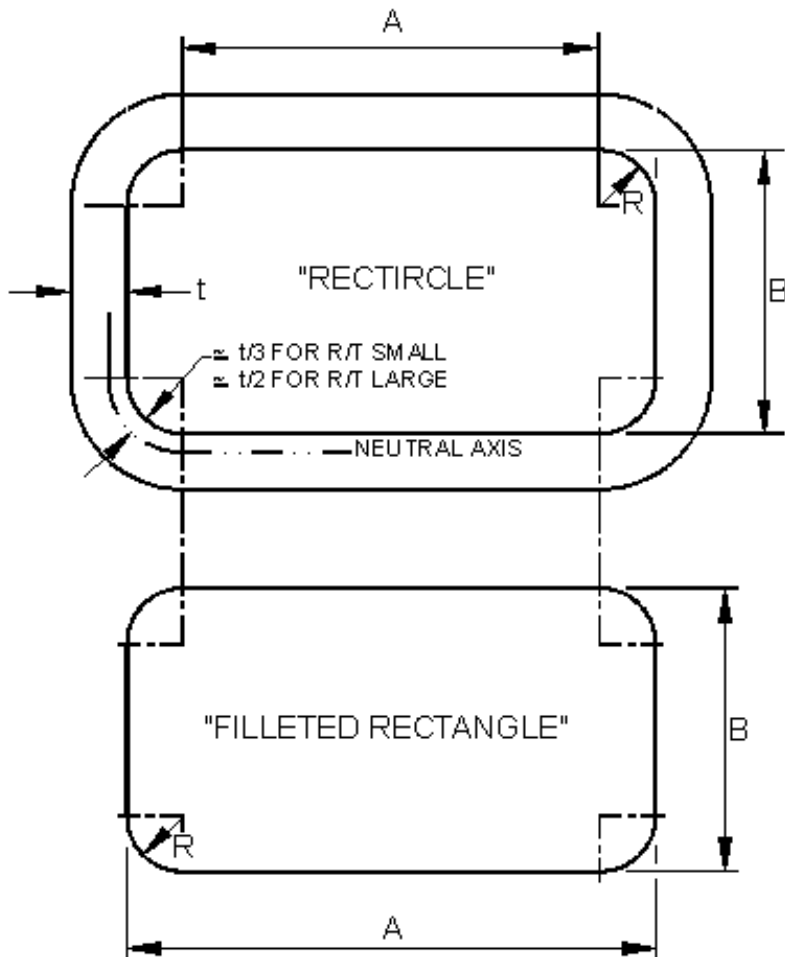
By the more correct third-of-thickness-plane calculation, the developed length would be:

- $2*(A+B) + 2*\pi*(t+t/3) = 2A+2B+8.4t.$

The boilermakers rule-of-thumb would give:

- $2*(A+2*t) + 2*(B + 2*t) = 2A + 2B + 8t.$

A number of FastSHAPES employ a device described as a ‘rectircle’ to facilitate the description of a wide range of cross-sectional shapes, which might or might not have tightly pressed corners. The rectircle is also defined by the internal sizes of a finite thickness assembly.



'RECTIRCLE', AND PRESSED CORNERS



Some FastSHAPES provide for an alternative formulation of a rectircle, described as a ‘filleted rectangle’, simply as a data entry convenience.

Warning: the user, should not confuse what in reality are two quite separate and distinct issues. FastSHAPES, will make its own corrections when a tightly pressed bend has been specified. These corrections are conjointly made with other compensation as described below. The user should not attempt to ‘doctor’ the input data describing the rectircle.

It is left to the user to ensure that where tightly pressed bends are specified, the radius used complies with material specification recommendation for minimum bend radius.

The second source of error, the setting out points spacing, is normally more significant and arises from the disparity between the actual length of a curve, and the length approximated by the sum of a series of chord lengths. Traditional lofting methods calculate or measure ‘true lengths’ as straight lines between setting out points, with the ‘true’ referring to the length being measured from a ‘true’ or ‘in-plane’ view of the line. The error has a two-fold effect on the pattern. First, the length of the pattern along edges representing curves will be less than that required, and second, the shape of the pattern will be distorted; non-parallel ‘breaklines’ or ‘bendlines’ will not have correct relative angular orientation.

One way of reducing the effect of this error is to use more numerous and closely spaced setting out points.

The error can also be compensated in other ways (as used by FastSHAPES) to ensure that even when relatively widely spaced, the ‘breaklines’ that appear on the pattern are correctly positioned and oriented relative to each other. Traditional lofting methods applied to cylinders may replace chord lengths with calculated arc lengths in order to correctly represent the developed girth. The same technique can be applied to a right circular cone, which can also be developed by

triangulation, although this becomes a little more difficult; it is necessary to calculate the angles between generators from the arc length and side length radius. Any assembly more complex than a right circular cone is generally just too difficult to bother with correcting in manual lofting.

The FastSHAPE ‘Dr.Cone’ provides both qualitative and quantitative comparisons of uncompensated triangulation with radial line method applied to a right circular cone which demonstrates very clearly the error effect of traditional lofting methods (uncompensated triangulation). In this particular instance, compensated triangulation as implemented by FastSHAPES and radial line development are equivalent and produce identical results, but this is not the case in general for other FastSHAPES.

FastSHAPES may apply either or both compensation types (thickness and spacing), and for simplicity are described as being developed by compensated triangulation. Other FastSHAPES apply neither and are described as being developed by uncompensated triangulation. (The Radial Line method discussed in the preceding Section does not require compensation.)

When used, compensation is applied by FastSHAPES to the 3 dimensional model from which the patterns are developed, leading to the following situations

	Pattern (2D)	Assembly model (3D)
Compensated Triangulation	‘Exact’	Inexact
Uncompensated Triangulation	Inexact	‘Exact’ (qualified)

FastSHAPES using compensated triangulation are certain transformers (‘Rectircle’, ‘Rort’, ‘Obcone’) and certain bends (‘Bend’, ‘Lobster’, ‘Elbow’). In these cases, the degree of ‘exactness’ arising from the compensation is dependent on the extent of deviation of curved surface elements from right circular conics.

The compensation applied to the 3 dimensional model is applied to those actual or notional elements of the assembly described as Collars.

A Collar is a prismatic section of ducting at either end of the assembly. The body elements of the transformer or bend are proportioned to fit between the collars, whether or not collars are specified. Collars are discussed in further detail elsewhere.

With the parametric form of data definition, it is possible to define surfaces for these FastSHAPES that *could* be developed with radial line method. (An alternate FastSHAPE using the radial line method is generally available in such instances). Despite this, the above listed FastSHAPES still use compensated triangulation. These possibilities provide test cases for the exactness of the compensation.

For example, a right circular cone or cylinder can be defined using FastSHAPE 'Obcone' by specifying no offsets, and its patterns can be compared with simple and precise calculations for a right circular cone or cylinder. At zero thickness, they will be found to be precise. (For finite thickness, bear in mind that the specified sizes apply to the collars to which the cone would be fitted)

Application of increasing offset values to obtain increasing obliquity of the cone introduces increasing imperfection in the compensation. (For the mathematically inclined reader, testing of the compensation algorithm involves use of elliptic integrals derived from a process equivalent to uncompensated triangulation of the oblique cone using an infinite number of development intervals. A cut square to the axis of an oblique cone defines an ellipse.)

For all above-listed FastSHAPES, user-defined surfaces that are reasonably practical in terms of manufacture have a target error in pattern setting out point location less than about 1 in 10000, or .01%, compared with common tolerancing requirements of the order of 1 in 1000 or 0.1%

FastSHAPES using uncompensated triangulation are 'Surface' only. The degree of exactness of both the patterns and the 3 dimensional assembly model are in the hands of the user – see the specific Section on 'Surface'. The exactness of the 3 dimensional model can be retained or obscured (as distinct from lost), at the users option.

Fairing and Splining

Direct development patterns are not faired unless noted specifically for the applicable FastSHAPE.

The feature common of patterns produced by both radial line and compensated triangulation methods is that, while the ends of the radial lines or breaklines are correctly positioned as points along a smooth curve (circumferential seam), some secondary treatment is required to generate those smooth curves, otherwise they can only be connected with straight lines.

In the case of FastSHAPE ‘Surface’ using uncompensated triangulation, facility is provided for the user to optionally spline through the specified setting out points prior to development. The exactness or otherwise of the development is thus in the hands of the user. Nevertheless, the patterns retain the common feature described above.

FastSHAPES provide for producing faired curves on circumferential seams, at the users option, and to a user-configurable tolerance.

Faired curves contain only straight lines and / or circular arcs up to a maximum radius. All the pattern setting out points as developed are guaranteed to lie within the user-specified tolerance of the faired curve.

Green and Longitudinal Seam Location

Most FastSHAPES permit the application of ‘green’ to certain edges of the pattern, usually to the longitudinal seams (‘End Green’), and in some cases to the circumferential seams (‘Edge Green’), of patterns for parts of 3-dimensional assemblies.

End Green on longitudinal seams may serve two purposes. First, it may be required to facilitate the forming process. Second, it can be used as a ‘margin for error’ to offset anticipated error of any sort or source, or to otherwise facilitate the joining process.

Edge Green on circumferential seams is not generally required, but may be applied for example when some damage to the edge may be anticipated as when rolling a cone using cone-rolling facilities with pyramid rolls.

With patterns developed by the radial line method, End Green can be obtained by adding a few degrees to the axial sweep angle before or after the longitudinal seam position, so that the circumferential edges of the pattern in the End Green area remain in their correct location. That is, the green can be applied effectively in the 3-dimensional setout and can be ‘developed into’ the pattern.

Similarly, it is possible with radial line patterns to locate longitudinal seams at literally any point in a 0-360 degree range of development sweep angles. Most but not necessarily all FastSHAPES using the radial line method permit seam location in this manner. If not, the method described below for the triangulation case is provided.



NOTE: A seam location dialog is available with each 3-dimensional FastSHAPE

Patterns developed by triangulation are not so readily managed, and green is added to the pattern after development. Because the extension of the pattern off the longitudinal seam end is not defined after development, the circumferential seam edges cannot be simply extrapolated without running the risk of cutting off some required part of the pattern. Therefore, the corner angle of the pattern is

bisected and the green material taken out along the bisector. This created a ‘fishtail’ appearance on many patterns.

Longitudinal seam locations for FastSHAPES developed by triangulation (and for some developed by radial line method) are predefined. These are as indicated by a dialog accessed through a command button for the relevant shape. Up to eight locations may be specified, depending on size of the assembly and any requirements to offset longitudinal seams to avoid cruciform joints.

Impact on Manufacturing

Parts developed using the radial line method are generally capable of being formed in pyramid rolls, although they may alternatively be pressed. The radial lines (surface generators) loosely referred to as ‘breaklines’ that are shown on the patterns can be omitted from the output if required, and are basically provided for reference and alignment of the workpiece in the rolls or press during forming. Typically, there are no sharp deviations from a smoothly curved surface which is part of a right circular cone. When formed by pressing, it is common to press at small regular intervals to achieve the best possible approximation to the required smooth surface.

Warning: Generators marked on cones, are not generally suitable for use as matchmarks between adjacent sections or strakes. This subject is discussed in detail elsewhere. Only in certain instances as specifically identified, generators may be used for this purpose.

The ‘breaklines’ (bendlines) shown on a pattern developed by triangulation may represent either a sharp bend or the equivalent of a radial line in a smoothly curved portion of the part. Breaklines are shown differently when required to be broken concave up (breakangle positive) or concave down (breakangle negative). The latter are uncommon and undesirable, but occasionally unavoidable.

FastSHAPES displays breakangles accurate to the nearest 1 degree of bend, measured as the angle between tangent planes on each side of the breakline.

Negative breakangles occur when a surface or part of a surface has a twist or wind in it, and sometimes if a warp exists. Warp can occur without negative breaks being required. Instances can arise in quadrilateral-shaped side panels of

transformers and bends, and generally in FastSHAPE ‘Surface’. It is essential that the presence of negative breakangles be recognized for what they represent.

The compensation applied by FastSHAPES in locating setting out points requires that compensated triangulated patterns be pressed to smooth surfaces where intended, in the same manner as for patterns developed by the radial line method.

In this sense, the breakangles shown are directly relevant only to a single tightly pressed bend. For smooth portions of the surface, the breakangle should only be used for general guidance as to the curvature required, and changes in it. For example, if 5 presses can be applied between each breakline as shown, then the target angle at each would be 1/5 of the angle indicated.

When a surface is developed with negative breaklines, it may or may not be necessary to actually reverse break the surface. Moderate twist can sometimes be introduced during assembly of the parts. The negative breakangles need to be taken into account in assessing the ‘average’ deformation to be applied to the concave-up bends. Pure warp is more commonly seen as constant or progressively changing breakangle on diagonal breaklines, none of which are negative, but when warp and twist are combined, their independent effects can be difficult to differentiate between.

Depending on the purpose of the Green which may be provided, this is usually removed after forming. In the case of the ‘fishtail’ effect on triangulated patterns, some parts of the End Green may be removed prior to forming, and can be nested ‘off the plate’ (tack a bit of scrap to the edge if necessary!) during cutting.

Locating longitudinal seams to avoid cruciform joints is readily achieved when joint location is specified by sweep angle with the radial line method.

For triangulated assemblies, or otherwise when only the eight predefined longitudinal seam is available (triangulation assemblies, some radial line assemblies), a **LongSeamOffset** dimension may be specified. This is applied in an alternating manner between adjacent parts or strakes in an assembly by adding / subtracting the specified offset dimension to any EndGreen, which may be specified. (When no End Green is specified, the reason for the previously mentioned ‘fishtail’ effect becomes apparent).

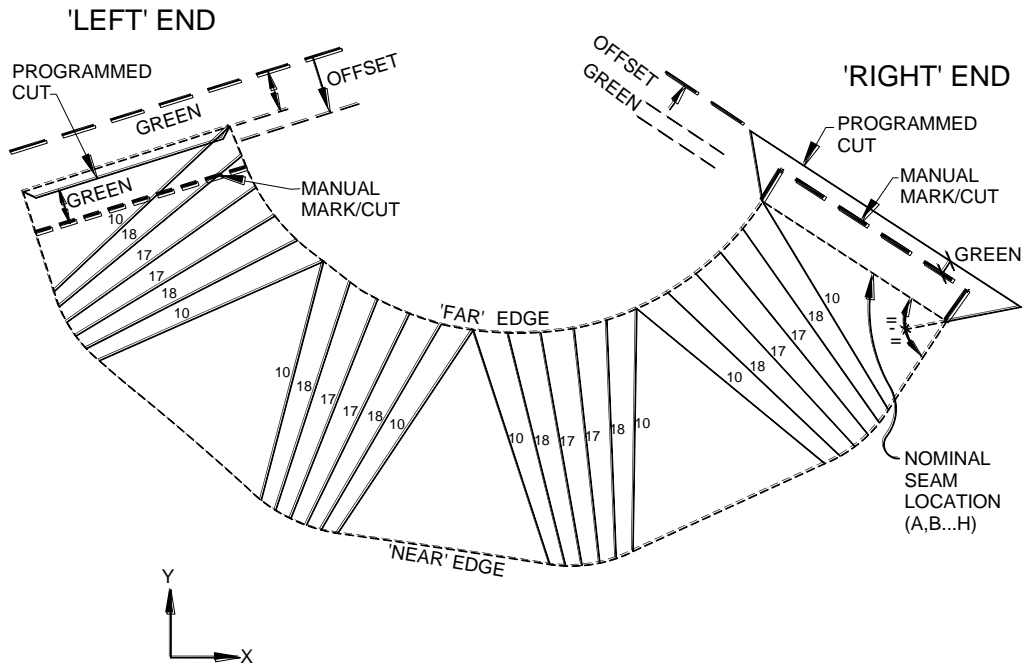
Unless all eight possible longitudinal seams are required in order to break the assembly down into patterns of available-stock-nestable sizes, it is often more effective to make two runs with the same layout data, the first with half the parts having the required longitudinal seam location, the second the converse. This procedure is facilitated through the ability to select all or just some parts for output.

A similar procedure can be followed for assemblies using different material thickness' for those FastSHAPES providing for a single thickness only.



Some FastSHAPES do handle different thickness'.

When all else fails, patterns can be edited using FastCAM.



GREEN AND LONG. SEAM OFFSET

Patterns as displayed in and / or printed from FastSHAPES do not contain any Job reference material, dimensioning or the like. In the event that this is required, FastCAM provides these services. So, obviously, does any CAD program of the user's choice. This enables production of Manufacturing Detail Drawings which can be fully annotated, check dimensions included as required, borders / title blocks added, etc. in respect of individual part patterns.

Other Manufacturing Detail Drawings which may be required, include Assembly drawings with Parts identification, Bill of Material data, Assembly or (formed) Part check dimensions etc. These may be produced using a CAD program and the 3-dimensional DXF output file.

Warning: check dimensions for Parts or Assemblies for 3-dimensional assemblies developed by compensated triangulation should ***NOT*** be taken from the 3-dimensional DXF file output by FastSHAPES for

reasons explained earlier. Pattern data on the other hand is as reliable as it can be created, regardless of development method. Assembly 3-dimensional DXF data for assemblies developed using radial line method is reliable, subject to qualifications made earlier concerning location of the setting out-surface.

Autonest (180 °Alternating Seams)

An “Autonest” is where each alternating gore seam, is rotated by 180 °. This option is available in:

- Elbow,
- Lobster and
- Bend

For the ‘Autonest’ the, mid cuts are programmed with no kerf compensation. Depending on the width of the cut and the number of gores, there is a tendency for the overall size to shrink, which can be offset by the user specifying an increase in bendradius of:

$$\frac{(n-1)*w}{\text{TANGENT}(0.5*\text{BendAngle}/(n-1))}$$

where: n is the number of gores,
 w is the width of the cut.

However, a material thickness effect common to both developments if the components are assembled without weld preparation is that there will remain a surplus of material on that side of the plate mid-thickness facing the inside of the bend. This tends to make the development grow in overall size, while leaving a small gap between the gores in the two sides of the development. This tends to offset the shrinkage of the ‘autonest’. If the surplus material is not to be removed, then the bendradius decrease needed would be:

$$(n-1)*t$$

where: n is the number of gores,
 t is the material thickness.



NOTE: For material thicker than a ¼ of an inch (5 or 6mm), it is not good practice to assemble without weld preps. Not so much because of poor weld quality as because of distortion likely to be created in the final welded assembly,

because of variation in weld gaps and included angles. This decision is entirely in the hands of the fabricator.

It can be seen that NO BendRadius adjustment would be needed for an ‘Autonest’-ed development without weld preps when:

$$w = t * \text{TANGENT}(0.5 * \text{BendAngle} / (n - 1))$$



NOTE: For a tapered cylindrical Lobster, to be developed using the ‘Autonest’ feature, the intermediate gores will NOT be of equal length. This is a corollary of development from a right-circular cone. Therefore an ‘Autonest’ can only be applied to a Lobster when it has constant centreline radius, or the diameter remains constant.

Setting Out

Patterns

Patterns as generated by all FastSHAPES are set out on the conventional XY orthogonal grid, X horizontal, positive to right, Y vertical, positive upwards.

In the case of patterns of Parts developed for 3-dimensional assemblies, the pattern has been pre-rotated to fit a minimal area enclosing rectangle, the size of which is indicated in the PartsList available for each FastSHAPE after development.



NOTE: All such patterns are presented on an ‘inside-up’ basis, and patterns generated for “Wrap-Around Templets” are presented on an ‘outside-up’ basis.

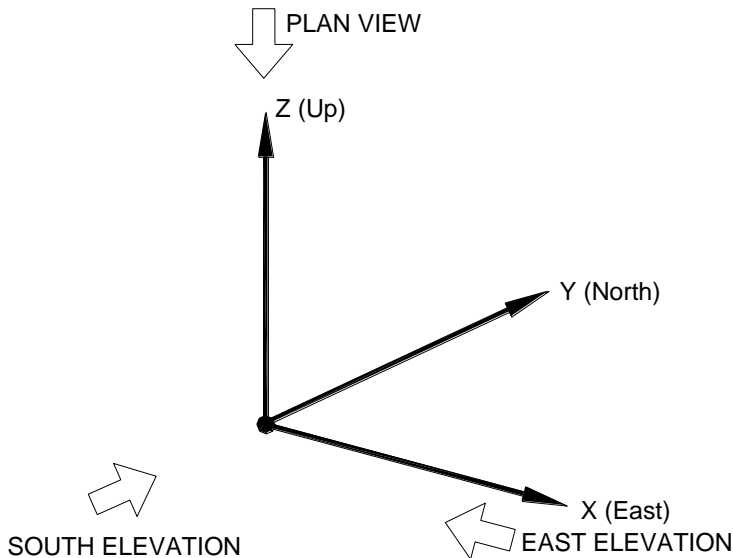
Unless specifically indicated to the contrary for a specific FastSHAPE, the enclosing rectangle size includes marking, some parts of which may lie outside the cut edge, and the cut edge will include any green specified.

Assemblies

Setting out for 2-dimensional parts or assemblies, is as described above for patterns.

Setting out for 3-dimensional assemblies is largely dependent on the individual FastSHAPE. However, the following general principles are applied

- a right-handed coordinate system is used, defined by axes X (east), Y (north), Z (up) is used

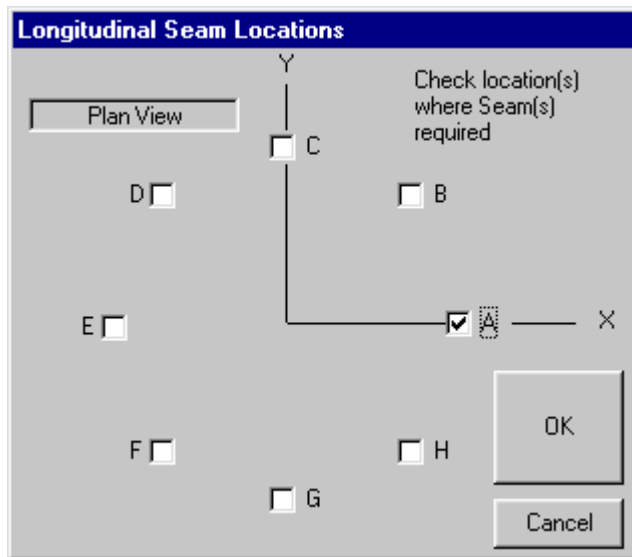


THREE DIMENSIONAL SETTING-OUT

- FastSHAPES providing for a maximum of 8 predefined longitudinal seam locations, have their ‘base’ in the XY plane, axis generally upwards so that ‘top’ has a positive Z value
- otherwise, the setting out is generally ‘free-form’ in the hands of the user, but may depend on the particular FastSHAPE. That is, the XYZ system remains right-handed, but in terms of east/north/up, the allocation to XYZ is in the hands of the user. It is recommended that the default definitions be used unless there is good reason to depart from them. The 3 dimensional viewing facility provides standard plan (on XY), east elevation (on YZ) and south elevation (on XZ) viewing positions, for example.

When up to 8 preset longitudinal seams positions are defined these are selectable from a dialog, and are (viewed in plan, from above, down the Z axis etc.)

- A X (east)
- B (north east)
- C Y (north)
- D (north west)
- E -X (west)
- F (south west)
- G -Y (south)
- H (south east)



These eight positions are generally as indicated, but vary with the shape, For example; B may appear at the northeast corner of a rectangle, rather than at 45 degrees north of east. The points may also suffer some relocation to facilitate the proper development of the part, but they retain their relative positions as indicated in sequence around the girth of the assembly.

FastSHAPES using this scheme are the transformers and bends using compensated triangulation development, plus 'CBranch'. For all except extremely large assemblies, eight longitudinal seams is generally excessive. The basic purpose in permitting up to 8 is to facilitate avoidance of cruciform joints in the assembly while leaving reasonable flexibility as to general location. For example, joints may be specified at A and E for a base collar, C and G for a transformer

body, then A and E again for a top collar. This is generally a more satisfactory means of avoiding cruciform joints than use of the Longitudinal Seam Offset described elsewhere.

Free-form setting out applies to ‘Surface’, and to FastSHAPES using radial line development other than ‘Cone’, ‘Dr.Cone’ and ‘CBranch’.

Collars and Mitters

Most 3 dimensional FastSHAPES provide for the definition of collars as prismatic sections of ductwork at base and top of an assembly, or if preferred, or in stronger context, inlet and outlet ends. The reasons for providing for collars are

- as mentioned elsewhere, when compensated triangulation development is used, the compensation is actually applied to the collar whether present in fact or only in principle.
- any departure in the assembly centerline axis from a straight line defines a bend in the axis at which a miter cut through the duct is either desirable (triangulation development) or mandatory (right circular conics / radial line development)
- when a miter cut is used, a collar greatly simplifies the specification of the assembly geometry, the manufacture of its parts, and the alignment and connection of the assembly to adjoining structural elements (ducts)

FastSHAPES using potentially more than one right circular cone are set out either explicitly (e.g. Bifurcate, Penstock, BifurcatePlus) or implicitly (e.g. CBranch) using the method of common central spheres.

A right circular cone is a surface, which is tangential to a pair of spheres when one sphere does not fully contain the other. Any two cones which are tangent to a common central sphere intersect in a plane, and the line of intersection drawn on that plane defines an ellipse. Lines connecting the centers of the three spheres define the centerline axes of the two cones.

A property of this construction, which is often misunderstood or not appreciated is that the center of the common central sphere, (the bend point of the cone axes) does not necessarily lie in the plane of intersection of the two cones. The axis

bend point lies in the plane of intersection *only* in the degenerate case when all three spheres have the same diameter, and both ‘cones’ are in fact equal diameter right circular cylinders.

A further property is that, when viewed in a plane containing both cone axes and the plane of intersection is seen as a straight line, the latter does *not* bisect the angle between the cone axes *except* in the degenerate case when the cylinder axes also intersect in the plane of intersection.

It follows that a miter cut is mandatory wherever the centerline axis contains a bend, and that at least a square cut not generally through the common sphere center is necessary at the intersection of two non-identical cones.

It also follows that any attempt to specify a miter angle or miter cut location which has been based on an incorrect setout is doomed to failure. It needs to be appreciated that the spheres generally, and the common central sphere(s) particularly, are the basis of a correct setout and not some mysterious property associated with a conceptual wish-list.

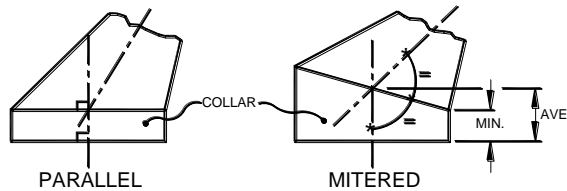
A final relevant property of the method concerns generator location and match points. Even when generators are referenced in identical manners, and equal sweep angle intervals are taken in two intersecting cones, there are only two locations around the joint at which the generators can be made to match. These are at the extreme inside and outside of the bend, where the plane containing both cone axes intersects both cones.

For example, if the assembly is set out such that this plane is horizontal, and sweep angles are started in each cone on this plane (0 degrees) then only the 0 and 180 degree generators will match. Neither the Top-Dead-Center (TDC) generators at 90 degree sweep, nor any others between and excluding 0-180-360, will match each other at the joint in a general sense. In the degenerate case when both are cylinders, 1-for-1 matching around the periphery will occur, but this is the only situation in which this can occur.

Based on the number of times we have seen assemblies drawn with impossible setouts, we make no apology for appearing to over-reinforce these messages. These effects can be seen quite graphically in FastSHAPE ‘Bifurcate’ which has been designed to demonstrate these points.

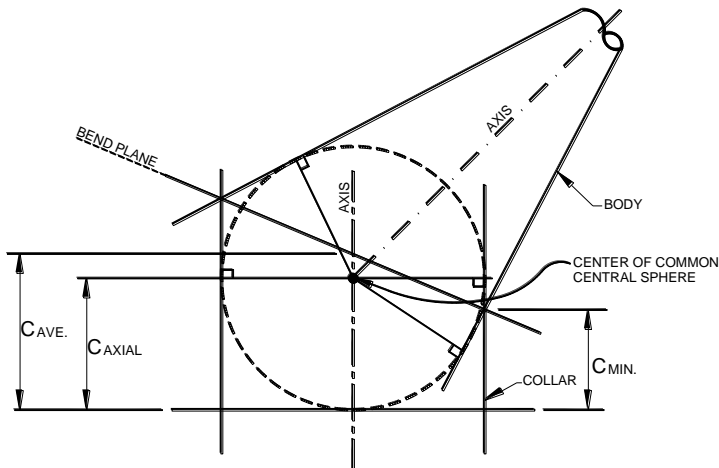
When a FastSHAPE provides for a collar to be specified explicitly, its prismatic cross-sectional shape is directly defined, and its length is always measured in the centerline axial direction.

Depending on the particular FastSHAPE, the length may be defined as minimum, average, or axial. The common feature is that the miter angle is never specified – it is always calculated by the FastSHAPE.



NON-CONIC SURFACES

COLLARS AND MITERS
CONIC SURFACES ONLY



For right-circular conical assemblies, when a common central sphere is not explicitly defined at a centerline axis bend, as with the collars of ‘CBranch’ for example, FastSHAPES inserts a hidden common central sphere for purposes of the bend calculation.

For assemblies other than right circular conics, FastSHAPES locates the miter cut plane through the centerline axis bend and bisects the bend angle – the very technique that is incorrect for right-circular conic assemblies!

Jointing Details

Assembly joints are classed as longitudinal (i.e. those generally lying in the same direction as the centerline axis of the assembly) and circumferential. As noted earlier, the principal jointing technology is welding.

Longitudinal joints in 3 dimensional FastSHAPES require little definition. They are characterized by being straight, both in flat pattern form and in the final assembly form, and by normally having a requirement for an unvarying weld preparation along their length.

Any required longitudinal weld preparation is readily definable and applied to square-cut edge as a secondary operation (often after forming and / or removal of green), and although details may differ, the overall characteristics of longitudinal joints vary little between medium and heavy plate construction.

Circumferential joints, in contrast, may have portions of them similar to longitudinal joints, but are more generally seen as being non-straight, both in flat pattern form and in final assembly form. Green is less likely to be required. These joints may or may not require to have variable treatment given to the weld preparation along the length of the joint, although generally, the heavier the construction the greater the potential need in order to maintain economy in the jointing process.

Ideally, what would be desired is for at least one cut to be made obliquely through the pattern material in a direction corresponding with cutting directly into the plane of intersection between adjoining parts in the final assembly. This would permit parts as cut and formed to be assembled without the need to remove any further metal. Any metal removed for the weld prep. itself would not interfere with assembly of the joint, and even very large root faces would butt together perfectly. Such facility is available to very few users.

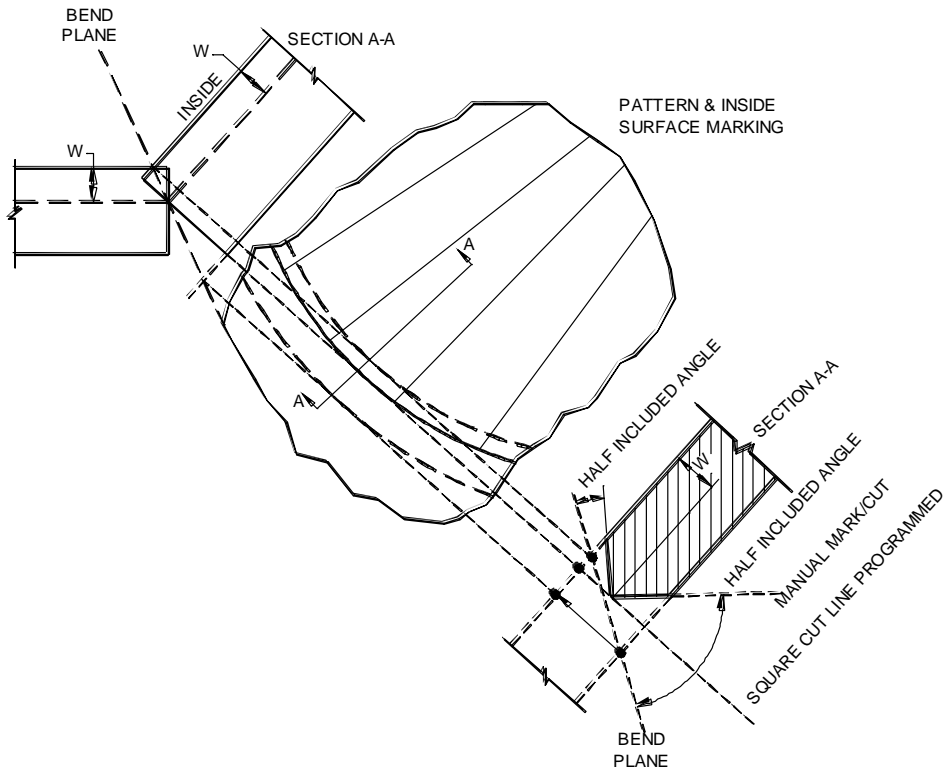
Other than as may be provided and documented in specially requested non-standard versions of the software, all cutting for circ. joints assumes a single cutting tool is available to make a single square cut through the pattern material. The specified weld preparation detail determines the location of this cut, and the location of any marking that may be applied to assist in the further cutting of the

preparation by secondary process(es), including any required pre-assembly metal removal.

FastSHAPES provide for a number of standard circumferential weld types to be specified, not all of which are available in all instances. The details are

Fixed Butt (Groove) Weld

This has a constant depth to root, w , along the length of the circ. joint. The depth is measured from the internal surface of the assembly (top face of the flat pattern.) The depth, w , may have a single value from 0 (i.e. a single bevel for fully external welding), to the material thickness (i.e. a single bevel for fully internal welding),



FIXED BUTT (GROOVE) WELD

or any intermediate depth within the thickness (i.e. a double bevel for welding from both inside and outside)

While circumferential development is always based on the mean thickness (other than in tightly pressed corners at previously discussed), longitudinal development is based on the depth ‘w’.

FastSHAPES utilizing development by triangulation (‘Rectircle’, ‘Rort’, ‘Obcone’, ‘Bend’, ‘Lobster’, ‘Elbow’, ‘Surface’) are assumed to be more appropriate to medium weight construction and have the depth, w, set at one half the material thickness with no user option, and no additional data is provided. In all but a few trivial cases (e.g. parts each side of the joint aligned with no angular deviation), some secondary treatment prior to assembly is required and usually undertaken as part of the beveling process if welding from both sides is required. In the event that welding from one side only is to be used, removal of metal from the other side of the joint in some portions of its length is usually required prior to assembly.

FastSHAPES ‘Bifurcate’, ‘BifurcatePlus’, ‘Penstock’, ‘Prun’, ‘Pbranch’ all utilize radial line development, are more appropriate for heavier plate construction, and provide for user-specification of depth ‘w’.

In these cases, three developments are actually undertaken, one on the inside surface, one on the outside surface, and one at the specified depth. All are transferred squarely through the material to the inside surface of the assembly where they can be marked on the upside surface of the flat pattern. Square cutting then follows the path representing the required depth, which usually removes portions of one or the other two marked paths. Marking that remains may then be used as a guide for the minimal amount of metal that must be removed prior to assembly. Clearly, any additional secondary cutting for the weld prep. itself is additional to this, and the bevel angles themselves do not constitute a user data-entry requirement.

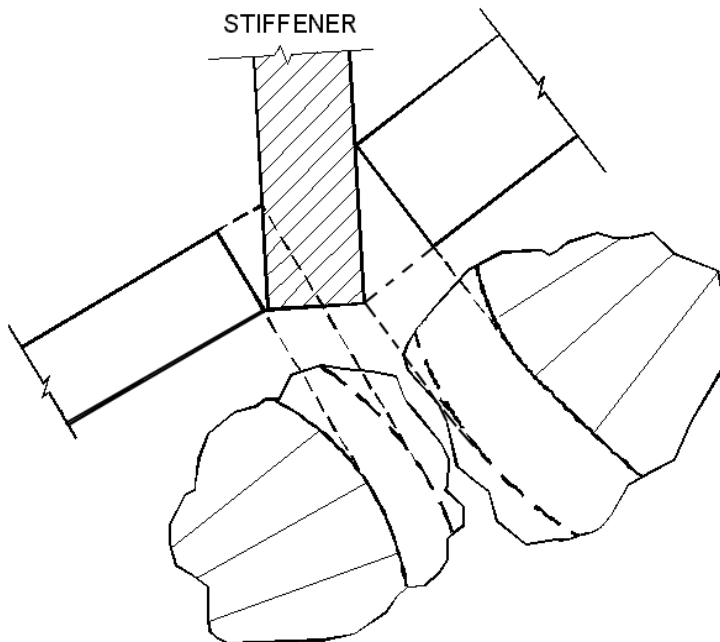
Combination, Butt – Fillet, or alternating Butt, Weld (the ‘unsafe’ square cut option)

Given that development is undertaken on both inside and outside surfaces in addition to at depth ‘w’ if not already at inside or outside, the cutting path may

alternatively be taken in one restricted condition along the inside or outside path which removes the most metal from the workpiece. The restricted condition is that the joint should be made against a flat element such as a set-in stiffener, and not between two developed shell parts intersecting in an infinitesimally thin plane.

This type of preparation has been given various names; two of which have been indicated, and none of which are critical to the main issue, that it should be used with some caution – it isn't 'safe' in the sense that removing a bit more metal is seldom cheaper than putting it back. In extreme cases, it can also lead to excessive weld included angle, hence welding cost, and distortion, hence further cost. It is generally confined in practice to medium rather than heavy plate thickness' for all the above reasons.

Its particular advantage is that the cut and formed workpiece can be immediately assembled to the stiffener.



**'UNSAFE' SQUARE CUT WELD
(COMBINATION, BUTT/FILLET)**

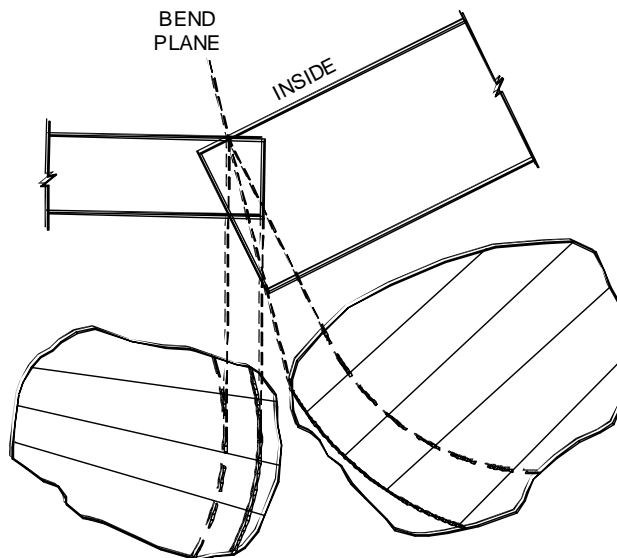
‘Safe’ Square cut Butt (Groove) Weld

The converse of the above is to cut along the developed line that leaves the maximum amount of metal on the workpiece. It can be applied without restriction and is always ‘safe’ – it is also necessary to always remove metal prior to assembly.

FastSHAPES ‘Bifurcate’, ‘BifurcatePlus’, ‘Penstock’, ‘Prun’ all provide a dialog from which to select / specify the required type of circ. weld. (Double click the data cell where circ.weld detail is requested)

FastSHAPE ‘Pbranch’ provides, in addition, several types of weld preparation which comply with certain national / international standards as discussed in its own Section.

FastSHAPE ‘Cone’ provides for all cone elements to be uni-axial and located by means other than planes of intersection. Hence, the question of circ. weld type does not arise, and in the standard FastSHAPES software, the detail is not requested.



'SAFE' SQUARE CUT WELD

Verification Facilities

Verification of data is provided at a variety of levels.

A primary verification facility attempts to advise when:

- data has been entered outside a mandatory or recommended range,
- mandatory data has not yet been entered, (data cell yellow),
- data having a typographical error has been entered. (*data cell red, e.g. when a letter has been entered where only numbers are expected, or a bad ‘inch’ fractional number is attempted*).

Each FastSHAPE contains a picturebox on the data entry form which will display a simple graphic of some appropriate view (plan, elevation etc.) or detail (e.g. tooth form) of the data. The graphic will not be displayed until a minimal set of sensible data is provided.

Two-dimensional FastSHAPES using the direct development method display their patterns on a form having a cursor location displayed. This provides for qualitative verification of the data, plus a limited degree of quantitative verification.

Further quantitative verification, if required, is best obtained by taking the pattern into FastCAM which has a powerful set of verification tools, entity identification and dimensioning facilities.

Three-dimensional FastSHAPES using either the triangulation or radial line development methods display flat patterns, and the same facilities as those described above are available for verification of the flat (2 dimensional) part patterns.

A 3 dimensional display is also available in this case to provide a qualitative verification of the assembly of parts. Cursor location is not indicated in this case. Its basic purpose is to verify that such things as 3 dimensional assembly offsets have been applied in the correct direction, etc., that is, that the assembly ‘looks right’ and has not been inadvertently opposite-handed, a sometimes common problem.

The assembly may be viewed as a wireframe model (with a stereoscopic viewing option), or, when not stereoscopic, may be shaded. Individual parts may be omitted from the assembly to assist with interpretation, and cut views / rotation in space are provided for the same purpose.

The purpose of the shading facility is to clarify and distinguish between internal and external surfaces when required. The internal and external surface colors are user-configurable. This matter can be critical with some FastSHAPES, ‘Surface’ in particular.

In certain cases, and particularly for branched assemblies, the use of triangular and planar quadrilateral facets to represent smooth surfaces and edges can create apparent imperfections in the model which do not necessarily exist in the real structure. These imperfections may be seen at circumferential joints, where the faceted edges intermittently intersect each other and show up as overlaps of, or holes between, the facets. The artificial interpenetration of facets can cause a failure to fully resolve all shading issues. In such cases, a slight change of viewpoint, or omission of one or more parts from the assembly, can allow the shading to be resolved.

When the surfaces on each side of a circumferential joint are developed from a common set of points located along the joint, as for most triangulation developments, the joint imperfections in the model tend not to exist. The radial line development does not use common points as the basis for development, hence the model imperfections will be more apparent in these cases.

The shading facility aims to be utilitarian rather than sophisticated. Taking the 3 dimensional DXF output into a CAD system will generally provide more sophisticated shading facilities if required. However, see warnings elsewhere concerning distortion of the model with some FastSHAPES depending on the method of development used – the 3 dimensional data may be treated quantitatively in a CAD system in restricted situations only.



NOTE: *Many of the structures to which FastSHAPES are most powerfully applied are specified by the designer on an assembly and welding*

manufacturing drawing, but there are no detail drawings provided for individual components at various stages of manufacture prior to assembly.

Creating such detail drawings is normally the most effective means of communicating to the workforce such details as green, seam offsets, joint locations, etc. etc. that have been allowed for in the development of the components. Check dimensions are also most useful manufacturing detail data.

FastCAM Inc. strongly recommends that the need for such communication should always be carefully assessed, and when appropriate, that FastCAM (or a suitable CAD program) be used to fully document the details embodied in the patterns or templets created by FastSHAPES. In very simple cases, handwritten annotation on pattern printouts from FastSHAPES may suffice.

Materials Control

FastSHAPES can optionally read the same Material.dat file used by FastCAM, or one of its own, or omit materials control facilities.

The Material.dat file is a simple text-editable (e.g. Windows Notepad) flat file which defines

Class, Specification, Density [,other ...]

in which ...

- Class is a class or group name for like materials
- Specification is a description of the particular product
- Density is the volumetric density expressed in kg/cub.m
- Other, if shared with FastCAM, is confined to use by FastCAM and explained in its documentation

The density is expressed in kg/cub.m regardless of the FastSHAPES installed Units (see Configuration). If density is known only in lb./cub.ft., then apply the following formula:

- **Density = 1000 x Rel.Density/62.4**

Where: Rel.Density is the Relative Density of the material, e.g. Carbon Steel = 490 lb/cub.ft.
62.4 is the Density of water in lb/cub.ft

Each FastSHAPE contains a dialog, which accesses the data from Material.dat and allows the user to select a material and specify its thickness. If the Material.dat file has not been provided or has been lost, the selection defaults to Class=(Unidentified) and Specification=(Unidentified) only, and permits entry of a thickness. However, density is not then known, so part masses cannot be calculated.

Similarly, Part masses cannot be calculated if no thickness is specified, or is specified as zero.

Those FastSHAPES which provide for Parts having different thickness' still provide a dialog. Any thickness specified in the dialog becomes the default thickness for all parts and is copied into any vacant data cells defining part thickness where it may then be changed.

Estimating, and Production Control

The purpose of Material Control is to contain the definition and selection of materials so as to facilitate automatic nesting in FastCAM, to ensure that only parts of identical thickness and specification are nested together. When FastCAM is not installed, or the automatic nesting facilities are not required to be used, it is nevertheless recommended that a Material.dat file be used so that Part masses can be listed in the PartsList.

Every FastSHAPE creates a PartsList table after development of the parts. Columns shown in the table are

- Part Description. This is created by the FastSHAPE according to the way the assembly has been broken down into strakes, segments and Parts
- Part Quantity. This is normally 1 per assembly, but may vary for certain FastSHAPES in which absolutely identical parts can be guaranteed regardless of data which creates them
- Width of a rectangle enclosing the pattern of the Part. Parts are tested for rotation and rotated as required to define a minimal area enclosing rectangle which includes any green applied.
- Thickness of the material
- Length of the enclosing rectangle
- Material Specification description
- Mass of ‘Quantity’ of the enclosing rectangle
- Nett mass of ‘Quantity’ Parts after all green has been removed. For triangulated parts in which a Longitudinal Seam Offset has been allowed, the mass is calculated for the theoretical pattern without seam offset, so the masses may be incorrectly distributed, but the total remains accurate
- Number of BevelCut paths. When bevel cutting is installed, (non-standard, special request item only) these are switched on/off in the Options menu under Output Options
- Length of BevelCut paths

- Number of normal (non-bevel) cut paths
- Length of normal cut paths
- Length of Breakline / Radial line marking
- Length of Edge Marking (the theoretical cut boundary, excluding any green, and excluding any Longitudinal Seam Offset for triangulated patterns)
- Length of any other marking which may be provided (certain FastSHAPES only)
- Further data may be listed as defined for specific FastSHAPES, generally forming radii and sets

Each row of the table represents a Part, numbered (1 .. number of different parts) and the last row shows total data for one no. Assembly.

Warning: the width and length of the enclosing rectangle is taken around all marking imposed on the pattern, it is not necessarily the nett size over the cut boundary, which also includes green, and some marking can occur outside the cut boundary.
Also due to rotation of some patterns during development and to slight differences in calculation the enclosing rectangle may differ slightly for similar patterns.

Columns always hold the same data as described above, and always retain the same relative position in the table. In the event that a column holds null data (e.g. mass, if thickness is zero, or density not obtained from Material.dat file), the column is hidden from view by collapsing its width to zero – it has not been deleted from the table. Column widths are assigned automatically by FastSHAPES, but may be adjusted by the user by dragging the gridlines between columns in the column header rows using the mouse

Estimating and Production Control Facilities

System clipboard facilities are provided. Individual cells may be selected by mouse dragging similar to MS Excel. Individual rows and columns may be selected by clicking in row / column heading areas. The entire table may be selected by mouse-clicking the top left cell in the headings area. The latter is generally the most appropriate. The menu Edit Copy (^C) function will copy the

selection to the system clipboard, from where it can be pasted to an Estimating or Production Control spread sheet

Any hidden columns in the PartsList table are copied to the clipboard, so that any spreadsheet set up as a paste destination should always be consistently constructed as defined above. Note that column and row headings are not included in selections, and are not copied to the clipboard.

After pasting, it is clear that any set of derived data can be obtained, recast for total required number of assemblies, specific PartNumbers allocated, etc. Such data is of the essence to Production Control.

Estimating Suggestions

Because all Parts can be accurately developed very quickly, and adjusted as to seam locations etc. to ensure that developed parts can be arranged to fit available stock materials, the Estimator’s job is greatly simplified and expedited. The PartsList plays a major role in estimating. Much of what would otherwise be value judgement, if not outright guesswork, can be removed.

One obvious approach to estimating is a full production ‘dry run’, using FastCAM facilities to auto-nest true shape parts and report further production cost factors. Whether or not such effort can be justified in a job bidding environment often depends on the quality and degree of finality of the source data. Often the effort is not justified, and can lead to unrealistically tight estimates. A coarser approach can often be sensible.

The Materials Control system outlined elsewhere, and facilities which permit copying of PartsList tables to a spread sheet, can open a veritable Pandora’s Box to the Estimator.

As a suggestion, with all PartsList tables copied and extended to total job quantities, and any other required data added,

- sort records by material specification and thickness
- reorder columns to find PartName, Qty, Width, Length (of enclosing rectangles)

- select ranges of the above for a given material and thickness
- clipboard each range in turn into FastCUT, a sister product of FastSHAPES, and nest it. (FastCUT provides a very powerful nesting capability for rectangular parts, or parts that can be contained within a rectangle.)
- clipboard FastCUT’s nested data (stock requirements) back to the estimating Workbook

Configuration

FastSHAPES contains a set of default configuration data which are established on the first-ever run if not already created in a FastSHAPES.CFG file distributed on the product release medium. This file is held in the same directory as the FastSHAPES.EXE file. In the event that this file becomes lost (deleted, renamed or moved), FastSHAPES will assume its default configuration.

The default configuration, or any current configuration, may be modified at any time by the user, either temporarily (‘Apply’ (for the duration of the current run)) and / or semi-permanently (‘Save’ (to .CFG file))

Configuration parameters are accessed from the Options menu or toolbar icon. Parameters are listed under various Tabs. Significant parameters are

Units

Dimensional Units may be millimeters or inches. Units may not be changed while any FastSHAPE remains open. With all closed / saved as required, the units may be switched for any subsequent data entry. FastSHAPES do not actually convert units – dimensional data must be entered consistently in the currently selected units set

Units are saved with data files. A data file saved in one unit set may not be opened when the other unit set is active.

Dimensional Units as selected give rise to a set of derived units as used in the PartsList table described elsewhere. Derived units are displayed in the dialog where Units are selected.

When Inch units are selected, all dimensions may be entered as decimal (e.g. 0.125) or fractional (e.g. 1/8) inches. No facility exists to recognize yards, feet, miles etc. 10’5-3/8 should thus be entered as 125 3/8, using a space to separate the whole inch portion from the fractional portion. Fractions smaller than 1/32 are not acknowledged. Decimal input is generally recommended.

When mm units are selected, input dimensions are usually whole numbers, but may be decimal.

‘Effective Zero’

In any work, there is a lower limit at which size ceases to be significant (often having an amusing if not outright crude description). This is it! Zero is not a permissible entry! Recommended value – the resolution of your NC machine, say .1mm or .004 inch – these are the defaults.

General Dimensional Tolerance

A bit like ‘effective zero’, but generally a bit bigger. Should be a realistic number appropriate to the user’s industry. For example, 2mm or 1/16 inch for a boiler shop. This parameter is used for a wide variety of purposes, including when fairing curves, as explained elsewhere. It also aims to prevent the transmission of ridiculously accurate results in response to assumed appropriate data.

Fairing

As explained elsewhere, fairing converts a theoretical set of point data into a smooth curve, to within the tolerance just defined. The maximum radius should be set to a ‘reasonable’ number – workshops are only so wide, and usually have internal columns! For users having NC machinery, these may have upper limits on they arc radius able to be handled, and this can be a good maximum upper limit.

‘Small’ arcs are those that are fully contained, including their centers, within the bounding rectangle of the pattern, (i.e. any arc whose end point(s) or center is outside the bounding rectangle is not a ‘small’ arc).

The ‘Lines only’ option ensures that only points within the specified General Dimensional Tolerance of a straight line are relocated onto that line. Points lying on extremely long radius arcs may thus be forced to define a faceted

representation of the long radius arc which has fewer facets than if left unfaired, but points defining shorter radius arcs will not be faired.

Generally, the last option, ‘arcs up to (radius)’ is recommended as a default. Other options may be tried in the occasional extreme circumstance.

Marking and Nicking

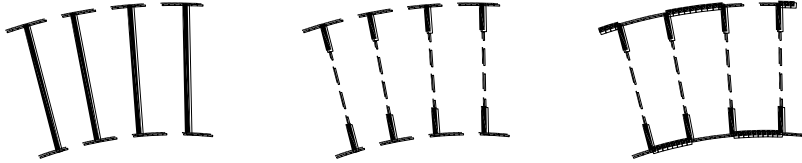
FastSHAPES generally assume that any NC equipment owned by the user, or accessible to him, will be fitted with line marking facilities, and that the user is interested in getting the absolute best from the machine.

Unfortunately, this is not always so. FastSHAPES provides user options to switch on/off various marking output facilities, and to truncate various line lengths.

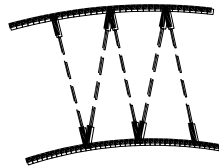
For users without marking facilities, the ‘nicking’ option is available. A ‘nick’ is a corruption of the cutting process, which causes the cutting tool to deviate from its normal path and travel down/back a ‘NickDepth’ distance on each breakline in a triangulated pattern. The sequence of operations is

- development of the pattern. In this stage, circumferential edges exist as a series of straight lines connecting the pattern layout points
- addition of any green specified to all edges as appropriate. In this stage, angles between the straight lines defining the circumferential edges are bisected. Additional layout points are then added on the bisector at the specified green dimension from, and parallel to, the original edges to form the actual cut boundary.
- when nicking is specified, the nicks are inserted down the bisectors from the pattern setout points on the cut boundary. If no green is applied, the nick still proceeds down the bisector into the workpiece, rather than through the green. (The nick does *not* run down any breakline, several of which may end at a common pattern layout point.)
- portions of the actual cut boundary representing the offset circumferential joints are faired. If nicks have been added, this corruption of the cut path inhibits fairing, so that the cut path remains a series of straight lines between nicks.

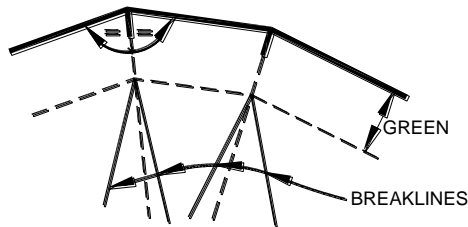
- the original (theoretical) edges are faired and marked. If green has been added, the cut path and theoretical edge as marked may not necessarily remain parallel.



BREAKLINE MARKING OPTIONS



CUT EDGE 'NICKING' OPTION
- NO GREEN



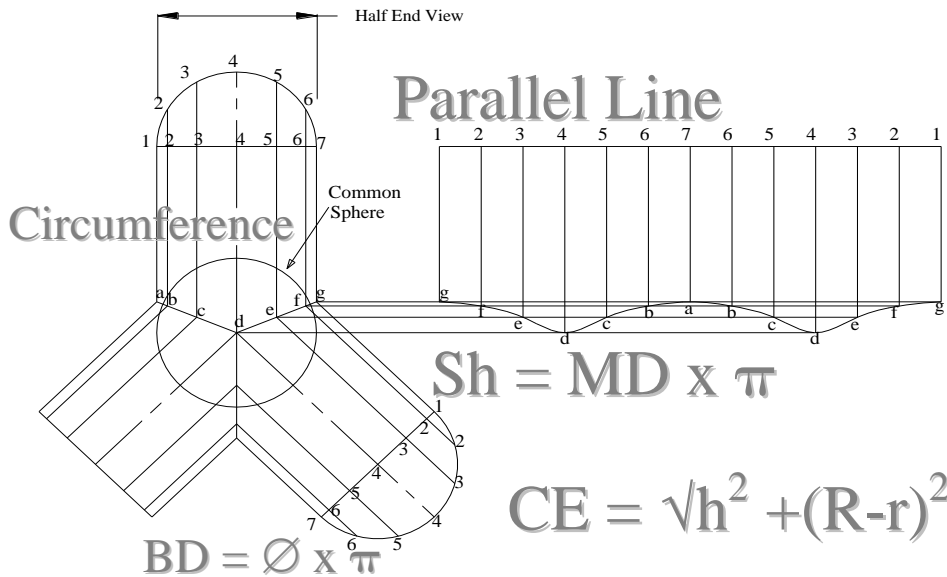
CUT EDGE 'NICKING'
OPTION WITH GREEN

CONFIGURATION - MARKING/'NICKING' OPTIONS

Provision exists in (some, not all, radial line development) FastSHAPES for the output of bevel cutting data to FastCAM for generation of appropriate NC code. (This is a specialized facility not generally available on NC cutting machines, although enquiry's are welcomed. To date, a number of customers have been very satisfied!) In lieu of such specialized facilities, applicable FastSHAPES use marking output to define the circumferential edge bevel cutting details. These are of little use without marking facilities being fitted to the NC equipment, naturally.

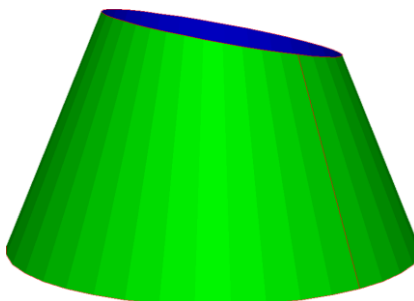
CHAPTER

2



Right Circular Cones

Dr.Cone



‘Dr.Cone’ is presented essentially to represent the classical trade school / college development exercise. The opportunity has simultaneously been taken to demonstrate the differences between radial line development as implemented in FastSHAPES and uncompensated triangulation development (the traditional lofting method taught in many schools).

In this particular instance, the compensated triangulation and radial line methods, both as implemented in FastSHAPES, are directly equivalent, and no distinction is drawn between them – the separation of the generators is calculated from the development sweep angles, and is unaffected by the number of development intervals chosen. The latter will be seen, both graphically and in the table of numbers, to have a major impact on an uncompensated triangulation development.

In the pair of patterns shown on the data entry form, the pattern shown in green is always the correct pattern as developed by radial line method, or by compensated triangulation with angular separation of generators computed from arc lengths. The pattern shown in red is always that found by uncompensated triangulation, using chord lengths. For clarity, generators are not shown on either pattern.

The table can be switched between solutions, with red values (option ‘Chord Lengths’) corresponding with the (approximate) red pattern, and green values (option ‘Arc Lengths’) corresponding with the (exact) green pattern. The set of green numbers will be seen to remain unchanged with different numbers of

development intervals, while the red set will change and become progressively closer to the green set as the number of development intervals is increased.



The table can be copied to the system clipboard then pasted elsewhere. To select any cells in the table, uncheck the ‘Collect Headings’ box. All cells can be selected by clicking the top left cell in the table headings area. Ctrl+C will copy to the clipboard, etc. – see the Edit menu.

In this particular case, the patterns on the data entry form and tabulated data for them are defined in a (U,V) axis system, and the 3 dimensional tabulation in (U,V,W), so as to leave (X,Y) and (X,Y,Z) available to the student for manual calculation and to facilitate comparison.



Note: The pattern origin is at the left end of the generator lying in the U axis, whereas the pattern origin in manual lofting will often be taken at the point corresponding with the cone apex.

All fairing of circumferential edges has been suppressed and cannot be activated through the Options menu.

The oblique cut at the top of the cone is optional, and included for educational purposes only.

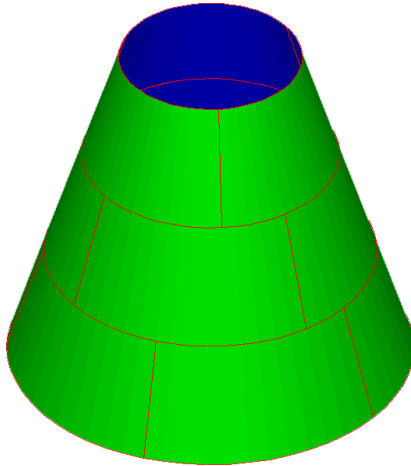
Thickness may be specified, but the cone layout dimensions are confined to lie at mean thickness.

Patterns and the 3 dimensional model obtained via the Display menu or by left/right clicking the patterns on the data entry form are always for the approximate solution. These may be compared with exact patterns and 3 dimensional model by running the same data through 'Cone'

The “PartsList” table will show differences in nett mass and overall pattern size between runs with different numbers of development intervals. Again, running the same data through ‘Cone’ will provide exact “PartsList” values for comparison.

If required for a real situation because of the oblique cut facility, specify the cone dimensions at mean thickness and use an appropriately large number of development intervals. Bear in mind that the circ. edges remain unfaired.

Cone



FastSHAPE ‘Cone’ is designed specifically to handle right circular cones in a variety of sizes and conditions between its degenerate extreme cases of a right circular cylinder to a flat annular ring. The radial line development is used and is exact.

The setout dimensions for the cone are based on an infinitesimally thick surface having specified base and top diameters and axial height. The side length is shown for reference. Finite thickness material is then disposed according to how this setout surface is intended to lie, Inside, Outside, or at Mean Thickness. *Refer drawing on next page*

The cone may be broken down into a number of individual strakes, each of different thickness.

Each strake may have a different number of segments, including fractional segments.

For example:

If 2.9 segments are specified for a strake, there will be two main segments and one makeup segment.

The (plan view) angle swept by each main segment will be:

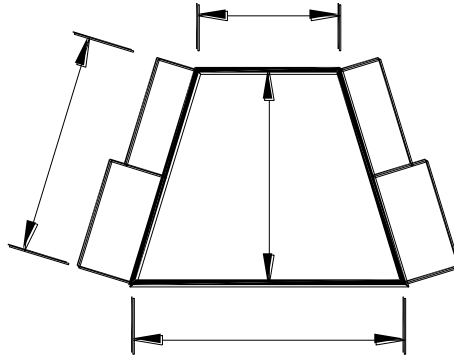
- $1 \times 360 / 2.9 = 124.14$ degrees,

and, that swept by the makeup segment will be:

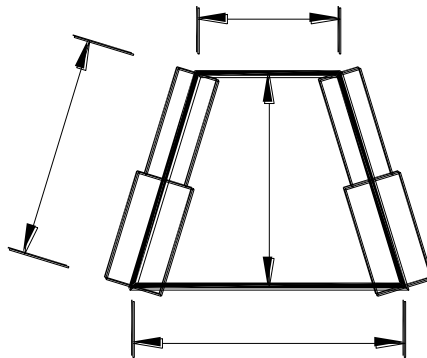
- $0.9 \cdot 360 / 2.9 = 111.72$ degrees,

with the total being:

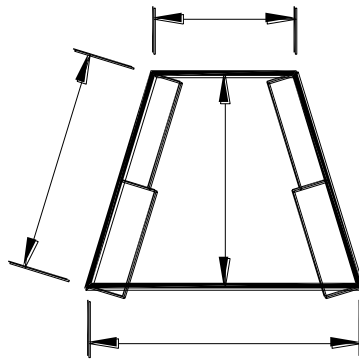
- $2 \cdot 124.14 + 111.72 = 360$ degrees.



DIMENSIONS
● INTERNAL



DIMENSIONS
● MEAN



DIMENSIONS
● EXTERNAL

'CONE' - SETOUT OPTIONS

The plan view shown on the data entry form, and the 3 dimensional model, locates all first segment longitudinal seams in direct alignment. The reason for this is that it simplifies interpretation of the diagram for purposes of ascertaining that individual strokes can be rotated on assembly to ensure that no seams are aligned.



Actual seam locations are seldom critical in a real structure of this type, provided they are not aligned, thereby requiring T joints in preference to cruciform joints where longitudinal seams intersect circumferential seams. This requirement may include the base and top circumferential seams where the cone may join with other elements.

Cone provides the option to include or exclude breakline marking (radial lines) as forming guides. These are not breaklines in the traditional sense, cones formed by pressing would normally be pressed at very close intervals. Whether pressed or rolled, such lines help the forming machine operator keep the workpiece correctly aligned in the machine.

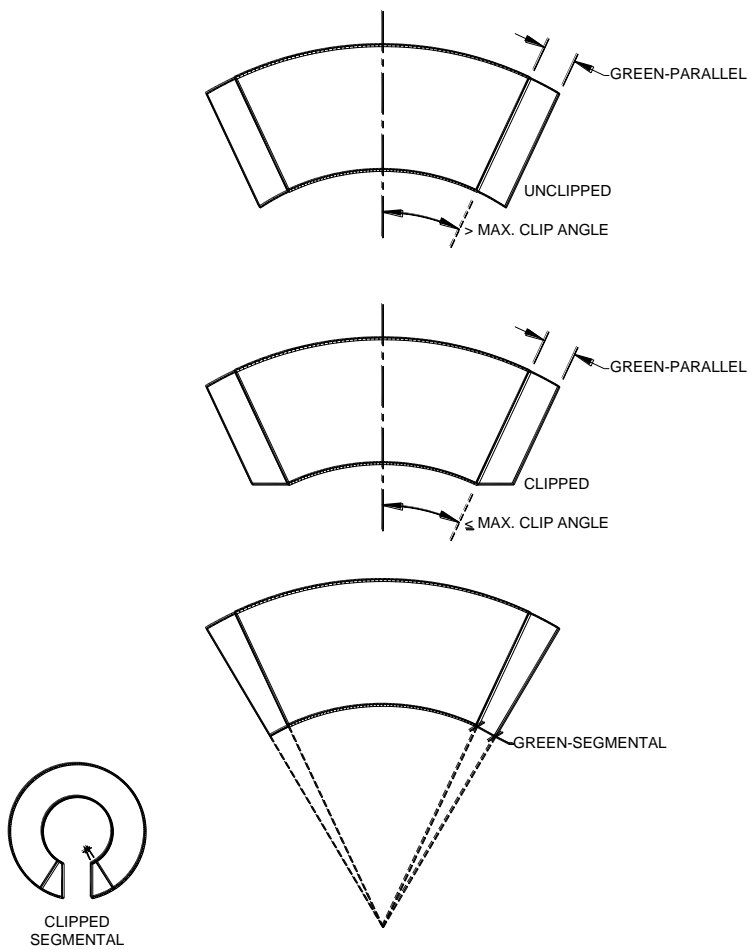
End green is optional, and when used may be Segmental or Parallel. *Refer drawing on next page*

Segmental green adds a wedge-shaped margin to each end of the pattern in which the specified dimension is held, approximately, at the smaller end of the wedge which always extends the arc on the smaller edge of the pattern to facilitate forming by rolling. Subsequent trimming of Segmental green is simplified when a marking tool is available to mark the required final end of the pattern, otherwise a manual markout is not as simple as when Parallel green is used.

Parallel green adds a constant width margin to each end of the pattern and is readily marked out manually as may be required by trimming after forming.

Either type of Green may be clipped, subject to one restriction, to facilitate nesting of patterns on stock sheets. The restriction is that the inner edge of Segmental green is not clipped – it always follows the inner edge arc of the pattern to facilitate forming by rolling.

Patterns are symmetrically disposed about the vertical Y axis. The ‘end angle’ for clipping purposes is the angle between the vertical axis and the theoretical end of the pattern (excluding the green). When the pattern end angle is under the specified angle (90 degrees max) and the total pattern less than a semicircle, the inner edge of Parallel green is clipped. If the total pattern is more than a semicircle, the outer edge of the pattern is clipped. The clip line is parallel to the X axis.



'CONE' - GREEN AND CLIPPING

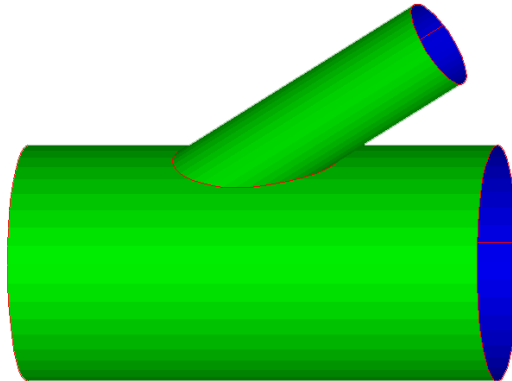
In the extreme (largely impractical but nevertheless possible) case of a single segment pattern for a very flat cone with large green specified, the pattern becomes almost a full circle and the green at each end would foul itself. ‘Cone’ then undertakes some additional clipping along the Y axis.

Edge green may be added to either or both the top / bottom circumferential edges of all patterns. This may be for purposes of providing a cutting margin, but more commonly may be necessary when forming using a cone rolling attachment on pyramid rolls, which can cause damage to the inner edge of the pattern. The term top / bottom is used in preference to inner / outer edge to cover the case of cylinders which are more likely to have green as a field trim margin than as a rolling margin.

In the standard version there is no facility for specifying weld details, and all edges are programmed for square cutting only.

Pipe Branches

Pipe Branches



Pipe Branches are covered in a suite of FastSHAPES named PBranch, including PBranchS for Structural applications, plus PBranchI and PBranchO for process applications using setIn and setOn details respectively.

Some differences exist in this suite arising from differences in the applications, and are, briefly ..

Most FastSHAPES specify element sizes internally. In this case, pipe diameters are specified externally to comply with line pipe standards, this pre-manufactured product being the most commonly used in PBranch applications. Wrap-around templets are provided for this most common case.

Flat cutting patterns are presented on the standard inside-up basis, to cover cases where standard line pipe may not be used and the pipe is rolled from flat plate. PBranch provides a possible and rare instance in which it may be desirable for patterns to be presented on an outside-up basis, as for example when the pipe is too small for internal access, and it is desired to mark penetrations prior to rolling, and cut them from outside after rolling. This is readily achieved for symmetrical assemblies simply by rolling inside out. For asymmetric assemblies, it is necessary to specify in PBranch an assembly opposite handed to that required, then rolling inside out will restore the correct handing

In PBranchS, the main has no penetration, and enhanced 3-dimensional viewing showing the branch thickness is provided to assist the visualization of a variety of weld details. Other than stiffeners in BifurcatePlus, this is the only FastSHAPE in which thickness is acknowledged in the 3-dimensional viewing module. Some weld details are similar to the FastSHAPES standard types –1, 0, 1 described elsewhere under Jointing Details. (In particular, the PBranchS General Purpose Butt (Groove) weld detail is specified differently from the standard type 1 weld). Other weld details are confined to this particular FastSHAPE.

In PBranchI and PBranchO, weld details follow the FastSHAPES standard types –1,0,1 as described in Jointing Details. A reinforcing element (compensating plate) may also be specified. The 3-dimensional viewing in this case also follows the general FastSHAPES standard, omitting thickness, and in this case not showing the compensating plate.

PBranchI and PBranchO

The standard edition provides for flat cutting patterns or wrap-around templets for branch, main and compensating plate elements of a pipe branch in which the main is penetrated for process applications. The branch may be set-in or set-on relative to the main penetration.

The wrap-around templet options permit specification of a thick external coating over which the templet would be wrapped, the templet containing data pertaining to the cutting paths required after removal of the coating. That is, the templet data is taken for the underlying pipe material and transferred to the outside of the coating. Templets are provided on an outside-up basis for plotting via FastCAM or any suitable CAD software, or for the NC cutting or manual lofting of sheet metal or other materials which may be used as a templet. Templet material is assumed to be of negligible thickness.

The flat cutting patterns are provided on the usual inside-up basis.

Provision exists for the addition of green to the longitudinal seam edges to facilitate rolling. There is no provision for end green, however, the branch pattern by default is cut rectangularly around the marked profile. The profiled edge may be cut by ticking the check box. The penetrations in main and compensating plate

are also marked only by default, and may be cut by ticking the appropriate check box.

The pipe OD and wall thickness of both branch and main are required, as is the thickness of the compensating plate. If the latter is specified as 0, the compensating plate is omitted.

The compensating plate pattern is cut as an obround, centered about the intersection of the branch axis with the outside wall of the main. It may be wider than long, or vice versa.

The branch angle relative to the main may vary from a minimum of 15 degrees to a maximum of 90 degrees, and the longitudinal seam may be located anywhere around the circumference of the branch. The seam in the main may not intersect the penetration, following usual good practice. In circumstances where this is unavoidable in practice, it is recommended that the seam be specified 180 degrees from its required location, and FastCAM be used to edit the patterns as required. The seam location may be specified in multiples of 5 degrees from the bottom-dead-center reference position.

The branch may be eccentric to the main, subject to the restriction that the penetration of the main must not exceed its internal diameter, and the face of a set-on branch may not lie external to the main outside diameter. Also see Special Notes – Extreme Eccentricity below.

The three FastSHAPES standard weld types –1,0 and 1 are provided for the branch / main joint, and are depicted in a sectional view accessed through the ‘Show’ command button.

The colors used in these views correspond with pattern colors for straight cutting and ‘other’ marking (defaults red and yellow) as set under the Options menu. Inside/outside mark and cut line at cut edges are shown, plus addition marking to facilitate assembly, all as repeated on the patterns. For a set-on branch, the outside of the main is marked around the outside of the branch. For a set-in branch, the outside of the branch is marked around the outside of the main.

For set-on branches, the gap under the branch is measured radially from the outer surface of the main. (The branch is developed to a cylinder concentric with the main, and having a diameter of $OD_{main} + 2 * Gap$)

For set-in branches, the gap around the branch is effectively measured in a cylindrical surface concentric with the main and located at the weld root. Direction of the measurement is ‘normal’ to the branch face.

The compensating plate gap is measured similarly to that for the set-in branch / main joint, with the weld root at the outer surface of the main.

Projection is shown on the ‘Show’ dialog, and may be specified negatively (limited to negative thickness of the relevant element.)

For a set-on branch, projection is measured radially inwards from the inner surface of the branch to define a cylindrical surface concentric with the branch and having a diameter of $ID_{branch} - 2 * Proj$. Intersection of this surface with the main defines the main penetration.

For a set-in branch, projection is measured radially inwards from the inner surface of the main to define a cylindrical surface concentric with the main and having a diameter of $ID_{main} - 2 * Proj$. Intersection of this surface with the branch defines the branch pattern, subject to the restriction that no part of the branch may project lower than the centerline of the main.

For the Fixed Butt (Groove) Weld detail (FastSHAPES standard type 1), ‘w’ is measured radially outward from the inner surface of branch or main, as applicable.

Path lengths are calculated at the relevant surface intersections of branch with main and branch with compensating plate, and may be used as a basis for estimating actual weld lengths.

Special Notes – Extreme Eccentricity

When a set-on branch is at its eccentricity extreme, or when branch and main have the same OD, the intersection of the branch and main outer surfaces occurs at a development ‘peak’ on a horizontal plane through the main axis. Such a

feature may or may not be acceptable, depending on the selected weld detail and the location of the relevant cutting paths. The compensation plate gap is measured from this intersection, and may result in mathematically correct but apparently excessive gaps in the compensating plate development, since it is assumed that the weld between branch and main is finished flush with the common branch / main outer surfaces.

This problem is mathematically simpler to deal with than is its specification of how to be dealt with it. No attempt is therefore made by PBranchO, and the user is left to edit the resultant patterns (using FastCAM or similar) to achieve the desired result local to the ‘peak’. In assemblies where the difference may be significant, the additional manual development time so spent can usually be readily justified.

The same problem tends not to exist with set-in branches unless the main thickness is very small relative to its OD.

A similar problem can arise as a result of misappreciation of the nature of the intersection of main and branch, or otherwise of simply slipshod drawing office practice. When main and branch have the same OD, or the eccentricity of the joint is close to its physical limit, it is not uncommon to find design or manufacturing detail drawings which effectively require a set-in detail at the crown of the main with a set-on detail at the cheek(s), or vice-versa.

If the root source of such a requirement cannot be rationally corrected, the real problem is then to determine how the transition between the two different details is to be effected, and is again simpler to do mathematically than it is to specify. Again, the user is left to edit the patterns manually, using FastCAM or similar. FastCAM (and most CAD software) allows two patterns to be superimposed to facilitate the task of defining a transition between the two.

PBranchS

PBranchS is intended for Structural applications. The standard edition provides flat cutting patterns or wrap-around templets for pipe branch elements which are set-on to a pipe main, for which only a wrap-around templet may be provided.

The wrap-around templet options permit specification of a thick external coating over which the templet would be wrapped, the templet containing data pertaining to the cutting paths required after removal of the coating. That is, the templet data is taken for the underlying pipe material and transferred to the outside of the coating. Templates are provided on an outside-up basis for plotting via FastCAM or any suitable CAD software, or for the NC cutting or manual lofting of sheet metal or other materials which may be used as a templet. Templet material is assumed to be of negligible thickness.

The flat cutting pattern for the branch is provided on the usual inside-up basis. Provision exists for the addition of green to the longitudinal seam edges to facilitate rolling.

The branch pipe OD and wall thickness are both required. The Main pipe OD is required, however, since only set-on branches are considered and the main is unpenetrated for Structural applications, wall thickness of the Main is not required.

The branch angle relative to the main may vary from a minimum of 15 degrees to a maximum of 90 degrees, and the longitudinal seam may be located anywhere around the circumference of the branch. The branch may be eccentric to the main.

A number of weld details are provided for, as described below. Some are similar to the jointing details discussed elsewhere, although they need more specific definition in the present instance. Differences in detail are best seen by switching between them, and using the viewing module.

General Purpose Butt (Groove) Weld

This has a constant specified gap, land (root face), and minimum included groove angle. It can be ‘made to fit’ a wide variety of joint configurations and standard specifications, and is best suited to joints not having extremely acute crotch angles. It is the equivalent of the Fixed Butt prep. used elsewhere; the depth w used in that case is similar in effect to the root face, but root face is taken in the present instance in the tangent plane, not in the radial square cut direction, and gap is now taken into account.

The root face is fixed in radial location, starting at the inside surface of the branch and running radially outwards tangentially to the surface of the main at the specified gap height above the main tangent plane. The weld groove angle is taken from the tangent plane in a view containing the local dihedral angle, and run to its intersection with the outer surface of the branch in all cases except where a natural groove angle formed by a square (radial) cut through the wall would provide a larger groove angle.

The data supporting the process of intersecting the outer surface of the branch with the weld groove cut line is clearly available to facilitate programming of NC equipment fitted with bevel cutting tools, although implementation of the data is heavily dependent on details of the programmability of the tool(s). For machines with the ability to rotate the beveling tool plane to follow normal to the path, the natural dihedral angle provides a basis for bevel cutting. For machines with bevel tool able to adjust in one only plane (typically, for some pipe cutting machines, adjustable in one only longitudinal / radial plane only), the bevel data can be transformed to that plane. The standard edition FastSHAPES does not support bevel cutting, however, the patterns adopt the latter interpretation and mark the groove face / outer surface intersection line on patterns. Points along this line will be found to be located on the pattern generators, apparent only when the manual lofting .PRN output is chosen, or a detailed analysis of the output undertaken as in FastCAM.

A total of three paths are marked on the pattern, groove face / outer surface intersection, groove face / root face intersection and root face / inner surface intersection. The cut path (square radial cut) follows the path leaving most material on the workpiece, hence secondary cutting of the preparation in at least the crotch area of the joint, using the other marked lines for guidance, is necessary before correct assembly and fitup can be obtained.

General Purpose Fillet Weld

‘Fillet’ weld is possibly a misnomer for this detail. The term is used principally because the preparation provided by Pbranch most closely resembles that used as a basis for a fillet weld preparation in many

standard specifications. It is able to accommodate acute crotch angles, and is the equivalent of the ‘safe’ square cut prep. used elsewhere.

This detail extends the outer surface of the branch to its intersection with the outer surface of the main, with any specified gap being applied radially to the main surface. The line of intersection of the branch inner surface with the main outer surface is similarly marked. The cutting path (square radial cut) follows the line of intersection which leaves most material in the branch workpiece. This is thus a ‘safe’ detail having a full (oblique) root face, in effect, enabling it to be converted by secondary cutting to literally any other detail required, including a fillet weld. Secondary cutting at least to the alternate marked intersection line is obviously required before the joint can be assembled. Note that this secondary cutting varies around the joint periphery from inner to outer face of the branch.

This detail prohibits extreme eccentricity of branch relative to main, or the use of identical OD main and branch. These are not practical configurations for this detail.

General Purpose Combination (Butt / Fillet or Groove/Fillet) Weld

This is the converse of the General Purpose Fillet Weld, and is the equivalent of the ‘unsafe’ square cut prep. used elsewhere. Its cutting path leaves least material in the branch workpiece. When cut square radially, no further metal needs to be removed to enable assembly of the joint which provides a feather edge fillet detail in the crotch area, and a feather edge groove detail opposite the crotch. At two intermediate points generally close to the crotch, there is a transition of the feather edge from outer to inner surface of the branch. The groove angle usually needs to be enlarged in many areas of the joint, and a suitable transition provided to the fillet weld in the crotch area.

The benefit of this detail is its economy. The disbenefits are the feather edge in the crotch area if heavy welding is required, and potentially deleterious effects of eccentricity of loading through the weld in this area. For larger branch / main angles, the latter problem tends not to exist.

Prequalified Partial Penetration Welds

Two are provided, as specified by AWS D1.1, and by the European Comite International pour le Development et l’etude de la Construction Tubulaire (CIDECT). They are similar to each other, having relatively minor differences. They are also in some ways similar to the General Purpose Combination Weld described above, save that in the crotch area when acute, the feather edge / fillet weld is eliminated in favor of a partial penetration groove weld. This preparation for acute crotch angles can be difficult to cut both automatically and manually, and almost inevitably requires hand grinding to finish.

Details of the preparation provided by Pbranch are similar to those for the General Purpose Butt (Groove) weld outlined earlier. There are no equivalents in the standard jointing details used for other FastSHAPES.

The ‘Designer’ Full Penetration Weld

This is based in AWS D1.1 full penetration weld detail, and reference should be made to this standard in order to get the best from Pbranch in this case. Terms used in Pbranch basically follow those defined in this standard. There are no equivalents in the standard jointing details used for other FastSHAPES.

AWS D1.1 defines permissible ranges of parameters (gap, root face, groove angle) within ranges of joint dihedral angle. This data is plotted graphically in a dialog accessible via a Pbranch command button.

The main chart plots a permissible groove angle envelope as a function of dihedral angle, and identifies the ranges labeled A, B, C, D as defined by the standard, together with transition zones A-B and C-D. The secondary chart shows the permissible root gap envelope as a function of groove angle, plus, for reference, the backup groove width.

A table is provided for entry of data over a range of dihedral angle covering the minimum and maximum applicable to the joint which are shown for reference at either end of the dihedral axis.

Data required is a set of dihedral angles (none identical) together with the required groove angle, root gap and root face. Pbranch will sort this data into sequence and plot it. Upon acceptance of the dialog form and request to develop the joint, Pbranch then traverses the joint, calculating the dihedral at each development interval, then looking up, with linear interpolation as necessary, the user's required groove angle, gap and root face. Details of the pattern and the 3 dimensional view of the preparation may then be studied, and data within the dialog adjusted as may be required.

The dialog provides for four sets of data to be held, any one of which may be selected by command button above the main chart. These are labeled Min, Sugg. Max. and As at Entry. The first 3 are samples to demonstrate the nature of the dialog. The fourth restores the user's data as it was when the dialog was last opened.

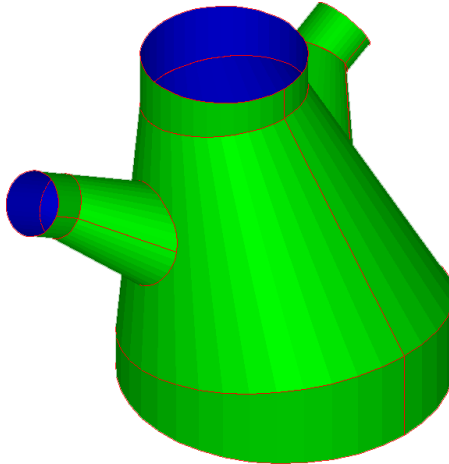
The dialog provides for the two classes of weld process defined by AWS D1.1, each having different permissible parameter ranges. Departure from those ranges will flag a data entry error in yellow or red cell background color, and may raise a further explanatory dialog.

A third option is provided which cancels all data checking against AWS permissibles, allowing the user to be guided but not bound by the AWS standard. This latter facility provides for sensible design of virtually any full penetration joint to any desired specification available to the experienced user.

The 3 dimensional viewing module and its shading facilities can be used to good effect in studying any of the above details. Note that because of the use of numerous flat facets to represent continuous smooth surfaces, when a small or zero root gap is specified, some of these facets may artificially intersect between branch and main, making the view difficult and or time consuming for FastSHAPES to resolve. It is recommended that the main be omitted from the view when shading is used, if difficulty is experienced.

Right Circular Conical Branches

CBranch



CBranch is a classical ductwork / blowpipe branched take-off, intended basically for medium rather than heavy weight construction. It is also designed to cover other situations such as penetrations or nozzles in conical hoppers, chutes and self-supporting conical roofs.

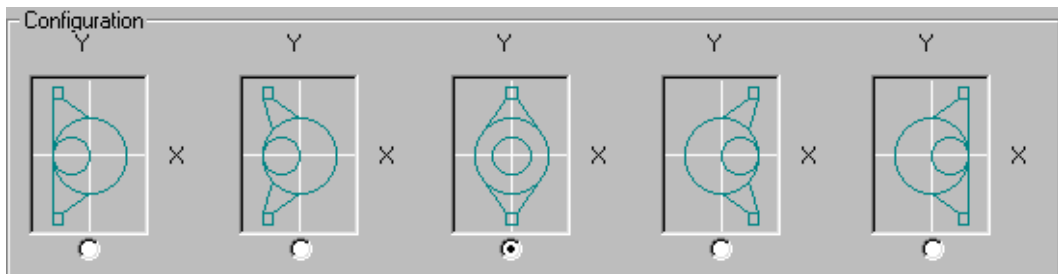
It is something of a hybrid in that, as an application, it fits more readily with the transformers and bends developed by triangulation. It uses their collaring and longitudinal seam location and jointing concepts for this reason, yet it uses radial line development, and some elements of its common central spheres setting out remain apparent to the user.

The setout starts with a right-circular conical body, which may be upright or offset to have a flat / vertical back (*west, or -X*) face or front (*east, or +X*) face. If upright, the base and top collar are optional. If offset, both these collars are mandatory. Longitudinal seams in both collars and body are confined to the four cardinal A,C,E and G locations. The A and E seams are always intersected by the XZ plane, but the locations of the C and G seams may be adjusted from exact +/- Y locations depending on branch details and body offset.

The height H is overall, and is inclusive of any base and top collars. It is the vertical height, not the sum of the axis lengths of an offset body

Either one or two branches may now optionally be added to the north (+Y) and / or south (-Y). Specification of branch data is always given, and displayed on the east elevation on the data entry form, for the actual or potential north (No. 1) branch; the south (No. 2) branch, whether actual or potential, is always set out opposite handed to the north branch. In the following, reference to ‘the branch’ means to the data given for the north branch, but can apply as appropriate to either branch.

In east elevation, the branch has no bend. In plan, the branch may or may not require a bend, as indicated in the five optional plan views indicating offset directions.



NOTE: If an offset main body setout is chosen, or if the branch body is non-cylindrical, a collar is mandatory. All collars are cylindrical.

Collars are sized using a common central spheres (CSS) setout, based on user’s specification of collar length as ‘**minimum**’, ‘**average**’ or ‘**axial**’. The ‘axial’ option directly locates the CSS on the collar axis, and automatically defines the bend plane inclination, and its location which, as discussed elsewhere, does not necessarily pass through the center of the CSS. The ‘minimum’ option causes CBranch to force the bend plane to pass through the lowest point of the collar, from which its inclination and the CSS location are then computed. The ‘average’ option causes CBranch to force the bend plane to pass through a point on the collar axis this distance from the end, then to compute the inclination and CSS location accordingly.

In the case of the main body; any specification other than ‘axial’ at both ends involves a complex calculation to locate the CSS at each end of the body. Unexpected results may appear from ‘just punching in a few numbers’ – it’s far more efficient to know what is needed from the outset!

Branch positioning and sizing is a little more complex. The branch is considered to spring from a sphere located somewhere in the main body cone. The position of the sphere center is defined by $(X0, Y0, Z0)$ and its diameter is $D0$. A sphere is chosen simply because it is definite as to position of its center and its effect on the definition of the branch body.

The branch length, L , is measured along the projection of the branch axis onto the YZ plane (as seen in east elevation) from the center of the base sphere to the end of the collar. The branch angle, G , (degrees) is the angle between the vertical Z axis and the branch axis projection, again as seen in the east elevation.

A number of possibilities for locating and sizing the branch base sphere present themselves as follows. There are subtle differences in the Auto-fit function, which should be noted.

1. MAIN BODY UPRIGHT

The branch base sphere may be generally located $(X0, Y0, Z0)$ and sized $(D0)$. Preferably, no part of the sphere should lie outside the main body cone internal surface, otherwise an error dialog may be issued when any development request is made. See note below.

If the Auto-fit box adjacent to $D0$ is checked, the $Y0$ data cell will disappear and the $D0$ cell become disabled from user data entry. Its value will show the diameter of a sphere automatically sized to touch the closest inside surface of the main body cone while its center remains at the specified $(X0, Z0)$ and with $Y0$ fixed at 0. If $X0=0$, the sphere will fill the main body cone.

2. MAIN BODY OFFSET, BRANCH BODY AND COLLAR OFFSET TO FULL ‘FLAT BACK’ CONFIGURATION

The branch base sphere may only be located by Z_0 . Y_0 is fixed at 0, and X_0 is determined automatically. The sphere always touches the upright side of the main body cone, and again should preferably not have any part external to the main body cone.

If the Auto-fit box is checked, the D_0 data cell is disabled and its value is set to the diameter of a sphere that will fill the main body cone.

3. MAIN BODY OFFSET, BRANCH COLLAR ONLY TO SAME OFFSET

The branch base sphere may lie generally at (X_0, Y_0, Z_0) and with diameter D_0 , again internal to the main body cone for preference.

If the Auto-fit box is checked, Y_0 is set to zero and its data cell disappears. X_0 is determined automatically by forcing the cell center onto the main body axis, and the sphere diameter is automatically sized to fill the main body cone. The height of its center remains at the specified Z_0 .

While location of the branch base sphere internal to the main body cone is preferable, it is essential that, wherever the branch base sphere is located, the branch body cone (extended as necessary) must intersect the main body cone. This requirement is most easily visualized and met when the sphere is fully contained within the body cone, and the branch angle set appropriately. Failure to meet this requirement raises an error dialog stating that any or all of X_0, Y_0, Z_0, D_0 are unacceptable.

CBranch also requires that the branch cone intersect the body cone only in the physically relevant area between the collars, and not in any extrapolated area outside this region. It similarly requires that the body cone intersect the branch cone and not the branch collar. Any non-compliance with this requirement leads to an ‘edge distance violation’ warning message. Development will proceed, but will not be useful as can readily be verified.

The minimum edge distance, J , defines a desired clearance in either the branch cone or the main body cone. Edge distance is measured along surface generators. In the body cone, it is the clearance between the circumferential collar joint and

the branch penetration, checked both above and below the branch. In the branch cone it is measured from the circumferential collar joint to the body cone. The default value is zero, but a larger value should generally be specified.



The actual edge distances are properties of the setting out, and the minimum edge distance is not a parameter used to define the setting out.

Branch longitudinal seam locations are restricted to A,C,E and G, as for the main body. The A seam is always on the east face. Seam locations in the no.1 branch correspond with those in the main body when the branch angle is zero, and retain the same relative positions at all branch angles.

For example:

If the no.1 branch has a branch angle of 90 degrees, the C seam is underneath the branch and the G seam on top.

There is no restriction on relative sizes at base and top of the main cone, or at base and end of the branch, that is either may taper in either direction. The branch angle is also unrestricted in range, although the data cell color shows a warning when outside the range +/- 90 degrees from a perpendicular to the main body cone side but does not, in this particular case, irrevocably prevent a development from being undertaken. The ‘clear’ range for G is thus when the branch is directed (base to end) away from the main body cone, or at worst parallel to the side slope. The ‘warn’ range is when the branch is directed towards the cone (from an external base)

The branch itself is always deemed to be the portion between the defined end of the branch (or its collar) and the main body cone. Any portion between the base sphere and the main body cone is ignored unless it lies between the main body cone and end of the branch.

Hence, the branch may be internal or external to the main body cone, depending on the location of the branch base sphere and the direction of the branch axis as defined by the branch angle, G. An internal branch may suffer an apparent inversion of its longitudinal seam locations.

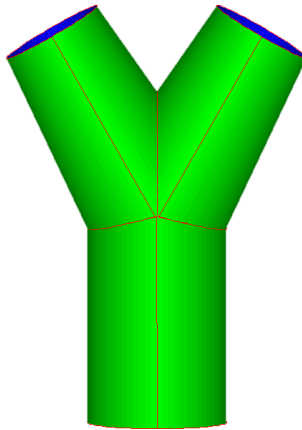
While the branches are set out with the “number-2” branch opposite handed to “number-1” branch. An asymmetric assembly can be made in two separate CBranch runs with the main body identical in each and main body cone longitudinal seams located in each case at A and E. Alternatively, patterns can be edited / overlaid using FastCAM®.

When the Auto-fit function is used for all but one of the layout possibilities described above, the base sphere becomes a common central sphere and the assembly becomes a fully-fledged, full-bore bifurcate (body plus 1 branch) or trifurcate (body plus 2 branches). Alternative examples of these can also be made with FastSHAPES Bifurcate and BifurcatePlus described elsewhere.

An extremely wide variety of different layouts and effects are possible, including ‘swapping’ the two branch positions through use of a negative Y0, and inversion of the branch longitudinal seam location definitions. Many of these effects can be subtle but significant.

It is recommended that both the 3 dimensional model and all patterns be studied carefully before adoption for manufacture, with particular care being taken with longitudinal seam locations in all elements of the assembly.

Bifurcate



This FastSHAPE can apply to any weight construction, including very heavy. However, because of setting out restrictions and limitation to a single material thickness throughout the assembly, it is generally more applicable to smaller, medium weight applications. Larger, heavier and more generally setout bifurcates are handled by BifurcatePlus

The radial line development method is used, with setout based on a common central sphere (CCS). The data entry form shows graphically and powerfully the basic features of the setting out.

The three branch axes are measured from the center of the CCS and are constrained to all lie in one common plane, the global XY plane, with the plan view being shown. There is no provision for individual branches to be subdivided by intermediate circumferential joints; each branch is a single complete cone, which may be subdivided only into segments by up to four longitudinal seams.

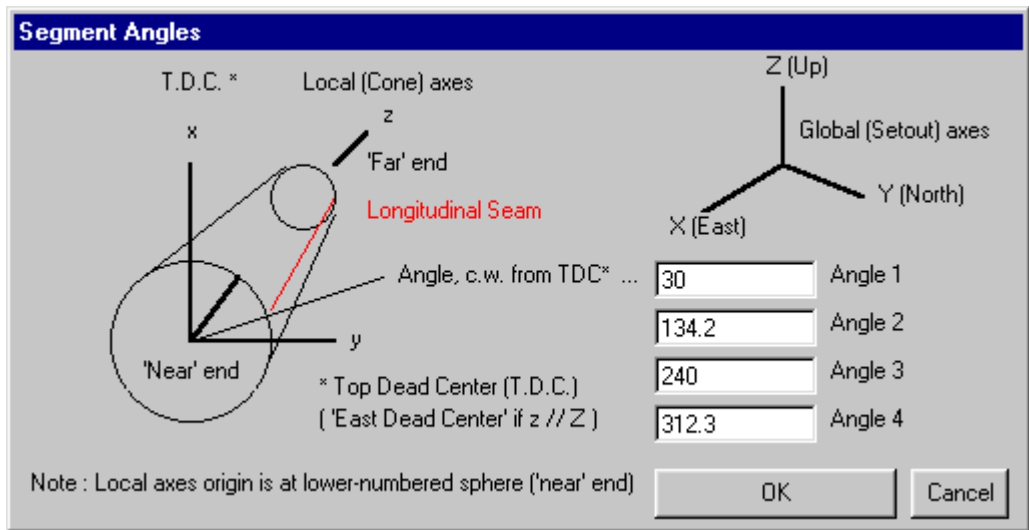
The branch axes are directed outwards from the CCS, and a local axis system (x,y,z) is established for each branch.

- The local z axis is the cone axis.
- The local x axis is vertically up (parallel to global Z), and
- The local y axis is horizontal.

From the viewpoint of an observer standing vertically in the CSS and looking out along the local z axis towards the end of a branch, x points to 12 o’clock, or Top-Dead-Center (TDC) while y points to 3 o’clock. The CCS end of the branch is seen as the ‘near’ end, and the outer end of the branch is seen as the ‘far’ end.

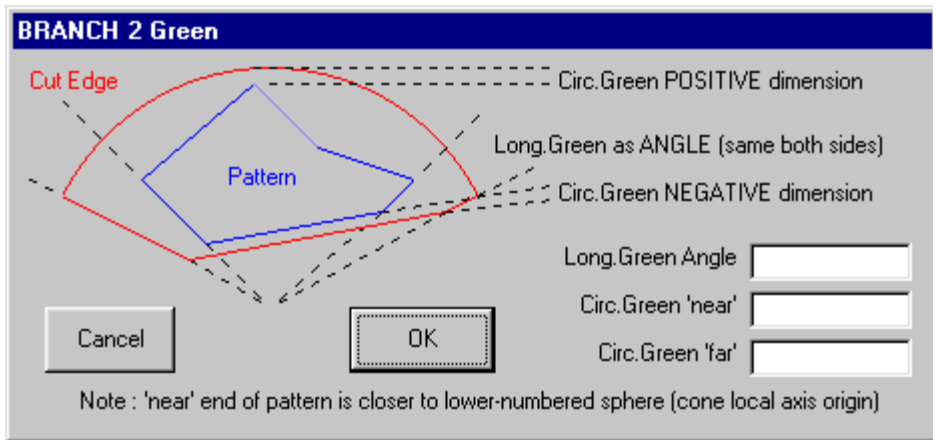
The radial line development sweeps its generator clockwise through 360 degrees, with 0 degrees at 12 o’clock, 90 degrees at 3 o’clock.. etc. Longitudinal seams in the branches are located by the sweep angle.

Up to four longitudinal seams may be specified in each branch. These may be entered directly in the relevant data cell (e.g. 30,134.2,240,312.3) or may be entered separately through a dialog opened by *double clicking* the data cell.



The branch cone is effectively developed by cutting along a longitudinal seam, say at TDC, unrolling it inside up, then lifting it flat into the global XY plane to represent its pattern. The pattern then has a ‘near’ and a ‘far’ circumferential edge, and a left and right (longitudinal seam) end, still as seen by the observer in the CSS. The sweep angles of the surface generators now have values increasing from right to left.

Green may be added (differently) to the near and far edges, and (equally) to the right and left ends. As with the longitudinal seam locations, green can be specified directly in the relevant data cells, or via a dialog opened by *double clicking* any of the relevant data cells.



End green is specified as a sweep angle, causing a wedge-shaped margin to be added to each end in the same manner as described for segmental green in FastSHAPE ‘Cone’.



NOTE: In that instance, a sweep angle was calculated from a specified arc length on the ‘inner’ edge. In the present instance, the sweep angle is specified directly in degrees.

Edge green may be specified either negatively or positively, as indicated in the dialog. The positive / negative acts as a flag to distinguish between the two types – regardless of sign, green always adds material to the edge of the workpiece.

- Negative green follows parallel to the edge.
- Positive green causes the cut boundary to follow a circular arc, which is kept clear of the most extreme point of the edge by the specified amount.
- Zero green leaves the edge unmodified.

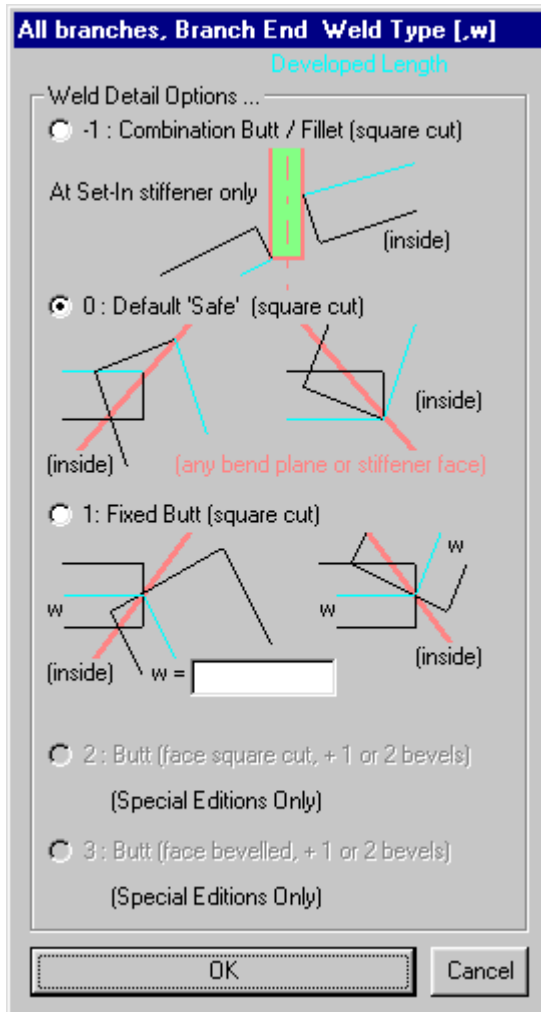
Negatively specified green is the equivalent of the edge green applied in FastSHAPES ‘Cone’ and for the same reason, as a field trimming margin, or possibly to allow for some edge damage if rolled in pyramid rolls with a cone rolling attachment.

It will usually be applied in this instance to the far end of the branch where the cone is cut square to its axis. However, in this particular situation, the effects of both negatively specified and positively specified green are identical – the edge is already a circular arc!

Positively specified green is more likely to be applied than negatively specified green at the near end, where, depending on branch angles in the setout and location of longitudinal seams, the edge can vary significantly from a circular arc. Generators (breaklines) can have significantly different lengths, causing variation in the formability over the length of the pattern. Application of positively specified green permits these problems to be overcome through making all generators the same length during forming.

No collars are provided, and the ends of the branches are dimensioned and detailed to fit directly to axially aligned pipe elements having the same internal diameters as those specified for the branches. That is, collars are implied and need not be specified.

Weld details at circumferential joints are provided for as described in detail elsewhere. One detail applies to all branch far ends, and one (possibly different) detail applies to all near ends. Data may be edited directly into the cell, or into a dialog opened by *double clicking* the data cell.



Patterns mark the weld details as described elsewhere, plus generators at 5 degree intervals. Bend match points on near and far edges are shown with an arrow. The TDC, BDC generators are identified by single / double triangular marks respectively.

A 3 dimensional plan view of the bifurcate amply illustrates the veracity of warning issued earlier concerning not using generators for matching cones adjoining each other through a bend.

Two forming templet lines are also marked. These are circular arcs when seen on the pattern, and appear as circles after the surface is formed. The PartsList table shows the radii of templets required to test the curvature at these two lines. The templets are required to be laid over at the indicated set, corresponding with the cone's half-apex angle. The two lines are as widely separated as possible on the pattern, subject to being located such as to produce rational numbers for the templet radii. Their location also uses as much green as has been made available.

BifurcatePlus

This FastSHAPE is a powerful extension of Bifurcate, The ‘Plus’es are

- crotchplane stiffeners, either set-in or set-on
- multi-strake cones (intermediate circ. seams) with different thickness’ in each
- general setting out, and not confined to a plane
branch bends / collars

The radial line development method is used, and setting out is directly and explicitly via the method of common central spheres (CCS)

Data is entered in four separate tables, with a layout of the setout spheres, discs and axes being able to be optionally updated as relevant data is entered.



NOTE: This feature can be disconcerting during bulk data entry and is switched off by default.

It can be usefully switched on if making minor changes after bulk data entry is complete. The layout is seen in plan and two elevations.

After all data has been entered in all tables, BifurcatePlus checks the setout and has certain requirements that are not apparent while entering data and are not indicated in the setout view adjacent to the setout table. Warning notices are issued when relevant.

One such requirement is that the bend plane / circ. joint between the branch cone and collar, (or the end of the branch cone if no collar is used), must be free and clear of the crotch plane. It must also be clear of the closer surface of any stiffener applied at that crotch plane. Section planes within the branch cone may however, intersect the stiffener face.

Basic Configuration data table

For each branch, numbered 1, 2 and 3, the following data is required

- number of sections (strakes) in the main body (minimum 1, maximum 4)
- number of sections (strakes) in the collar (minimum 0, maximum 2). Zero means collar is not required.
- end component, disk or sphere. A sphere is recommended at any time, and especially if FastSHAPE ‘Penstock’ will be used to extend the branch, using a bifurcate collar as a matching element. A disc or sphere provide the same result in the case of a cylindrical collar. At the end of a conical element, whether main branch body cone or collar, a disk will imply the presence of an axially aligned cylindrical element not included in the specification. BifurcatePlus inserts a sphere for its own use to achieve this end. Double clicking the data cell toggles between Disk and Sphere
- stiffener existence / type. Stiffeners are numbered according to the branch number they lie opposite. They thus lie between the ‘other two’ branches. Double clicking the cell opens a dialog which defines the two types provided for, plus the third option, Omit. Stiffener data may be entered at this time, at later.

Setting Out data table

A minimum of four and maximum of seven spheres are required for the setout. Sphere 0 is the CCS. Spheres 1, 2 and 3 define the ends of branch body cones 1, 2 and 3 respectively. Spheres 4, 5 and 6 locate the ends of collars attached to branches 1, 2 and 3 respectively.

Data required are the (X,Y,Z) global coordinates of the sphere centers, and their diameters which define the Internal diameters of the various elements. Where a disk may be specified in lieu of a sphere, it has negligible thickness and its center point is required.

It is recommended that the XYZ=East,North,Up convention be followed where possible. Discussion below assumes this, and appropriate reinterpretation may be required otherwise.

Stiffener Details data table

Double clicking any cell in a row whose number corresponds with the stiffener number opens the dialog, which graphically defines the data required. Data may also be entered directly into the table, but any change in type must be made through the dialog, and will also be reflected in the Basic Configuration table.

Even when the three branch cone axes are non-planar, the three crotch planes defined by the three pairs of cone intersections all intersect in one common line, termed here the ‘bifurcate axis’. The bifurcate axis provides the basis for setting out of the stiffeners.

The more appropriate type, 1 or 2, is not always self evident, and it may be necessary to try both types. The appropriate choice is often governed by how far out-of-plane the three main cone axes are, and any shape specified by the designer which is best approximated by which type. It may be necessary to use FastCAM to edit finer details of the stiffener patterns.

Either type may be set-in or set-on, relative to the adjoining branch shell plates.

When developed, stiffener thickness is equally disposed on each side of the crotch plane, and pattern marking shows both the bifurcate axis, and lines of surface intersection with the adjacent stiffeners. The top portion of the bifurcate axis is marked with a double triangle, to facilitate matching of all three.

Lines of intersection with adjoining branch cone shell plate internal surfaces, and the bend plane match point (arrow) are also marked for set-in stiffeners. Set-on stiffeners show marking for outer surface intersections of adjoining branch cones, and the line of intersection of these two surfaces at (or near, if the shells are of unequal thickness) the stiffener mid thickness.

It is not always possible to cut the edges of set-on stiffeners, which for a variety of reasons including this one tend to be confined to smaller size assemblies where the stiffener thickness can be reasonably contained.

Stiffeners are assumed to be made from the one class and specification of material. Thickness' of each may differ, although this is unusual in practice.

Branch Details table

All branch and collar cones are assumed to be made from the one class and specification of material. Thickness' may vary between branch body and collars, and between sections (strakes) of each, which is often necessary depending on variation in cone radius of curvature. When a single thickness is specified in the material dialog, this is the default thickness until edited over in the Branch Details table.

Cones are assigned a local axis system (x,y,z) in the same manner as previously described for Bifurcate. The collars continue and extend the same system, the observer looking out the ends. In this case however, the cone centerline axes (z) are not constrained to lie in a horizontal plane. We now use a notional vertical plane to cut the cone through its Top-Dead-Center (TDC) and Bottom-Dead-Center (BDC) generators, and the local x axis lies in this plane, perpendicular to the local z axis, and pointing to the TDC 12 o'clock position. The local y axis is perpendicular to the cutting plane and points horizontally to the 3 o'clock position. In the event that the cone local z axis may itself be vertical (parallel to global Z, and directed either up or down, local x and global X are set parallel and in the same direction, so that $y=\pm Y$. The 'TDC' position from which generator sweep angles are measured is still determined by the local x axis.

The table contains a column for each branch, and grouped sets of data in rows, progressing from the weld detail at the stiffener / crotch plane at the top of the table to the end of the collar at the bottom of the table. Each section of the body cone, then collar, is dealt with in turn, with the section plane data between.

Double clicking any cell in the crotch weld (first three) rows opens the weld detail dialog from which the required detail may be specified. It may also be directly edited into the cell. Because of the way stiffeners / crotch planes are numbered, stiffener 1 details are omitted for branch 1, etc.

For each section (strake), data required is Thickness (directly editable) plus the following 4 items, all as previously described for Bifurcate, all directly editable in the data cell, or in a dialog opened by double clicking the cell.

- segment angles, maximum 4 no.
- longitudinal (end) green angle
- circ. (edge) green, Near end
- circ. (edge) green, Far end

Section planes / joints data appear in two rows between the Section data groups. Section planes cut generally transversely through the cone and contain the circ. joint between adjacent sections. They are defined by distance to plane from local axis origin (sphere center), and optionally by one or two angles, all as indicated in the dialog opened by double clicking the data cell. The circ. joint weld details are standard as already discussed.

At the branch main cone / collar joint, the plane is predefined by the setting out parameters, and only the weld details are required. Similarly at the end of the collar.

After a development request is issued by the user (left or right click the layout diagram, select Display menu or PartsList tab), all data is comprehensively checked and developed if possible. Any data errors preventing development are shown as usual by data cell background color. Any layout errors preventing development are reported in a dialog.

Certain problems can arise during development which are reported as warnings in a manner similar to layout errors. The most common of these are the ‘split segment’ and ‘null segment’ warnings. Note that branch cone section planes / joints (but not the branch / collar joint) are permitted to intersect the stiffener face (or crotch plane if no stiffener is used). It is therefore possible for one or more sections of the branch cone to be incomplete. If no longitudinal seams are located in the ‘null’ area of the cone, a ‘split segment’ has been requested; some segment contains some irrelevant data. If two longitudinal seams are located in the null area, then the segment between them is totally irrelevant – it is all null. These features are easily verified by viewing the patterns. They may be quite acceptable

to the user prepared to edit the patterns in FastCAM, or who has an alternative means of disposing of the unwanted material that will be required to cut the parts as programmed by BifurcatePlus. The warnings are issued primarily to alert the user that any masses or areas shown in the PartsList table, nett masses particularly, are corrupted.

Because it is difficult for the user to predefine the null areas precisely, BifurcatePlus does this job. Locating a single longitudinal seam anywhere generally in the null area effectively instructs BifurcatePlus to ignore it, and to terminate patterns of adjacent segments when they intersect the stiffener face. A small amount of trial and error may be involved in getting all longitudinal seams located for best results.

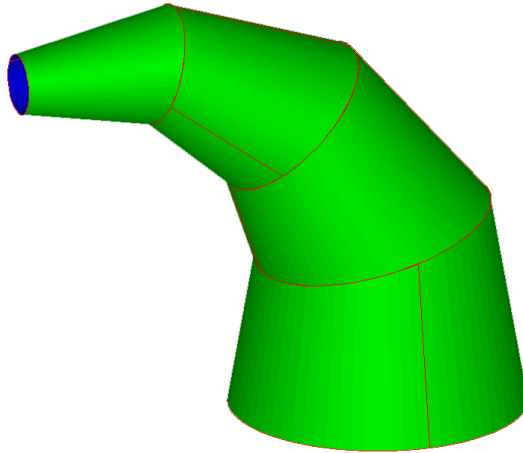
Also note that it is possible for patterns to contain two weld details on their near edges, one for the circ joint to the adjoining section, and one for the circ. joint to the stiffener. There may also be two instances of the latter, replacing the usual pattern ‘ends’ at long. Seams.

Patterns otherwise contain all the features described elsewhere for Bifurcate. The locations of the bend match points (arrows) potentially assume greater relevance in this instance, and correspond with those also shown on the stiffeners.

The general warning concerning not using generators as match lines through a bend is reiterated. However, because there is no bend at a section plane, the generators may provide useful assistance at these locations only.

Bends

Penstock



This FastSHAPE is designed specifically as a companion to BifurcatePlus. It is best suited to handle the distributor pipework systems often associated with connections between bifurcations, valve chambers, turbines, and the longer-run penstocks feeding these systems. Distributor systems are frequently characterized by tight setouts and numerous closely spaced bends, and are common in water supply, sewerage reticulation and hydroelectric power generation industries. ‘Penstock’ in this sense is not a fully comprehensive name for this FastSHAPE; a further companion Prun is sometimes better suited to the longer-run penstock system proper, more often characterized by fewer and less pronounced bends in constant diameter pipework.

Penstock is set out on the same Common Central Spheres (CCS) basis as a single branch of BifurcatePlus, it uses identical concepts, and produces patterns having identical features. The branch collars in BifurcatePlus provide the linking element between these two FastSHAPES, and no more needs to be said for the benefit of users having any real need of either or both.

Penstock also provides a powerful general purpose facility for a long sequence of cones and / or cylinders in any ductwork or pipework in any weight construction

from sheet metal to very heavy plate. Some typical examples are discussed at the end of this section.

A maximum of 50 cones may be defined, and each may be subdivided into a maximum of two sections or strakes through the insertion of a section plane (circ. joint). Each section may be further subdivided into up to four segments through the insertion of longitudinal seams.

The setout is through the CCS method, with spheres being numbered from 0 to N, where N=number of cones. Any cone ‘n’ then runs from a Near end starting at sphere ‘n-1’, and extending to a Far end at sphere ‘n’. The start at ‘0’ and end at ‘N’ may be defined as disks rather than spheres. The existence of cylindrical coaxial collars is then implied, having internal diameter of the specified size. ‘Penstock’ then inserts actual spheres hidden from the user, and these are sized and located to produce the specified result.

The spheres are set out in the global (X,Y,Z) = (East,North,Up) system, with their diameters defining the internal diameters of the assembly. End disk(s) when specified are infinitesimally thick with center at X,Y,Z.

The layout of the spheres is shown in plan and two elevations in a diagram adjacent to the setout data table. Auto-checking and updating of the diagram may be switched on or off



NOTE: The default is off.

Each cone ‘n’ is assigned a local axis system x,y,z in which z lies along the cone centerline axis from sphere ‘n-1’ center to sphere ‘n’ center. As in BifurcatePlus, the local x axis is then arranged to intersect the TDC cone generator at a sweep angle of 0 degrees (12 o’clock) with the y axis pointing to 3 o’clock, as seen by an observer standing in sphere ‘n-1’ and looking towards sphere ‘n’. In the event that the local z axis is parallel to the global Z axis, in either an upwards (+Z) or downwards (-Z) sense, the local x axis points east (+X) and the local y axis points either north(+Y) or south (-Y)

Details of the Penstock cones are entered in a single-column table in data groups representing individual sections separated by section plane / joint weld detail and

bend joint data, the bend planes being automatically defined by the setout parameters (X,Y,Z and D) for each sphere.

For each section (strake), data required is Thickness (directly editable) plus the following 4 items, all as previously described for Bifurcate, all directly editable in the data cell, or in a dialog opened by double clicking the cell.

- segment angles, maximum 4 no.
- longitudinal (end) green angle
- circ. (edge) green, Near end
- circ. (edge) green, Far end

Section planes / joints data appear in two rows between the Section data groups. Section planes cut generally transversely through the cone and contain the circ. joint between adjacent sections. They are defined by distance to plane from local axis origin (sphere center), and optionally by one or two angles, all as indicated in the dialog opened by double clicking the data cell. The circ. joint weld details are standard as already discussed.

Patterns are again, as in Bifurcate and BifurcatePlus, described as having a near and far circumferential edge, and right and left longitudinal end, and show the following features:

- Patterns mark the weld details as described elsewhere, plus generators at 5 degree intervals.
- Bend match points on near and far edges are shown with an arrow. The TDC, BDC generators are identified by single / double triangular marks respectively. (Three dimensional views of the penstock amply illustrate the veracity of the warning issued earlier concerning not using generators for matching cones adjoining each other through a bend).
- Two forming templet lines are also marked. These are circular arcs when seen on the pattern, and appear as circles after the surface is formed. The PartsList table shows the radii of templates required to test the curvature at these two lines. The templates are required to be laid over at the indicated set, corresponding with the cone’s half-apex

angle. The two lines are as widely separated as possible on the pattern, subject to being located such as to produce rational numbers for the templet radii. Their location also uses as much green as has been made available.

Examples of possible applications of Penstock other than as distributor systems in water / sewerage / hydro-electric power systems are to be found throughout industry whenever in-plane or out-of-plane bends are required in circular cross-section ducting or pipework. This can include such amusement structures as water slides and the like.

FastSHAPES Lobster and Elbow also provide solutions for in-plane bends in medium weight ducting which contain a feature not available in Penstock, while FastSHAPE Bend provides solutions for in-plane bends having circular or general ‘rectircular’ cross-sections. That feature is the Autonest, which is discussed in detail with those FastSHAPES where cross-reference is made to Penstock in relation to a tapered lobsterback bend.

A tapered lobsterback bend can be set out in Penstock using a sequence of uniformly decreasing diameter spheres placed at uniform intervals along a circular arc. Chords of equal length connecting the sphere centers then define the centerline axes of the cones. Other than in the degenerate case when all spheres have the same diameter, the bend planes at each sphere containing the intersections of adjacent cones do not pass through the sphere centers, and the axis lengths of the cones between bend planes vary.

If full cones are made up but not joined circumferentially, they can be assembled to form one complete right circular cone after rotating every second cone 180 degrees around its axis. The development of such a cone is simple (see FastSHAPES ‘Cone’ and ‘Dr.Cone’ for example), and all the individual cone elements are perfectly nested (ignoring thickness considerations) within a segment of a flat annular ring.

The converse expression of this is that a right circular cone, when cut correctly, can be re-assembled to form a tapered lobsterback bend which complies with the CCS setout as described. Any one longitudinal cut made in the original full cone is then seen as longitudinal seams alternating 0/180 degrees in alternate cones

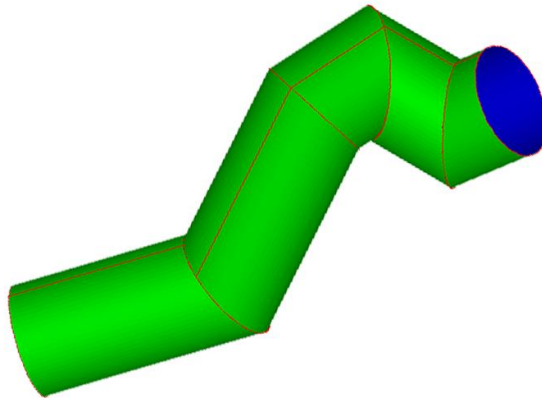
within the bend. This is the basis of the Autonest facility in FastSHAPES ‘Bend’, Lobster, and ‘Elbow’.

An autonest facility is not offered in Penstock, which is designed to handle more generalized setting out, variable thickness components, and specific circ. weld details, any or all of which would prevent a true autonest from being developed. However, if the setout described above is followed:

- all components are assigned the same thickness,
- all circ. welds taken as fixed butt (groove) with root depth at half the material thickness, and
- longitudinal seams appropriately alternated,

Then it will be found that the individual patterns can be laid out and nested in the manner described for the autonest.

Prun



Technically, Prun is a contraction of, and companion to Penstock, with a few added features to provide companionship with FastSHAPES; Pbranch and Elbow, plus a sister program FastFRAME.

FastFRAME is a standalone program, which develops the ends of members in tubular structural 3-dimensional trusswork. A truss basically comprises two or more chords (which in FastFRAME may be straight, cranked or curved), and ‘web’ members connecting between setout points along the chords, at which points the chords may be cranked, or bent. The web member axes may pass through the chord setout points, or may be eccentric to them. FastFRAME is principally concerned with developing the ends of the web members where they intersect with the chords and with other web members. Prun provides a consistent development of the chords.

Where in Penstock a common central spheres (CCS) setout was used explicitly, with sphere diameters defining internal diameters of the penstock, Prun now uses CCS setout implicitly, and the constant diameter of all pipes in the system is defined as outside diameter, which is consistent with both FastFRAME and Pbranch. The distinction between inside and outside diameters is usually a function of method of manufacture of the pipes.

Prun requires first that the axis of the piperun be defined, with start, end, and all intermediate bend point being assigned an (X,Y,Z) coordinate in the global setout

system. It is again recommended that where possible the East/North/Up convention be followed.

The outside diameter and wall thickness of all pipes in the run is required, plus the required circ. weld detail to be applied at each join. This is again able, to be directly edited in the data cell, or in a dialog opened by double clicking the data cell.

As in Penstock, each pipe in the run has its own local coordinate system for the purposes of defining longitudinal seam locations. The TopDeadCenter (TDC) generator (or East face generator if the axis is vertical) is located at sweep angle 0 degrees, and sweep angle is measured clockwise when looking along the axis from ‘near’ (lower bend point number) to ‘far’ (higher bend point number) end of the section of pipe.

There are two development options, wrap-around templets with optional coating thickness, similar to Pbranch, and patterns for flat cutting, as for Penstock.

The wrap-around templet option requires only the optional external coating thickness, and an optional templet length. The templet is always cut longitudinally at TDC, and its girth includes any coating. Marking is on the outside of the templet. The ends defines the lines at which the pipe should be cut radially square to enable production of the specified circ. weld preparation using a secondary cutting or grinding process. Both ends of the pipe section are placed on the one templet to ensure that the cuts are made at the correct circumferential offsets. Bend plane match points are marked with an arrow.

When the templet length option is unchecked, the templet length is full size. When the templet length option is checked, the templet is compressed or expanded to the stated length (usually corresponding with the size of a plotter) while retaining true scale at each end.

The PartsList table shows the length of the templet as selected, shows material as TempletOnly, and no masses. Because pattern sizes shown in the PartsList table (enclosing rectangle size) normally include marking, and in this instance the PartsList Length data may be required to define actual nett cut pipe lengths, the outer surface circumferential marking is omitted.

The flat cutting option produces patterns on the usual inside-up basis with girth at the correct developed circumference and detail otherwise as described for the wrap-around templet. The TDC and BDC generators are also marked. Data required are Long.Seam Base Angle, Long.Seam Offset Angle, Longitudinal Green Angle, Circ. Green Length, Minimum Stripped Plate Width and Maximum Stripped Plate Width.

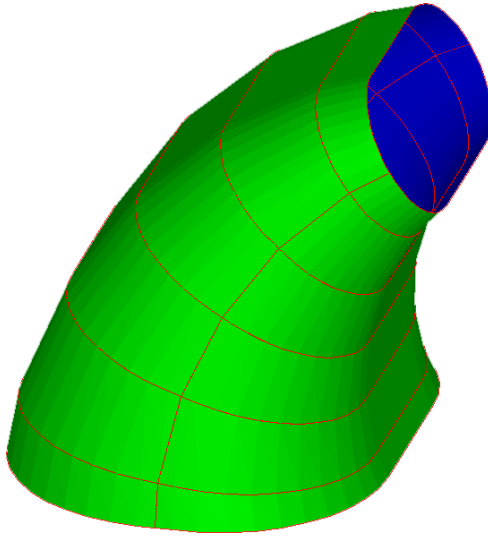
Prun takes the min and max stripped plate widths and first allocates circumferential joint locations between bends over the length of the run to divide each pipe into sections. The longitudinal seam locations are then fixed, taking Base+Offset as location in the first section, Base-Offset in the second, and so on along the run.

If a pipe between two bends requires only one section, both miter cuts are programmed in the one pattern. If two or more sections are required, the end sections are programmed separately. Any intermediate sections required to make up the full length programmed and displayed as though made up from a single plate having width equal to the length required. This is not a wholly useful pattern for production purposes, but in this case there is little production difficulty in any event since standard width barrels are readily made up from the max. stripped width plate, and any required make-up sections readily made similarly. The 3 dimensional display shows only one longitudinal seam for the central length, this being for its first section so the seam offset is always seen correctly in the display at the near end. At the far end, the seam offset may or may not appear correctly depending on whether the central length contains an odd or even number of sections.

The PartsList table shows the required lengths of the end sections, plus the total length of the central length. The PartName for the central length also indicates the number of standard length (max. stripped plate width) barrels required, with any fractional part representing a make-up shorter-than-standard length section.

Prun attempts to eliminate make-up sections in central lengths by adjusting end section lengths where possible between the two extreme stripped plate width specified, and having regard to the miter angles at the bends.

Bend

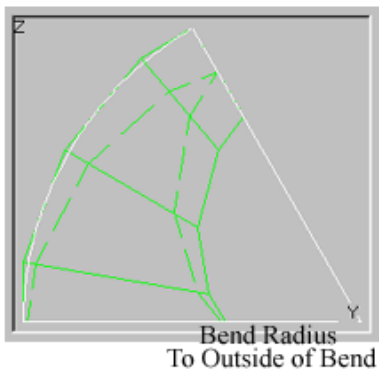
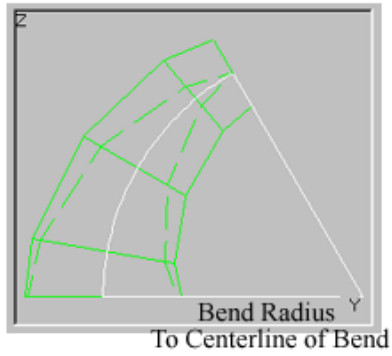
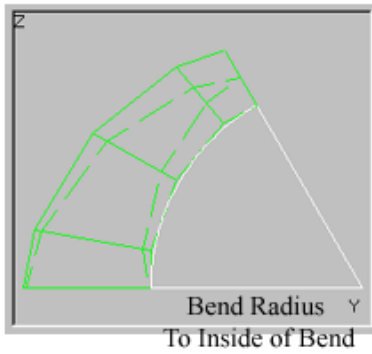


This FastSHAPE provides a general solution for in-plane gored bends in medium weight materials of constant thickness.

The cross-section is defined as a general rectircle, or filleted-corner rectangle, thereby providing through appropriate setting of parameters for rectangles, obrounds and circles. Cross-sections at each end of the bend may be of different size and shape, thereby providing for transforming bends.

The plane of the bend lies in the global YZ plane, Z vertical up, Y north, and viewed in east (X) elevation. The center of curvature of the bend is located on the Y axis at a nominated radius north of the inside north face, center, or inside south face, of the base. That is, the constant radius may be defined to describe the inside, center, or outside of the bend. The bend angle is measured between radii at each end of the bend, and may be specified within a range from 1 to 540 degrees.

A maximum of 20 gores may be specified. The minimum number of gores is not less than 2, and may be higher depending on other details of the setting out.



'BEND' SETOUTS

Gores may be set out in one of three ways, as described below. In the first two, planes between adjacent gores which contain the circ. joints, when extended, intersect in a common line parallel to the X axis and passing through the center of curvature of the bend. In the third, this may or may not be the case.

1. half-angle gores at each end, full-angle gores between
2. equal-angle gores throughout
3. gore angles, lengths and other details automatically determined by Bend

Case 1 is the most common, especially for prismatic bends (constant cross-section size and shape). The centerline axes of the end gores of prismatic bends are perpendicular to the end planes, hence any collars (which are always prismatic) may be integral with the end gores. This (half-angle end gore AND prismatic bend) is the only case in which integral collars are possible – in all other cases, a collar must be attached via a circ. joint to the end gore and is developed as a separate item.

Case 2 is common for transforming bends. The ‘Eschenberg’ bend is a typical example, often found as ventilation shrouds over mine shafts, or draft tubes under turbines, and providing transformation from a rectangular base to a circular top, or vice versa. Regardless of the bend radius location, the centerline axes of the end gores are never perpendicular to the end planes.

Case 3 in certain circumstances, leads to the Auto-nest development discussed at the end of the section on ‘Penstock’. The circumstances are

- prismatic bends of any cross-sectional shape, regardless of bend radius location. This reverts to Case 1
- circular cross-sectional shapes with differing diameters at base and top, when the bend radius is at the centerline axis of the bend. This is the exact case discussed in ‘Penstock’
- obround cross-sectional shapes which can be envisaged initially as the above case, which is then cut through the plane of the bend, and separated by constant width strips inserted on the inside and outside of the bend

If a request is made for Bend to auto-size/locate gores and the bend complies with the above, the Autonest option is enabled and can be undertaken using the radial

line method. If the bend does not comply with the above, the Autonest option is disabled and only development by triangulation (compensated) is possible.

When Autonest is enabled but not selected, triangulation is used, and the 3 dimensional view is enabled. Both the patterns and the 3 dimensional view will indicate non-existent warp in the surfaces (unless the bend is prismatic, when there is no warp, and none will be indicated). The reason for this is that unwarped surfaces of cones have generators that do not generally correspond at the bend plane of generating curve between the two surfaces, as discussed elsewhere. Using the triangulation development forces coincidence between points which lie in different places on two otherwise identical generating curves, and thereby incorrectly cause surface breaklines to indicate warping. This is discussed later in the section on free-from chined surfaces.

When Autonest is enabled and selected, Bend develops all patterns in the Autonest format described in the ‘Penstock’ section, including longitudinal green margins, and using cuts with no kerf compensation between the adjacent gores. No 3 dimensional model may be viewed in this instance. (All 3 dimensional models provided by Bend are based on the triangulation development) Longitudinal seams for the Autonest may be located at standard A,C,E and G positions only and are fixed in the base gore. Only 1, 2 or all 4 seams may be specified, and if 2 are specified they must be diametrically opposite (AE or CG). The purpose of these restrictions is to enable alternate segments to appear in alternate patterns.

Use of 1 only longitudinal seam automatically provides for offsetting of seams (by 180 degrees) in the final assembly.

Use of 2 or 4 seams will retain alignment of seams after rotation of alternate gores by 180 degrees. So if offsets are required, this needs to be specified together with any green, and will appear on the edges of the patterns with the green beyond the area containing the intermediate circ. joint cuts. Green and offset in this case follows the standard for triangulation as described further below, not for radial line development as used in Penstock and elsewhere.

When only 2 seams are needed, a further alternative for offsetting them is to take 2 runs of Bend, using the same data. For run 1, use seams at AE and take odd numbered gores. For run 2, use seams at CG and take even numbered gores.



NOTE: *For further discussion on the Autonest concept, see the section on ‘Penstock’ and ‘Autonest’ in Chapter 1. The remainder of this section deals only with the triangulation development for the general bend.*

Cross-sectional shapes between gores are interpolated between the specified base and top shapes and locations according to the specified bend radius location and gore-angle apportionment. All such intermediate cross-sections may also be defined as general ‘rectircles’. Allocation of development points around the rectircular peripheries for development by triangulation is a simple and well-known task.

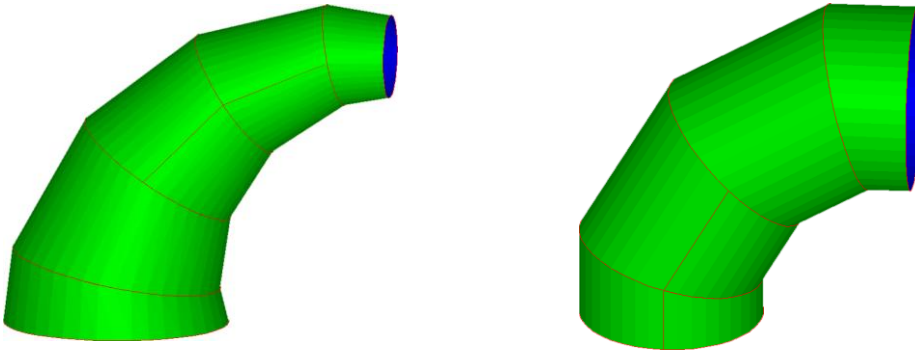
Longitudinal seams may be located at any of the 8 standard locations ABCDEFGH. From the perspective of an observer looking through the bend from base to top, gores will have a near and far end (circ. joint) and a right and left end between any two long. Seams. When laid out as patterns, inside up, the near / far edges and left / right ends are retained. A constant amount of green may be added to both the left and right edges of all patterns. Any long. Seam offset dimension is added to right end green and deducted from left end green in odd numbered gores, and the converse in even numbered gores. The effect can be reversed by specifying a negative offset dimension.

Unless all 8 seam locations are required because of the size of the bend relative to available stock-sized plates, a superior alternative to using seam offsets in this manner is to take alternate gores from two runs in the manner described above for the Autonest.

A feature often noticed in the patterns of the general rectircular bend is warp in quadrilateral-shaped cheek plates (side panels, as seen directly in east elevation). This warp is virtually unavoidable, and results in negative breaklines being required, as seen in the patterns. When long. Seams are located at both A and E, the quadrilaterals are approximately halved, and the effect is mitigated through a long. seam departure from flatness. If a seam is not specified at one of these two locations, the full panel could be broken once across one diagonal. Bend arranges for the cheek panels to be at least symmetrically treated, regardless of seam location.

Warp may also be observed in the corner quadrant areas. These are not and cannot be portions of an oblique cone. They may be portions of a right-circular cone if the Autonest feature is possible. Even in the latter case, when triangulation is used the setout obscures the fact that they may not necessarily be warped.

Lobster and Elbow



These are cut-down versions of Bend, with the cross-sectional shape confined to a circle, now described by internal diameter.

The purpose is to facilitate and expedite data entry in many common situations.

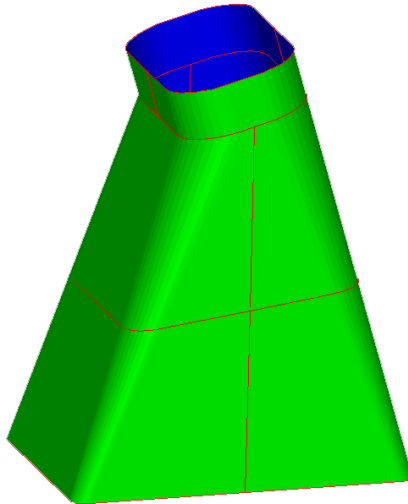
Lobster may have different sized circles at each end (a tapered lobsterback). Providing the bend radius is specified at bend centerline, and the gore ‘auto-size/locate’ option is taken, the patterns are always Auto-nestable.

Elbow is confined to equal sized circles at each end (i.e. the elbow is prismatic), and regardless of bend radius location patterns are always Auto-nestable provided the gore ‘equal-angle’ option is not selected.

See sections on Bend and Penstock for further discussion.

Transformers

Rectircle



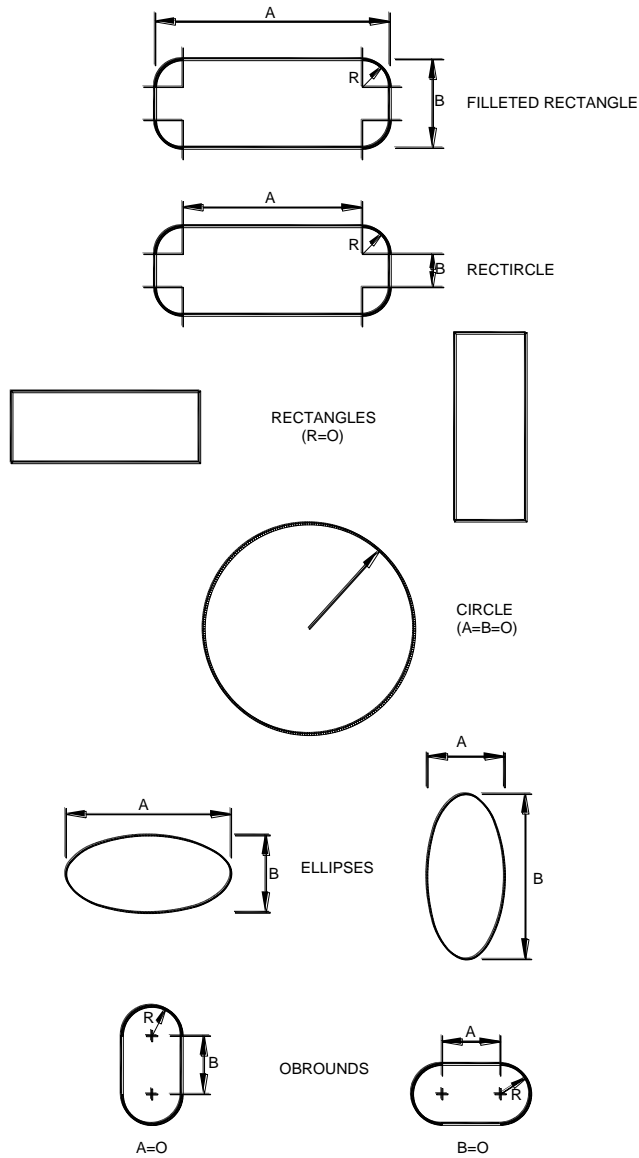
This FastSHAPE provides a general solution for the ‘almost anything’ to ‘almost anything’ transformer in medium weight materials of constant thickness.

The cross-section is defined as a general rectircle, or filleted-corner rectangle, thereby providing through appropriate setting of parameters for rectangles, obrounds and circles. Cross-sections at each end of the transformer may be of different size and shape, and the notional or actual ducts connected by the transformer may have any relative position, twist or tilt in their axes.

One of the essential features of the setout of the transformer to be born in mind is that the base and top shapes are defined in terms of the internal sizes of prismatic sections of ductwork that the transformer would connect between. Collars, when used, are in effect short portions of that connecting ductwork. This concept avoids the mental gearshift otherwise required to deal with thickness issues on an oblique (or any other) cut plane through the body of the transformer.

Collars are optional, and when used may be parallel or mitered, the latter being the more useful; a parallel collar really adds little to the definition of the transformer other than to reinforce the concept of the end shape definition given

above. Any adjoining section of ductwork, whether in concept or in fact, is assumed to be cut square to its axis, so that any perpendicular taken from the defined end planes and through the center points directly along the adjoining ductwork axis.

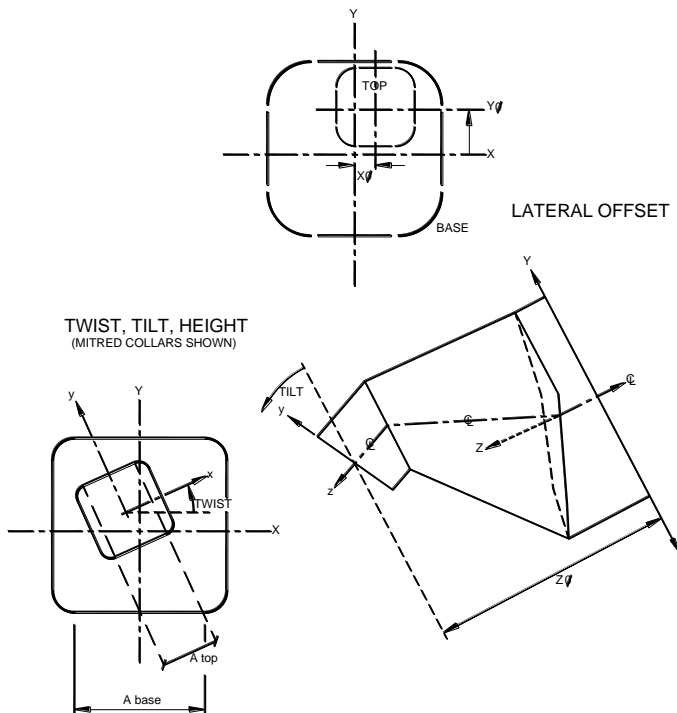


Standard Rectircle Setout Options

The transformer is set out with its base shape in the XY (East/North) plane, with the straight portions A and B of the rectircle being parallel to X and Y respectively. The centerline axis of the duct at the base is the global Z axis.

The overall height, Z0, of the transformer is measured vertically from the center of the base shape to the center of the top shape, inclusive of any collars specified. The locations X0 and Y0 of the center of the top shape are its lateral offsets in the X and Y directions from the Z axis.

The questions of twist and tilt of the top duct relative to the base duct next need to be addressed,. It is critical that they be addressed in the correct manner, and in the correct sequence. The following discussion assumes that both top and bottom ducts are of general rectircular cross-section, each with flat side portions, with or



without radiused corners. Circular ducts are discussed later.

We first address the twist, and need to consider two situations, (1) when the top duct centerline axis is parallel to the base duct centerline axis, and (2) when it is not.

When the top duct centerline axis is parallel to the top duct centerline axis, the top duct may have only a twist relative to the base duct; there is no tilt. The top duct shape is readily projected onto the XY plane, and it's A and B rectangular sizes ascertained, with the A side being the more closely parallel to the X axis. The twist is then the angle from the base A side to the top A side, being measured counterclockwise positive. (The alternative of switching the, A and B in the top and defining a larger twist is possible but not recommended – minimum twist is best).

When the top duct centerline axis is not parallel to the base duct centerline axis, we project the top duct centerline axis onto the XY base plane. The twist angle is the angle from the X axis to the projected top duct centerline axis, measured counterclockwise positive. If the twist angle so obtained is closer to +/- 90 degrees than it is to 0, than we should reverse our definitions of A and B in the base duct, and repeat the exercise until the twist is minimized. (If it was originally +/- 45 degrees, we cannot do much about it, and either way is as good - or as bad - as the other)

Next, take a vertical plane containing the top duct centerline axis, and ‘stand square on’ to this plane to obtain a true view of the top duct. The top duct axis will be seen to be inclined at an angle to the global Z axis, and this is the required tilt angle, measured counterclockwise positive from Z.

If the two flat sides of the top duct lie in vertical planes parallel to the top duct axis plane, then the setout can be handled by Rectircle. (If not, then FastSHAPE Surface will be needed to create the required transformer.) A corollary of requiring that the flat sides of the top duct lie in vertical planes is that the top rectangular shape will always have the B portions of the shape as lines parallel to the base XY plane, and any tilt in the top axis will only tilt the A portions of the shape relative to the base XY plane.

We have now defined the ‘hole’ between two portions of notional or actual ductwork into which the transformer must fit, and have assumed that both ducts have been cut off square to their axes.

As stated earlier, use of collars is optional. They might be omitted if the space Z0 is very tight, but they are generally recommended – they provide a logical place to allow at least a field trim margin for final fit-up.

Space permitting, a mitered collar is generally preferable to a parallel collar. The collar length may be specified as minimum or average, these amounting to the same thing for a parallel collar. Using a practical minimum puts the most material into the body of the transformer, and results in the best possible product flow conditions through the bends at each end of the transformer, subject to any flow limitations imposed by the cross-sectional shapes themselves

The main transformer body may be dissected into up to 10 strakes. When the bend planes (containing the circ. joints) at each end of the body are extended, they either meet at infinity (i.e. do not meet - the bend planes are parallel) or they meet in a line. If the bend planes are parallel, the strake is based on the specified percentage of the perpendicular distance between the two planes. If the bend planes are not parallel, the strake size is based on the specified percentages of the angle between the two planes.

Each collar, and each body strake, may be further dissected into segments defined by up to 8 long. seams defined by the standard positions at ABCDEFGH. At the top end, these positions are twisted together with any twist in the top duct.

Green may be added to the long. ends of each pattern, +/- a specified amount of Long.Seam Offset, generally as described for Bend, with which Rectircle is most nearly equivalent.

One final option presents itself: the **Setout Type**. Options are:

- ‘**Most Appropriate**’,
- ‘**Simple**’ and
- ‘**Exact**’.

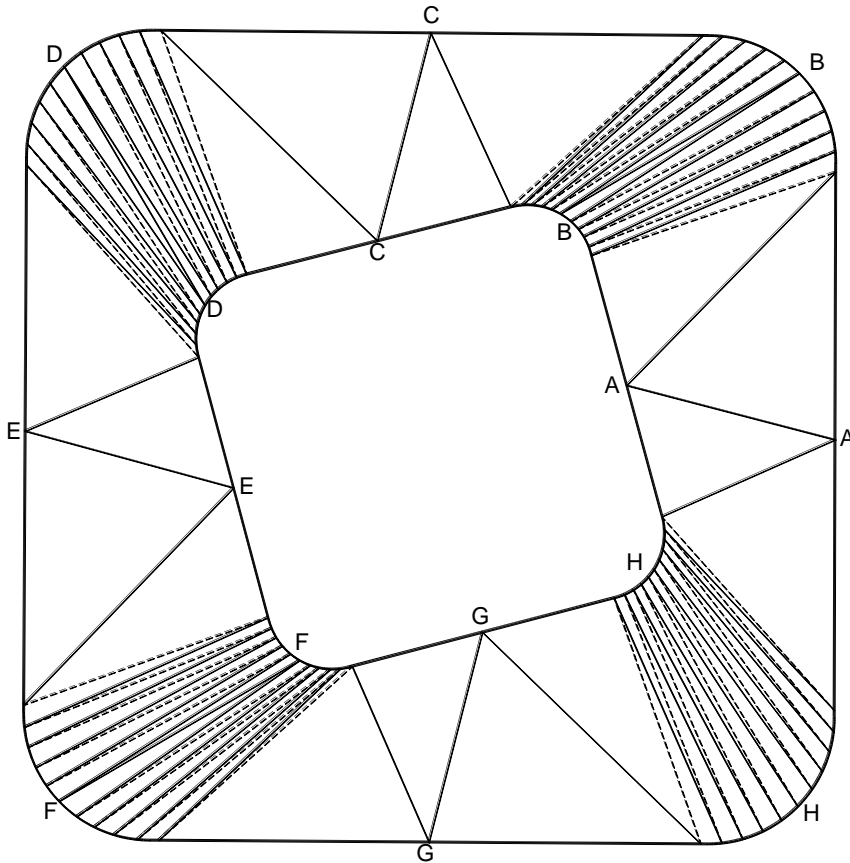
‘**Most Appropriate**’ is the easy option; Rectircle will decide. However, its decision can only be one of the two following:

- The ‘**Simple**’ development is based on regularly spaced development points set out around the periphery of each circ. joint.
- The ‘**Exact**’ development really means ‘best possible’ rather than ‘absolutely exact’, but the distinction may be more academic than practical. To understand the difference, we need to examine the setting out in more detail, and its impact on development using triangulation.

Suppose we start out with two parallel rectircular shapes (the ends of the actual or notional ducts, cut off square to define the ‘hole’ into which the transformer is to fit). Assume collars are omitted, or are parallel collars, and that there is no relative twist between the two ends.

Placing equally spaced points around the quadrant corners poses no problems; the surfaces in these areas are portions of oblique cones, they are not warped, and their development leads to reasonable patterns. The quadrilateral-shaped side panels are in fact flat, and their development shows no warp.

This is the basis of the ‘**Simple**’ option.



'RECTIRCLE' - 'SIMPLE' SETOUT

Next, assume a tilt of the top relative to the base, but still no twist. The body now assumes the same characteristics of a Bend body gore as discussed elsewhere. The corner quadrants no longer remain portions of oblique cones, and two of the four quadrilateral side panels no longer remain flat. (The A side panels are warped, the B side panels remain flat.) Six of the eight distinct components of the body have become warped. Negative breaklines in patterns will scarcely be surprising, and the greater the tilt, the less the surprise. We could, with vast futility, play around forever with repositioning our development points on the base and top rectircles without ever altering the fact that the surfaces are warped. The ‘simple’ option is bad, but nothing else can improve on it!

Next, forget the tilt. The end planes are parallel, and we will apply a twist. If we leave our ‘simple’ development points in fixed positions, equally spaced around the quadrants in the corners of the rectircles, we will introduce some dramatic distortions into the developments, and quite unnecessarily.

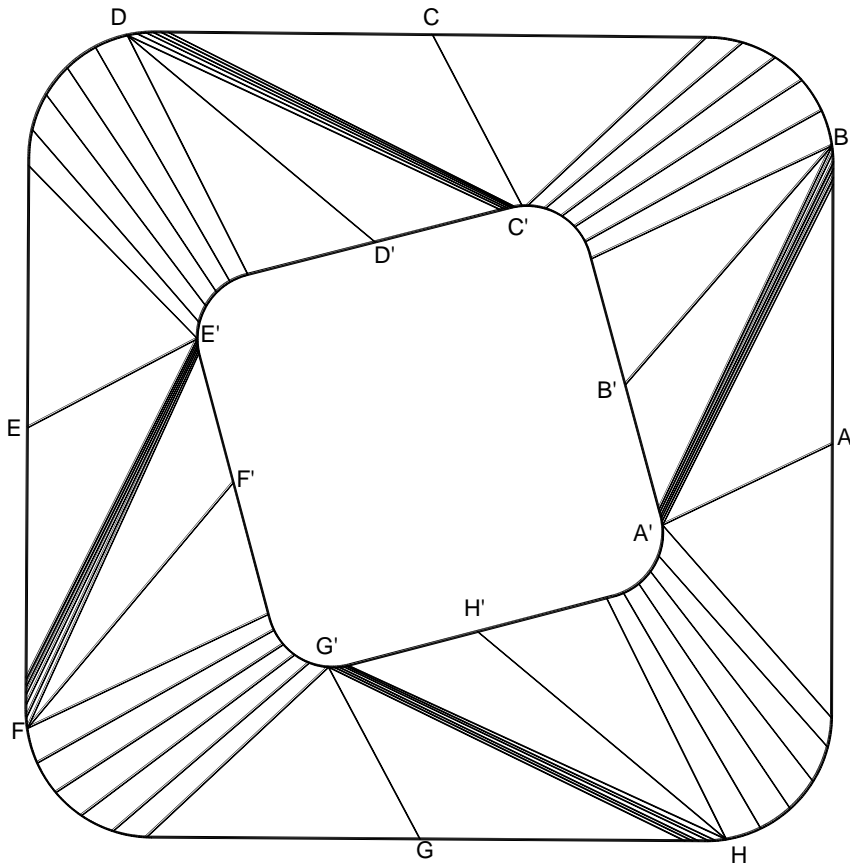
Relocating some of the development points and redefining their relativities between top and base enables us to restore some order to the shape. If a picture is worth 1000 words, then an animated model is worth at least 10 pictures.

Such a model is readily set up using Rectircle. It is recommended that the reader define a base rectircle with equal A and B dimensions, and corner radius about 1/4 of A, then a top rectircle of half that size, with a Z0 about equal to base A. Leave X0=Y0=0, and no twist or tilt initially. Omit collars and specify body long. seams at all 8 positions ABCDEFGH. Select the ‘Simple’ Setout Type option, and view the 3 dimensional model in plan. It is very regular, as expected, and shows 10 development intervals around each corner, with breaklines defining quadrilateral shaped facets since the corners are portions of oblique cones. The centers of the straight sides are also shown, dividing the side panels into pairs of quadrilaterals, and there is an obvious and direct correspondence between points in the base and top rectircles.

Now apply a twist of 5 degrees (tilt 0). The body has become distorted, but not badly so. Breaklines in the corners now define triangular facets, but the 1 for 1 point correspondence remains obvious.

Next, increase the twist to 15 degrees or more. The body has now become more distorted. Corner breaklines still define triangular facets, but half represent concave-in breaks (solid) and half represent concave-out breaks (dashed). Not good!

Now, switch the Setout Type from ‘Simple’ to ‘Exact’. The plan view has changed dramatically.



RECTIRCLE - 'EXACT' SETOUT

First, each of the 10 corner intervals has separated into 2 separate but adjacent groups of five intervals. Second, the 8 longitudinal seams have moved. Their original positions are still identifiable in the base, but they have been redefined around the top – the base A seam now connects to the top H position, etc. – and the seams now appear only as dissectors of flat triangular panels, the cones themselves are not cut by the seams. Third, in each of the side panel areas, one quadrilateral has shrunk to a triangle while the other now has an extra ‘fat break’ through it which is actually a narrow oblique cone. Fourth, there are now a total of 8 oblique cones, all of which are defined by quadrilateral facets. Fifth, the

concave-out (dashed) breaks and triangular facets in the cone areas have disappeared.

If we now introduce some moderate tilt while retaining the twist, the quadrilateral facets in the cone areas will be broken up into triangles again, but the view otherwise still looks much as before. Introducing greater tilt will introduce more concave-out breaks, indicating increasing degrees of warp in the surfaces. Switching between ‘Simple’ and ‘Exact’ setout type will not eliminate either the warp or the concave-out breaks, although the setout changes; and in this sense the description ‘Exact’ setout type should be qualified to mean ‘Best when tilt-free’.

The process of redefining and properly locating development points is well-known in manual lofting, and is fairly straight forward when the end shapes are parallel (tilt-free), but the process is both complex and time consuming otherwise, and best handled by computers, obviously.

The same model should be used to study the effects of introducing offsets X_0 and Y_0 , plus the effect on longitudinal seam positions when the ‘half-base-shape’ option is chosen, as discussed further below.

When mitered collars are specified, they usually result in non-parallel bend planes. The effect on the body shape is then, at best, as described above for tilt. Depending on the offset parameters X_0 and Y_0 , the effect may be worsened as the miter planes can then effectively be both twisted, and tilted in two directions, relative to each other. (Previously, we studied the effect of tilt in one direction only.) The ‘Exact’ setout type tends to be superior in these cases, but not always. It is recommended that both types always be tried.

In determining the setting-out parameters, and in particular twist and tilt, ducts of circular cross-section are best thought of as rectircles in which the A and B straight side lengths have been shrunk to zero, but continue to exist conceptually as tangents to the duct at the four relevant locations. Longitudinal seam locations in the 8 standard positions may also then be referenced correctly.

Rectircle also permits ellipses to be defined as base and top shapes. These use the A and B dimensions as elliptic major / minor diameters, and ignore the radius component R of the general rectircle definition. Since an ellipse is defined by an oblique cut through a circular duct, some confusion may arise. While in all

probability the ellipse will in fact be the result of such a cut, it is best to treat the case as though the duct is of elliptical cross-section, cut square, and to handle the setout in the manner described for circular ducts, with zero-length straight portions at the ends of the major and minor diameters. Any collars requested will be developed as elliptical cross-sectional elements, so unless the duct actually is of elliptical cross-section, the corresponding collar is normally omitted when an elliptical end shape is specified. A parallel collar may be specified, but a mitered collar is prohibited in this instance.

A further option available is to take the ‘+Y half only’ of the base shape. A base collar is then usually omitted, but may be parallel. A mitered base collar is prohibited. For this option, Rectircle relocates all the base development points having negative Y onto the X axis ($Y=0$) while retaining their X location. This option tends to work best for untwisted top shapes, and using the Simple setout type, but this still covers a wide variety of practical cases. One very common application is in bifurcated hoppers under bulk materials storage bins, using a semi-circular base with transformation to an offset rectangular top shape to accommodate a feeder. (The assembly is installed upside down from its setout in Rectircle)

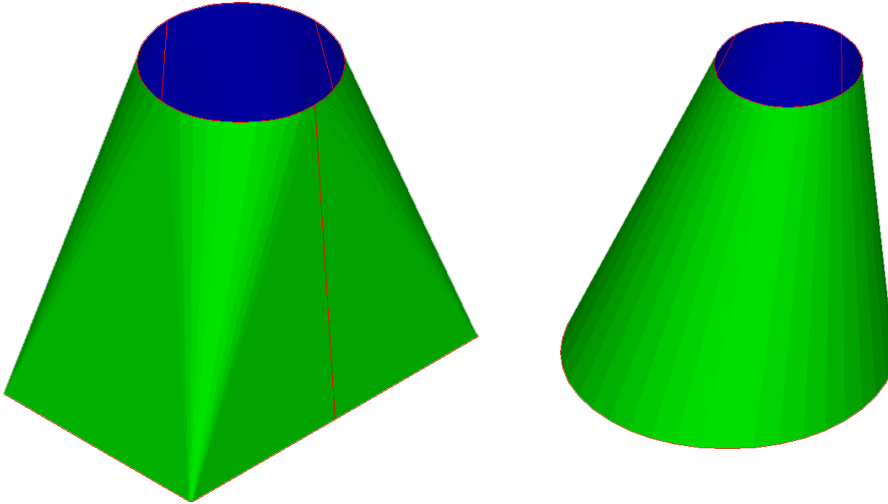
Finally, we come to the matter of dissecting the transformer body into a number of strakes through the imposition of circ. joints, between the body /collar circ. joints, or end-of-body planes.

When these two planes are parallel, either by accident or by design, the perpendicular separation between them is divided into specified percentages. Note that the perpendicular separation need not be parallel to the Z direction.

When these two planes are non-parallel, they are extended until they intersect in a line, and the minimum angle between the two planes ascertained by taking a view along that line. The angle is then dissected into the specified percentages, and the planes so defined extended to cut the body.

Each strake is developed independently. If a single-strake body contains warp, then each strake of a multi-strake version of the same body also contains warp, but this now distributed between them. Points on the circ.seam principal generators are interpolated between the ends, and diagonal breaks are inserted as required in each strake between these points.

Rort and ObCone



These are cut-down versions of Rectircle which are constrained to have parallel base and top, both twist and tilt are eliminated, mitered collars are prohibited, and any other unnecessary data entry features are removed. As a result, no warped elements appear in either of these two FastSHAPES. The Setout Type option is eliminated, and only the Simple setout type is necessary and used. The section on Rectircle should be consulted for specific details.

The purpose of these two FastSHAPES is to facilitate and expedite data entry in many common situations, eliminating at the same time any avoidable problems such as component warp.

One of the more significant points to bear in mind when using Rort(*Square to Round*) and Obcone(*Oblique Cone*) is that the base and top shapes are specified as internal sizes of prismatic (constant cross-section size and shape) ducts to which the transformer body would attach, whether or not the collar options are used.

Collars are always prismatic and best thought of as short transitional pieces of the adjoining ductwork. The total height as specified always includes the height of any (parallel only) collars that may be specified.

Free-form Developable Surfaces

Definition:

Briefly, a developable surface is one on which at any point a straight edge may be placed in some orientation and make full contact over the full extent of that surface, and which, by a process of unfolding at sets of such lines can be laid flat without stretching or tearing the surface material in that flat plane.

The most trivial example of a developable surface is a flat plane itself – it is already developed. Less trivial common examples are cylinders and cones, either right-circular or oblique, and numerous examples are included in the FastSHAPES suite described elsewhere in this manual.

A further example is a surface defined by sets of straight lines (generators) connecting between two curves (generating curves) in space. Cones and circular cylinders are specific cases in which the generating curves are circles lying in parallel planes. Prismatic tubes are specific cases where the generating curves are non-circular but are of equal size and shape and lie in parallel planes. The surface of a pyramid frustum is an example of a non-prismatic tube.

When the generating curves lie in non-parallel planes the surface may become more difficult to describe and develop correctly, and examples of this have been discussed in the section on Rectircle dealing with the corner pieces of transformers, where the surface may contain warp or twist / wind.

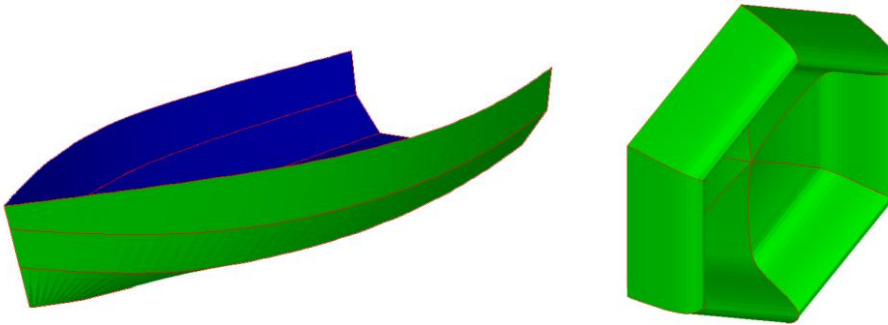
More generally again, the generating curves need not lie in any plane. The surface may become progressively more difficult to describe and develop, but not necessarily so.

When any two adjacent surfaces share a common generating curve, the straight line generators in each surface at any point on the common generating curve may or may not be parallel, and may or may not lie in a common plane. If they do not, the generating curve defines a knuckle joint between the two surfaces. The circ. joint between collar and body of a Rectircle transformer, or the circ. joints between Bend gores, provide examples of knuckle joints. The intermediate circ.

joints in a multi-strake Rectircle transformer are examples of joints where no knuckle is formed.

Such multiple surfaces are common in metals manufacturing. The aircraft construction and shipbuilding industries provide numerous examples. The yachtsman’s ‘chined hull’ is a specific case in point, where the knuckle joints are described as chines (also meaning ridges)

Surface



FastSHAPE Surface develops chined surfaces using uncompensated triangulation. The generating curves are termed chines for simplicity, whether free edges or knuckle joints. While Surface can clearly be applied to marine structures, it is by no means confined to this one application, and does not represent itself to the marine fabrication industry as an ‘expert system’ in that field.

A maximum of 21 chines may be defined, hence up to 20 individual adjacent surfaces may be used.

Each chine is defined initially by a set of node points, each of which is located by (X,Y,Z) space coordinates. It is recommended that the usual East/North/Up =X,Y,Z convention be followed unless there is good reason to do otherwise.

The generating curve is considered to pass smoothly through all node points on a chine. The term ‘node point’ or simply ‘node’ meaning ‘point of no displacement’ is used for good reason, explained later.

A **maximum of 200** nodes may be defined for each chine.

Where do the node coordinates come from? There are a number of possibilities

- from a spread sheet via the system clipboard, possibly having been calculated in the spread sheet by a macro program written by the user

- from a CAD or other program having written a comma-delimited text file in a format defined later, and read in via the Import command button (this may import more than just the node point coordinates)
- similar to the above, but confined to the node table and imported via the Edit menu Import function.
- from using the Surface GenerateData function, providing a few examples of parametrically-defined shapes and surfaces of revolution, described in detail later.
- manual data entry direct into the table, by copying from a table provided by the designer of the surfaces, such as a table of marine offsets for a marine hull

Be warned that it is not generally easy to just invent a full set of workable node coordinates. The closest Surface comes to providing this level of simplicity is to permit the user to define a single chine (usually in one of the three main planes), so that all X or Y or Z values are zero, and then to rotate this around one of the three axes to represent a surface of revolution.

Also be warned : the number of chines and nodes/chine should be known before data entry is started, and entered in the first two data cells. This will size the node table accordingly. Rows can be added or deleted via the Edit menu, but columns cannot. If more chines are needed, the table width can be expanded only by adjusting the required number of chines. If in making this adjustment the number of chines is temporarily or inadvertently reduced, the table width will contract and data will be lost. A good safe option is always to use Surface in conjunction with a spread sheet, and to copy / paste data between them.

The nodes may be distributed along the length of the chine in any desired manner, but must remain in sequence from start to end. They need not be uniformly spaced, and there may or may not be some identifiable physical relationship between nodes on adjacent chines as they appear in rows of the table, although there are certain restrictions.

Most commonly, nodes given on any one row of the table across all chines will have some relationship with each other, usually arising from the way in which the data was compiled. For example, when taken from a table of marine offsets, a row of nodes may represent a frame or bulkhead within the hull, and all points will hence tend to lie in one common plane.

In one Surface option, the Omit (splining) option, the nodes as defined are used directly as the basis for development by uncompensated triangulation. The nodes on each row of the table should have some meaningful correspondence with each other in this case. The quality of the developed surfaces is entirely dependent on the quality of the node setout, and is entirely in the hands of the user. In this case, every chine must have an identical number of nodes, and all data cells in the table must have data in them. Any blank rows must be deleted (see the Edit menu) so that the number of rows equates with the required number of nodes, and equates with the maximum number of nodes/chine as shown in its own data cell. For reasons explained below for other options, data cells must not be left blank. If a zero value is intended, a specific 0 should be entered.

The graphic adjacent to the node table shows in plan and two elevations the node ‘net’ to assist with identifying any typographical errors, etc. It can be refreshed ‘online’ as data is entered (which can be disconcerting until most data has been entered), or it can be refreshed on request (the default).

The ‘net’ view shows the nodes along each chine connected with green straight lines, and across the table rows using blue straight lines. These straight lines (plus others) will reappear exactly in the developed patterns when no splining is used.

Note that the first and last nodes on a chine may have identical coordinates, to define a closed curve, otherwise the curve is open. Similarly, if all node coordinates are identical on the first and last chine, they represent identical curves, and a closed tube results. If all chines are also closed, a closed tube (e.g. a chined torus) may be defined.

In the unsplined option, the net nearly is the surface – not all breaklines have yet been added, but otherwise it is complete.

The surfaces represent the mid thickness laminas of finite thickness material, which can be defined in the table below the net diagram. Surfaces are named according to the boundary chine numbers, thus surface 3-4 lies between chines 3 and 4. The thickness is used to calculate the masses of parts in the PartsList table. It plays no part in the development, which also assumes an equal-sided K butt weld preparation for which no detail is shown in the patterns.

Each surface may be dissected into a number of segments in the table below the materials table. The column heading show the surface name enclosed in \gg if the surface is closed (both edge chines are closed curves), or in $\langle \rangle$ if the surface is open. For the unsplined option, the number of rows in the table equals the number of nodes on each chine, and each row represents the line between matching nodes on the edge chines. A longitudinal seam may be placed at any row by double clicking the cell to toggle the word Seam on/off. Cursor control arrows will also select the cell, and any keystroke will toggle.

Up to 8 segments may be defined, hence 9 seams for an open surface, including both ends, or 8 for a closed surface in which the start / end need not be a seam.

Green may be added to either or both of the two ends (‘low node’ or ‘high node’ end according to the node numbering sequence) and two edges (‘low chine’ and ‘high chine’ edge, according to chine numbering / surface naming).

Development is initiated by left or right clicking the net diagram, or selecting the Display menu, or the PartsList tab.

The Omit spline option provides a very powerful general purpose development tool to any user familiar with the process of properly setting out development points (the nodes) around a generating curve. As an example, the GenerateData function contains a ‘SQOUND’ or Square to rOUND transformer set out using 2 closed chines with 17 nodes on each, nodes 1 and 17 being coincident on each chine. The round chine is defined in the nett as an octagon. Note the use of coincident nodes along both chines, to an extent greater than necessary to create the required net. The reason for this is discussed further below.

When developed unsplined, the sqound remains as seen in the nett, with an octagon in lieu of a circle. We will later see how to convert this to a circle through splining. If left unsplined, what would be needed would be to define more points around the circle. If we inserted two more nodes between all existing pairs of nodes, we would have a total of 24 distinct development intervals, each subtending 15 degrees of arc in the circle, plus a number of zero length / zero angle intervals. The locations of such points are readily found by simple calculation. All that is then needed is to insert 8 pairs of blank rows into the table, then edit in the new data. The corresponding points on the base are the corners of

the square, which are already shown in the table, and the Edit Copy/Paste functions quickly deal with those.

The result is still not as good as the same from Rort, but not bad. Changing the base from a square to a rectangle is just as easy. However, Surface can now do what Rort cannot. Suppose the base was not square or rectangular, but some polygon of more general shape. The vertices of the polygon are generally known, and are easily set out in the table. Properly locating points around the round takes a bit more effort – see for example discussion under the ‘Exact’ setout type in the section on Rectircle. Nevertheless, the principle of the setout in SQOUND, using coincident points alternately along both chines, is readily followed.

One further point is evident when we look at the patterns generated for SQOUND – all the breaklines are negative, i.e. the plate is to be broken concave down, and if we shade the 3 dimensional model, we will see that it is inside-out. This is because of the way in which the surfaces have been defined. We could correct this by reversing the point sequence along the chines, or by swapping the chine locations, but there is an easier way – just check or uncheck the option box under the node table labeled Reverse Inside/Outside.

Also, if the data issued by the designer had been drawn from, say, a 1 / 10 scale model – a not uncommon practice in some industries – then we could have applied a scale factor of 10 instead of 1.000 – see the data cell above the node table.

Splining:

Splining in Surface is a mathematical process equivalent to the loftsmen’s use of a physical spline and ‘ducks’. The physical spline is a thin, flexible, usually round, rod. The ducks are small heavy (would you believe, vaguely duck-shaped?) weights used to hold the spline in a curved position while allowing it to rotate and slide freely. The deflected spline follows a smooth curve between fixed points at the duck locations. It lies wherever it wants to, subject to the effect of the ducks, and is said to be ‘relaxed’ or ‘free’.

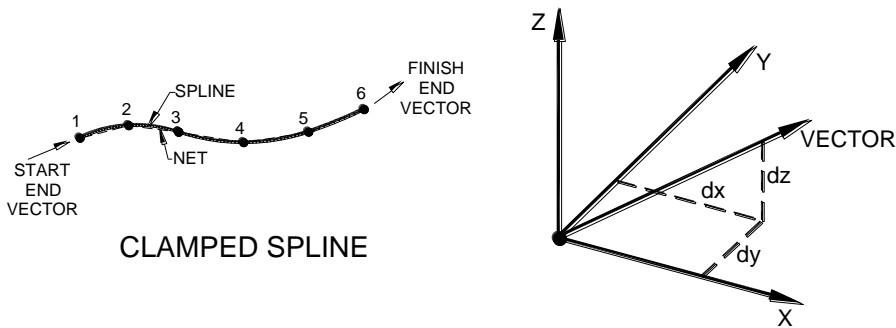
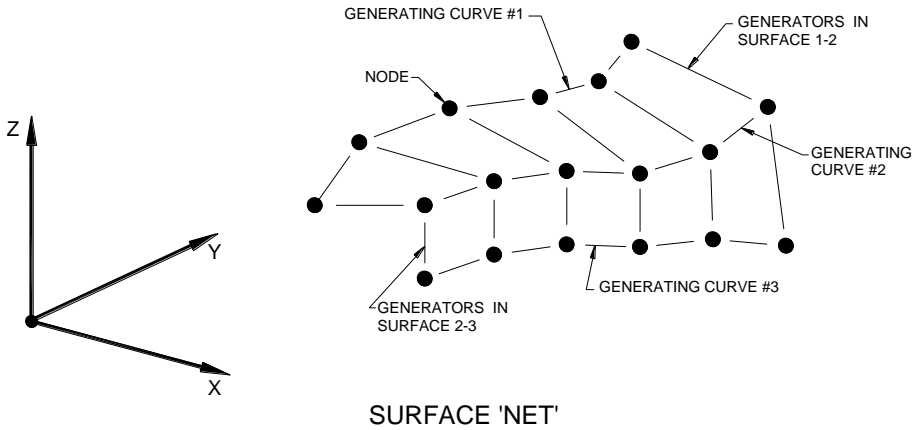
The shape of the spline can be changed in a number of ways. One obvious way is to move the ducks, but this may not be acceptable. Other ways, without moving the ducks, are

- take the end at the last duck and clamp it in some other direction than that in which it currently lies
- ditto at the start
- if the curve is closed, join start to end using a ferule, after the manner of a snake eating its tail
- if the curve is closed, tie the ends together after the manner of a tennis racquet frame
- apply some tension to the ends of the spline, thereby causing a tendency to straighten it out a little

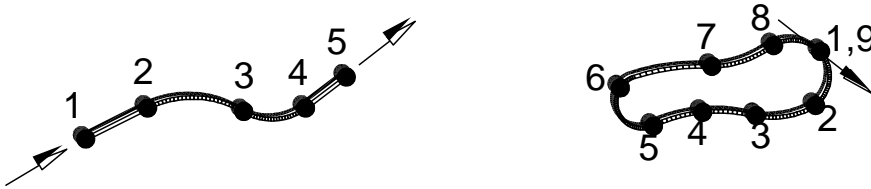
In the first two cases, the spline, is now said to be ‘clamped’. In the third case, the spline is said to be ‘cyclic’ while in the fourth case it is said to be ‘anti-cyclic’. The last case is said to be ‘tensioned’. Any other possibilities are basically variations or combinations of these.

The deflected shape of the spline over any length between ducks under any of these conditions can be closely approximated by what mathematicians and engineers describe as a cubic function. Other functions are sometimes used for particular purposes, but the cubic function is most commonly used because it most closely represents the physical situation in a reasonably simple manner. (A cubic function can be written in one line. Any better representation literally fills pages, and is not much better. It is sometimes necessary if a large amount of tension is applied.)

Surface provides for all except tension, so uses the cubic spline. The relaxed, cyclic and anti-cyclic conditions only need to be selected as appropriate, but either of the two clamped conditions requires information from the user as to the direction to be forced at either end. For this purpose, we take tangents to the spline at the end points (ducks). The tangent vector may have any length and is defined by its components dX, dY, dZ in the three main axis directions. The direction of the tangent is significant, and is defined in the direction of the spline, from start point towards end point.

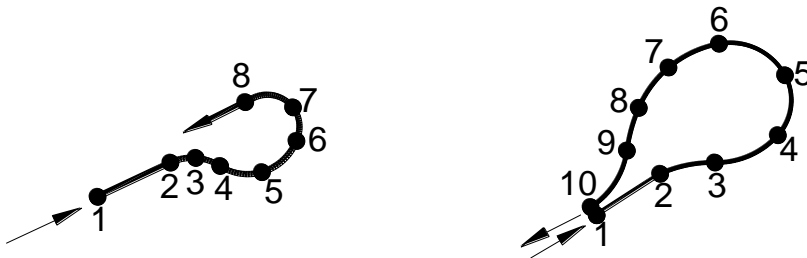


In the cyclic case, the tangents are both in the same direction, although their direction remains unspecified – it is whatever the mathematics says it should be.



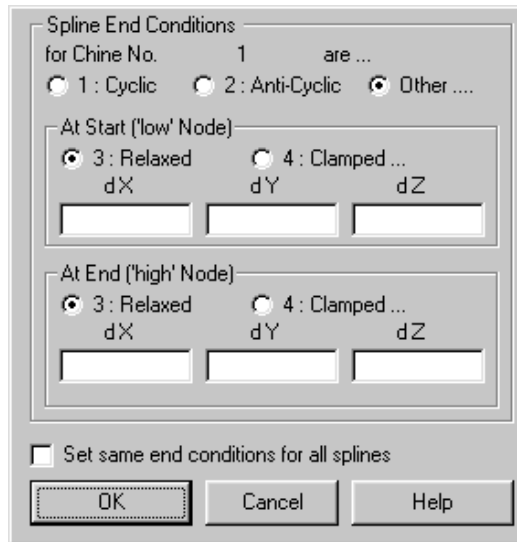
CYCLIC SPLINES (END PARALLEL TO START)

Similarly in the anticyclic case, except that the tangents are now equal but oppositely directed.



ANTI-CYCLIC SPLINES (END PARALLEL/OPPOSITE TO START)

The table below the node coordinates table provides for the spline end conditions to be specified. Columns are defined for each chine (spline). The first row is for the ‘low node’ or start end. The second row is for the ‘high node’ or finish end. *Double clicking* a cell opens a dialog for the specification of end conditions for that chine (spline). The dialog contains provision to set the end conditions identically for all splines.



The default end condition is relaxed. The cyclic and anti-cyclic end conditions tend not to be particularly useful in most practical situations – see further below. One of the curious properties of the cubic spline is that, even though the nodes through which the spline passes may be symmetrical, the spline itself does not in all cases define a symmetrical curve.

The clamped end case is potentially the most useful, especially when it is known what direction the end tangents should be. For example, suppose a number of points are laid out in the XY plane, equally spaced around a circle, starting and ending on the X axis. If the ends are left in the relaxed condition, a peak will be seen where the start and end meet. If we then impose cyclic end conditions, the start will connect smoothly to the end, but the tangent at this point will not be in the Y direction, and the curve will appear distorted from a circle. If we now replace the cyclic condition with ends clamped to (0,1,0) tangent components, the tangent will be in the Y direction as required, the curve will be symmetric, but

will still not provide a good approximation to a circle. A practical demonstration of these points is soon to be given.

Clamped end conditions are particularly useful in the case of very large assemblies, which may need to be broken down into subassemblies, either for manufacture, or for development in Surface. Very large marine hulls built using modular construction provide a good example. The end of a chine in one module can be forced to match exactly with the start of the corresponding chine in the adjacent module.

Surface provides three basic splining options, both of which are potentially affected by the end condition considerations discussed above.

1. Omit Splining
2. Spline with a specified number of intervals over the length of each chine
3. Spline with a specified number of intervals between node points

The same treatment is applied to all chines according to the selected option.

In general mathematical terms, a cubic spline can only be calculated when 4 separate and distinct points are defined, separated by three non-zero distances. When more than 4 distinct points are defined, the spline is then technically known as ‘piece-wise cubic’, details of which are outside the scope of this manual. If there are only three distinct points a cubic function cannot be found, and the only two options are a parabolic curve or a circular arc, assuming the points are not co-linear. It should perhaps be no surprise that Surface always opts for the circular arc – more on which subject below. If there are only two distinct points, the only option is a straight line, and a single distinct point remains exactly that.

The spline end conditions only affect the chine when the cubic spline is applied. They will have no effect on an option (2) chine having fewer than 4 distinct points, and may have no effect at all on an option (3) chine, depending on how it is defined – it may end in a line or circular arc. Note that although the same option treatment must be applied to all chines, different chines may have different end conditions.

Technically, the spline mathematics breaks down when any two points are coincident, and raises questions as to what the coincident points may mean in reality. Strictly speaking, we should terminate the spline and start a new one as soon as coincident points are discovered in traversing a chine from start to end. The other alternative is to simply ignore second and subsequent instances of any point. There is nothing inherently right or wrong in either case, it is simply a matter of convention.

Splining Option # 2

Surface splining option (2) ignores coincident points after the first incidence, and continues on its way from start of chine to end. The total length of the chine is then dissected into the specified number of equal length intervals, and a complete set of new points computed. The splined curve passes through all the specified node points, but the latter are not included in the set of new points other than by pure accident or coincidence. Obviously, all new points lie on a smooth (piecewise cubic) curve provided we found 4 or more distinct points. On a circular arc if we found only 3 distinct points, on a straight line if we found only 2 distinct points, and remain as specified if we found only 1 distinct point. Regardless of how many nodes in total, or the number of distinct points that were defined, every chine now has the same number of points along it, and for purposes of surface development, these are now assigned a 1 to 1 correspondence between chines.

In the node coordinate table, the start and end points of any chine can be anywhere in the depth of the table. We take the first point on any chine as that in the highest row with non-blank entries in its three (X,Y,Z) columns. The last point is in the lowest row with all blank entries in its three columns, or at the bottom of the table. It is clear that the digit 0 should be used when zero is intended, and that blank rows are very undesirable features of the table which could cause untold confusion.

In order to make splining option (2) work, we need to specify a greater number of intervals than occurs in the ‘longest’ chine, as defined by `max.nodes/chine`, or the number of rows in the table. This is a bit coarse, but nevertheless fairly realistic. We can see how this works by returning to the previous SQOUND example. Specify more than 16 intervals, say 18, select splining option (2), and then take a plan view.

Disaster! - What has happened is that all the sets of coincident points so carefully built into the data have been treated as single points, and smooth curves fitted through balance. To ensure some semblance of sanity, specify a sensible number of intervals, say 40. The result is still a disaster. Apart from the loss of square corners in the base, the top, which is supposed to be a circle, is anything but that. Its most apparent defect is a peak on the side. Using cyclic end conditions eliminates the peak, but introduces a noticeable flat spot. Clamping the ends to

(0,1,0) vectors in lieu of cyclic end conditions also eliminates the peak, but not the flat spot.

Splining option (2) is extremely useful in many practical applications, but this obviously is not one of them.

Splining Option # 3

Now consider splining option (3) – specified number of (equal size) intervals between node points. The piece-wise cubic spline, (or lesser circular arc / straight line / single point contractions of it), means that it is possible to retain the original node point(s), and to simply insert additional points between.

Applying this to SQOUND with 3 or more intervals between nodes now gives the desired result.

First, Surface did not spline through coincident points. Second, circular arcs were fitted between sequences of 3 distinct points, straight lines were fitted between sequences of 2 distinct points, and distinct points were forced to remain as distinct points. The node table breaks the circle into 4 quadrants, each defined by 3 distinct points (0,45,90 degrees), and coincident points have been inserted between the quadrant ends to inhibit splining through them and ensure retention of the 3-point arcs. Using 3 intervals between nodes puts the new points at 15 degree increments around the original arc quadrants. More is better, of course – consult Dr.Cone if in doubt. Surface’s GenerateData function also contains a Cone, which demonstrates the 3-point arc feature more clearly still - it defines two closed chines set out as triangles.

It will also be found that changing the spline end conditions will have no affect whatsoever on the development – the cubic spline has not been used in either chine.

Under splining option (3); the long. seams table will be seen to have changed in length. There is a row for every point set, and the original node point rows are highlighted. Switching back momentarily to option (2) reveals that the table changes to provide the total requested number of intervals, but the original node rows are no longer highlighted - they no longer exist.

Whenever the splining option is switched, or the number of intervals is changed, Surface attempts to relocate specified seam locations by proportionally spacing them. This is not infallible however, and the reallocations should always be confirmed by the user. Generally speaking, allocation of seam locations is best left till last, after satisfactory surface developments have been obtained.

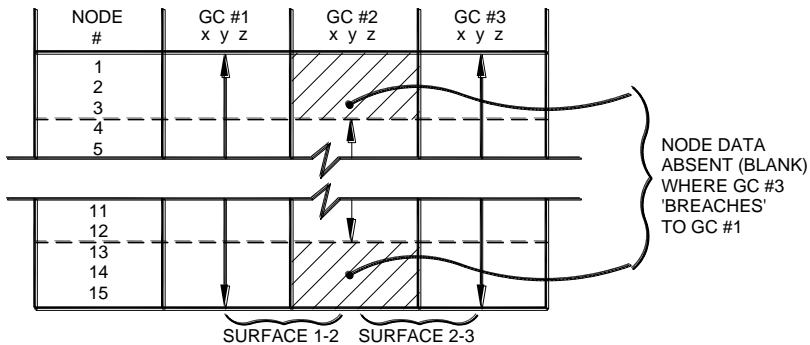
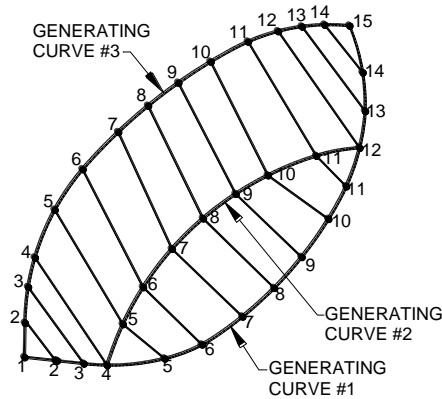
Under splining option (3) there is a sub-option, ‘allow breaching to chine no. _’. This facility provides for assemblies in which some surfaces may be shorter than others, typically as found in marine hulls where the higher plates tend to extend the full length of the hull, and lower plates start from the keel line behind the prow end. The longer (higher) plates can be said to breach the gap created by termination of the lower plates by ‘breaching to the keel’. The ends of the surfaces in the breached areas are typically triangular in shape.

If the node table does not have equal numbers of nodes on every chine, then splining must be used – the Omit option (1) cannot work, literally by definition. Splining option (2) will function in this situation, but as seen, may not produce the desired result, and tends to work best when all chines are defined by the same number of nodes.

Splining option (3) will only function in this situation when a breaching chine is specified, and a few other basic rules are followed. These are

- one, and only one, breaching chine must exist
- the breaching chine must extend over the full length of the assembly (other chines may also do so)
- the end node point of a short chine must coincide with the node point on the breaching chine for the same node set (table row) Any of a variety of warnings will be issued if any of the above are violated. The net diagram will usually help to resolve the issue. Breached surfaces are very simple to deal with once the basic concept is grasped

Return now to splining option (2). We have seen that while it will always function, it is not a happy substitute for option (3) when node points have been set out specifically for this latter option.



'BREACHING' IN 'SURFACE'

Option 2 is best applied to surfaces in which

- there are no inherent straight lines on any chine, hence no need to use coincident points to force their appearance
- if any chine is a circle then a 4-point closed chine will when splined provide points having adequate correspondence with points on other chines that may not be circular.

- the chines do not otherwise have extreme curvature, or rapid changes in curvature
- node points can be well distributed along chines, and there is a logical connection between sets in each chine
- chines are generally of similar length and shape

Data Generator:

Surface’s GenerateData function contains three examples.

1. ‘**Cone**’ is an example of using two closed 4-point chines. Its net shows the chines as triangles. After splining with either option 2 or option 3, the triangles are converted to true circles.
2. ‘**Moebius**’ is just a fun thing, but nevertheless demonstrates the points made above. It could just as easily be a helical ramp, for example. We might, with a bit of modification, also have a doubly curved box girder supporting a freeway ramp.
3. ‘**Gulp**’ – the water tower – is also a fun thing. It demonstrates the same points, plus provides an example of a surface of revolution.

The surfaces of revolution are of some interest and have numerous practical applications. Given that Surface splines only along chines (down the node table columns) and not around node sets (across the node table rows), there are always two ways to tackle a setout.

‘Gulp’ was created by defining nodes along chine 1, and then rotating it about the Z axis. When viewed in plan, the circles are seen to be faceted, and the chines are evident. (More (20 max) or less (3 min.) are readily got by just clicking option 7 in the GenerateData function after having made ‘Gulp’). When viewed in elevation, we see smooth curves set out through the original 19 nodes, and expanded by splining option (2) to any extent desired (within reason).

An alternative (superior) way to make ‘Gulp’ would be to stack up a series of cones, giving all perfectly circular horizontal knuckle joints. All that is needed is

to invert the setout of the nodes in the table, defining the elevational profile across node row 1. We then rotate node row 1 about whatever axis we set out the points along, presumably still Z, and request 3 intervals in the circle. This will make four node rows with the first and the fourth identical. Using splining option 2 or 3 converts the chines into perfect circles. Note that if more than 3 intervals in the circle are requested, the spline will not be forced to a circle. However, while we are limited to 21 chines / 20 intervals when rotating a chine, we have no such limit when rotating a row, so we can request as many intervals as necessary to develop the cone by triangulation to the desired level of accuracy. (When 3 intervals are used, the number of spline intervals required is based on the same considerations, of course - consult Dr.Cone for details.) These exercises are left to the user to perform.

Because many of the surfaces that can be defined may be different in subtle ways from those defined by other FastSHAPES, Surface uses a modification to the uncompensated triangulation development algorithm.

The surfaces defined by other FastSHAPES are, or are intended to be, formed basically concave inwards. It is seen in other sections that while it is sometimes necessary to use negative breaklines where surfaces are warped or twisted, the surfaces overall remain basically concave-in. During their development, FastSHAPES aims to insert diagonal breaklines in such a manner that twist / warp effects are minimized and the overall curvature is maintained as consistently as possible.

In Surface, surfaces may be required to have twist (e.g. a helix), or may be required to be concave-in in some areas, concave-out in others. The latter is common in marine hulls and similar structures. These features require that development points in the general vicinity of both the principal surface generators and the diagonal breaks be examined in greater detail than otherwise – it can no longer be assumed that the surface is intended to be consistently concave in one direction or the other. There is also the more general question of what is the ‘inside’ in any given case. When the user switches the definition, Surface simply reverses its reading of the data to ensure that in the patterns laid out inside-up show positive breaklines when curvature is concave-up.

In the marine industry in particular, there has been considerable effort put into techniques other than triangulation for surface development. The objective of

most of these is to eliminate warp in the hull plates, or put into the context of triangulation, to eliminate the need for the diagonal breaks across quadrilateral panels. This is a task, which has met with limited success, and has been accompanied by the apparent realization that in a general sense it is futile. When a surface is warped, it is warped. The best way to eliminate warp is to define the surface in the first place in such a way that it is free from warp.

For example, in the section on Rectircle, we saw how oblique conical corners could be distorted by the application of body twist using the ‘Simple’ point set out, with the result that the development incorrectly indicated corner warp. Revising the point setting out eliminated the indicated warp. We also saw that when some tilt was applied, the corners ceased to be portions of oblique cones, were warped, and remained so. To summarize, an unwarped surface can be made to appear to be warped through use of inappropriate point location, but a warped surface can never be made to appear unwarped.

The desire for unwarped surfaces, which are generally simpler to fabricate, has led to the concept of the so-called multi-conic surface, comprising at worst, portions of oblique cones connected smoothly together. Constructing such a surface can be more difficult than it sounds, but when successful, it can be developed in a series of quadrilateral panels provided the source points from which the surface was set out have not been lost.

In the section on Bend, we also saw that an unwarped surface (Autonest-enabled) could be made to appear to be warped by inappropriate allocation of development points and the use of triangulation. In order to be correctly developed using triangulation, the multi-conic (Auto-nestable) surface in Bend would require two sets of points along each generating curve (circ.seam / chine), one set for each surface.

Regrettably, it often happens that by the time a table of marine offsets, or its equivalent in other industries, has been derived from elements of the surface other than its source setting out points, any chance of ‘reinventing’ the original points has been dramatically reduced. Positions of bulkheads, frames etc. may be totally irrelevant to the node positions needed to properly define the surfaces.

No amount of splining will restore the original points and the 1 to 1 relationships needed between them. If it is indeed possible to find them, assuming they even

exist, a far more comprehensive search technique is required than is provided by Surface.

Despite the potential deficiencies of the method, simple triangulation still remains the most commonly used development technique for chined surfaces.

Power Transmission Components

The development of high quality cutting systems such as laser sees an ever-increasing number of, basically flat mechanical components being taken off milling machines and put onto NC profiling machines

This is especially so in cases where the profiles or forms can be defined, or adequately approximated, in terms of straight lines and circular arcs, the two fundamental cutting path types on NC profilers.

FastSHAPES provides for both roller chain sprockets (a.k.a. ‘chainwheels’ – FastSHAPES uses only the term ‘sprocket’) and involute spur gears. All are programmed using direct development.

All sprocket forms are defined using straight lines and circular arcs, hence can be developed precisely. Involute gears involve a degree of approximation discussed in detail below.

Depending on FastSHAPES purchase and installation options, the suite may include these components for users not wholly familiar with their details, but having an occasional or potential need to use them.

For some reason, sprockets can be a mysterious device to such users, who typically ask ‘but where do I specify the diameter?’.

The answer is, ‘You don’t’. A sprocket is defined primarily by the chain that it drives or is driven by and; the required number of teeth. Exact formulae are defined in various standards for calculating diameters defined in specific manners, and these should be consulted for the details. As a very rough rule of thumb, the pitchline diameter is about $NP/3$, where N is the number of teeth, and P is the chain pitch. Add the chain roller diameter, D_r , for an estimate of overall sprocket diameter, or rough square size of blank from which the sprocket can be cut.

On the other hand, diameters are fundamental in setting out gears.

Occasional users are best advised to confine themselves to components defined absolutely by a standard, and to ensure that their customers have correctly specified their requirements.

Roller Chain Sprockets Generally

FastSHAPE Sprocket covers roller chain sprockets to ASA / ANSI / ASME B29.1, ANSI B29.15 and ASME B29.15M, in which the B29.1 sprockets also cover JIS S-form and U-form sprockets.

FastSHAPE Sprocket + adds two generic and user-definable types, designated Type A (straight sided teeth) and Type B (teeth having a topping radius)

ANSI B29.15 and ASME B29.15M are examples of generic Type B, with a specified set of setting out data. Terminology used in Sprocket / Sprocket + basically follows definitions given in the ANSI standard. Nomenclature may differ.

Other than ASME B29.1, which has a form specifically applicable to short pitch roller chains; most other common standards use either the generic Types A or B. For example: ISO 487 and ISO 3512 use Type B; although the latter permits a modification that makes it then more similar to the B29.1 tooth form.

Further additions to Sprocket + are customized for specific manufacturers, and availability is limited to them.

Standard features common to the FastSHAPES producing sprockets are

- option to add a bore hole and a sequence of spoke holes
- option to modify the tooth height and details of its outer periphery
- use of color to indicate various design details of the tooth form on the data entry form

Spoke holes comprise a specified number of circular holes laid out evenly spaced around a pitch circle of specified diameter. Partial borehole and a typical spoke hole are indicated on the tooth profile graphic when their option boxes are checked, omitted otherwise. Similarly for the pattern. In order to see larger scale detail of the tooth form, switching the options off temporarily is suggested.

The question of tooth height and outer periphery detail is left open in most standards. Provided the tooth has sufficient height (including allowance for chain

stretch), these details do not affect the mechanical efficiency of the sprocket. They may need to be considered depending on attachments made to chain links, and components such as buckets, scrapers etc. that may be connected to them.

The two basic tooth height options are

- clip the top of the tooth flat, parallel to the pitchline and at a nominated height above the pitchline
- radius the top of the tooth. The center of the arc is at the center of the sprocket, but the height is given as the radial distance outside the pitch circle

These two options are readily visualized when pitchline clearance is zero. The radiused top is unaffected by finite pitchline clearance, but the flat top definition is revised as the radial height on tooth centerline above the chainlink centerline at its mean position between drive and brake conditions.

In the case of finite pitchline clearance, (which can be quite large in certain applications), if a chain attachment existed parallel to the chainlink centerline at a height that would otherwise interfere with the tooth, clipping the tooth height to a flat top or simple radius can involve a fair amount of calculation to determine the appropriate height to specify for a guaranteed required minimum clearance.

Sprocket therefore provides a third option

- profile for link fittings. The height to be specified is the perpendicular distance from the chainlink centerline to the attachment, less the required clearance. The profile may include a central radial crown, and flat shoulders either side of the crown.

A relief radius, when provided for, is simply a small fillet between to top and side of the tooth, and this is added as required to any specified clipping or profiling of the top.

The tooth form diagram shown on the data entry form is laid out basically is shown in the relevant standard, although additional features are shown. Features shown and colors used are as follows.

In all cases, bore and spoke holes are shown in cyan, with the bore center at the center of the lower edge of the diagram. As noted earlier, these details have the effect of diminishing the displayed size of the tooth form, and may be switched on/off.

For the B29.1 sprockets, two half-teeth between part-radii and the gap between are shown in dark red, the pitchline in white and roller in green. No attempt is made to indicate pitchline clearance, (if any, and usually too small to show adequate detail), and the roller is shown in the center of the tooth gap.

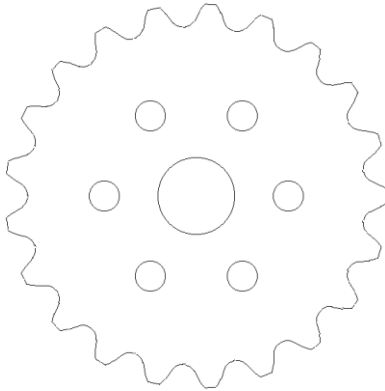
For all other sprockets, the tooth itself is shown in profile, with a partial gap each side.

The tooth profile is generally shown in cyan for the standard teeth in Sprocket, and may be either cyan or yellow. Yellow indicates that the sides of the teeth intersect below an anticipated normal tooth crown profile. When a standard is used (Sprocket) it may be assumed that this is acceptable, although see further discussion below. For Sprocket+ generic user types A and B, the yellow color can be considered to serve as a warning that tooth thickness may not be adequate. (If a relief radius is specified, this may have the effect of further reducing tooth height in this case.)

A pair of rollers and pitchline (link centerline) is shown in white at one extreme of pitchline clearance, and in green at the opposite extreme.

For the generic user-defined types A and B in Sprocket+, a clearance line is added, shown in dark blue if there is no interference between roller and tooth as the chain wraps on/off the sprocket, and shown in red if there is interference.

Sprocket



Sprockets covered by this FastSHAPE are essentially of U.S. origin, at least in terms of published standards.

Early editions of ASA B29.1 previously defined two types, Type I having a finite pitchline clearance, and Type II having zero pitchline clearance. At some point in time lost to this author, the tooth topping radius was increased. Earlier editions indicated it being centered on the pitchline at $1.24 \cdot Dr$ from the roller center, while later editions now show this at $1.4 \cdot Dr$, where Dr is the roller diameter. In all cases, Sprocket uses the later $1.4 \cdot Dr$.

The ASA B29.1 Type II sprocket reappears in today’s edition of ASME B29.1 – 1993 without reference to any Type. The ANSI B29.1 Type II and J.I.S. S-form sprocket are identical with it. ‘Sprocket’ omits reference to Type II, and covers all three under ASME B29.1

The Type I sprocket is not mentioned in ASME B29.1. However, this is identical with the J.I.S. U-form sprocket. The pitchline clearance is $0.002'' (0.051\text{mm}) + 0.07 \cdot (P - Dr)$, where P is the chain pitch, Dr the roller diameter. (As noted above, the topping radius is located at $1.4 \cdot Dr$)

‘Sprocket’ refers to ANSI B29.1 Type I, rather than ASA or JIS for the Type I sprocket. No real confusion need or should arise.

Sprockets covered by these standards are applicable only to short pitch chain, having P/Dr ratio between 1.5 and 2. Standard chains covered by these standards actually range in this ratio from 1.56 to 1.92, with 1.6 to 1.8 being more prevalent. ‘Sprocket’ declines to develop when less than 1.5 or greater than 2.

A minimum of 4 teeth, and maximum of 200 teeth may be specified. However, ASME B29.1 notes that ‘Normally, large sprockets should not exceed 120 teeth’. A practical minimum number of teeth is 7.

Chain for these sprockets is usually precision made (machined) chain for higher speed and / or mechanical power transmission applications, although see introductory remarks concerning laser cutting devices. ISO 606 attempts international unification in this area, nevertheless the B29.1 sprocket persists and endures.

Optional tooth height clipping is confined to the flat top option only, and the standard height of $0.3 \cdot P$ is offered, where P is the chain pitch.

ANSI B29.15 covers steel roller chains, attachments and sprockets more specifically intended for use in chain conveyor / elevator applications. These are generally lower speed, more ‘agricultural’ applications than those for the ANSI / ASME B29.1 chain, and not necessarily machined.

The 1973 edition of ANSI B29.15 is used by Sprocket. It is believed that there have been no technical changes affecting sprocket details since that data.

Chains having P/Dr from 1.5 to 6.0 are defined, with number of teeth, N, from a minimum of 6 to a maximum of 36. However, there is an apparent deficiency in ANSI B29.15 – 1973 when $P/Dr < 1.8$. ‘Sprocket’ declines to develop when data lies outside the range $1.8 \leq P/Dr \leq 6.0$, and an error dialog is issued when $P/Dr < 1.8$. The dialog states ‘Working Face Length and/or Topping Radius Too Small’. Use of Sprocket + generic Type B is required in this event, but some data needs to be modified – the specified form simply fails with the standard data.

Pressure angles specified in ANSI B29.15 – 1973 have been subject to minor rationalization in Sprocket. For less than $N = 12$ teeth, the standard specifies pressure angle as $1.5 \cdot N$, but then rounds down to a whole number. For 12 or more teeth, up to $N \leq 36$, pressure angle is specified in whole numbers from 17

degrees (12 teeth) to 27 degrees (36 teeth). This latter sequence of pressure angles is readily seen to be equivalent to rounding of figures calculated from a consistent Complementary Angle (see later discussion) of 32 degrees. The latter is used by Sprocket.

The Working Face Length is set at the specified $0.01 * P * N$. ANSI then qualifies this as a maximum. ‘Sprocket’, perhaps not unreasonably, assumes that the maximum is workable over the ranges of form parameters covered by the standard. A warning ‘Working Face too long’ is issued if using the maximum violates the qualified conditions, but no other action is taken.

All three tooth height options are available. The minimum pitchline clearance of $0.1 * D_s$ is used, where D_s is the pocket (roller seating) diameter, equal to the maximum permissible roller diameter (i.e. nominal plus maximum tolerance).

The ‘Show details’ option will reveal all technical details of the ANSI B29.15 sprocket, although these may not be modified by the user of standard ‘Sprocket’. They may be modified under ‘Sprocket +’. Modifications possibly required include Working Face length.

The ASME B29.15M (- 1997) sprocket is different from the ANSI B29.15 – 1973 sprocket in several ways, described below. These two standards should not be confused.

- a different sequence of pressure angles is used. Pressure Angle = $(N + 3)$ degrees for number of teeth, $N \leq 17$, and for $18 \leq N \leq 36$ can be derived after rounding to whole numbers from a constant Complementary Angle (see later discussion) of 30 degrees.
- the range of P/D_r ratios is $1.78 \leq P/D_r \leq 3.0$, with an accompanying change in the ranges of chain pitches and roller diameters. Where ANSI covers a total of 31 standard chains in pitches from 3” to 30” and roller diameters from 1.5” to 6.0” , ASME covers only 9 standard chains in 3” , 4” and 6” pitches and roller diameters from 1.5” to 3”.
- pitchline clearance ranges from minimum $0.1 * P$ (adopted by Sprocket) to a maximum of $0.15 * P$, substantially larger than the ANSI $0.1 * D_r$.
- topping radius is $0.5 * P$, compared with $(P - D_s)$ for ANSI.

‘Sprocket’ otherwise handles ASME B29.15M in the same conceptual manner as ANSI B29.15

Sprocket +

User-defined generic Types A (straight sided teeth) and B (with topping radius) may have from 4 to 999 teeth. The upper limit being perhaps outrageous, (100 or so being generally more practical) and are confined to P/Dr ratios from a minimum of 1.2 to a maximum of 999 (that is, far higher than could ever be practical).

All other details of the tooth form then need to input by the user. These details should be entered by reference to an appropriate standard – the ‘boutique sprocket’ should be confined to experts having some knowledge of mechanical design principles and requirements.

For the expert, a B29.15 sprocket, (either ANSI or ASME) may be defined, with details revealed by checking the ‘Show Details’ box. Switching from a B29.15 to a Type B sprocket will leave those details set in place and make them accessible for editing by the user. Switching between Types A and B is permissible, but setouts for each as shown in the tooth form graphic may not be available for either or both unless data is within sensible ranges.

For both the expert and the non-expert, a variety of National and International standards exist. They tend to be very similar in result, although the way in which the data is presented, and terminology used, may differ.

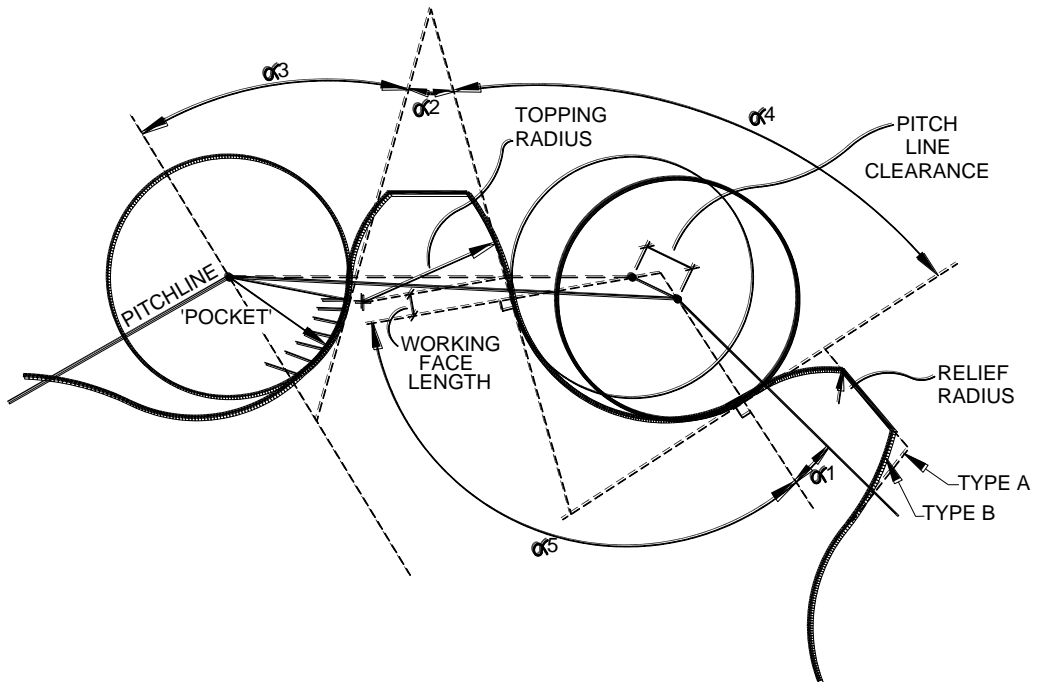
‘Sprocket +’ uses the terminology and definitions given in ANSI B29.15, which are closely followed by ISO 3512. Both define sprockets of the generic Type B.

While a generic Type A sprocket differs dimensionally, it is effectively just a Type B sprocket with the topping radius omitted, hence application of the same terminology in either case is appropriate.

One of the more apparent differences between various standards is the way in which pressure angle, or more generally a Working Face Angle, is defined. ‘Sprocket +’ offers 5 choices

1. Pressure angle, measured between pitchline and pressure line
2. Tooth thickness angle, measured between working faces of a tooth

3. Complementary angle, measured between working face and radius through adjacent roller center
4. Included tooth angle, measured between working faces of a tooth gap
5. Included pressure line angle, measured between pressure lines in adjacent teeth



SPROCKET WORKING FACE ANGLES

- α_1 = PRESSURE ANGLE
- α_2 = TOOTH THICKNESS ANGLE
- α_3 = COMPLEMENTARY ANGLE
- α_4 = INCLUDED TOOTH ANGLE
- α_5 = INCLUDED PRESSURE LINE ANGLE

These 5 angles are all inter-related. Specification of any one defines the other four. The pressure line is a line taken through the pitchpoint (center of roller when hard against the tooth face) and perpendicular to the working face. By definition, it lies at the Pressure Angle ‘inboard’ of the pitchline, or chainlink centerline between rollers.

When the pitchline clearance is zero, the relationships are readily seen. When the pitchline clearance is non-zero, the angle subtended at the center of the sprocket between roller centers at either end of the tooth gap (drive/brake locations) also enters the relationships.

For the Type B sprocket, the Working Face Length is measured outwards from the pressure line. Some specifications, e.g. ISO 487 for general purpose chainwheels, contain no working face and the topping radius center is located on the pressure line. The included pressureline angle (option 5) is specified, and varies with the number of teeth required. Pitchline clearances as specified are a relatively large proportion of the roller diameter.

ISO 3512 permits the working face to be either flat or convex. Only the flat option is covered by Sprocket + generic Type B. (The convex face option is geometrically more akin to the ANSI / ASME B29.1 standard sprockets in form, but not necessarily in detail.)

For the flat working face option, ISO 3512 is very similar to ASME B29.15M, as compared below.

- pressure Angles are defined by $(N + 3)$ for number of teeth, $N \leq 17$. For $18 \leq N \leq 100$, a constant Complementary Angle (option 3) of 30 degrees is used, (although this is tabulated in terms of pressure angle, rounded off.) ASME B29.15M is identical, except that it is limited to a maximum of 36 teeth.
- pitchline clearance is $0.1 * P$ for non-machined sprockets, or when operating in a dirty environment, and $0.003 * P$ for machined sprockets, or when operating in a clean environment. ASME B29.15M uses $0.1 * P$ min. to $0.15 * P$ max.
- the working face length is $0.01 * P * N$, subject to qualification for which see the standard (ANSI B29.15 and ASME B29.15M have identical requirements, differently expressed in all cases).
- a range of 8 standard chains is defined with pitch, P from 2.5” to 7.0”, roller diameter from 1.25” to 3.5”, and $1.89 \leq P/Dr \leq 2.28$. This is very similar to ASME B29.15M.

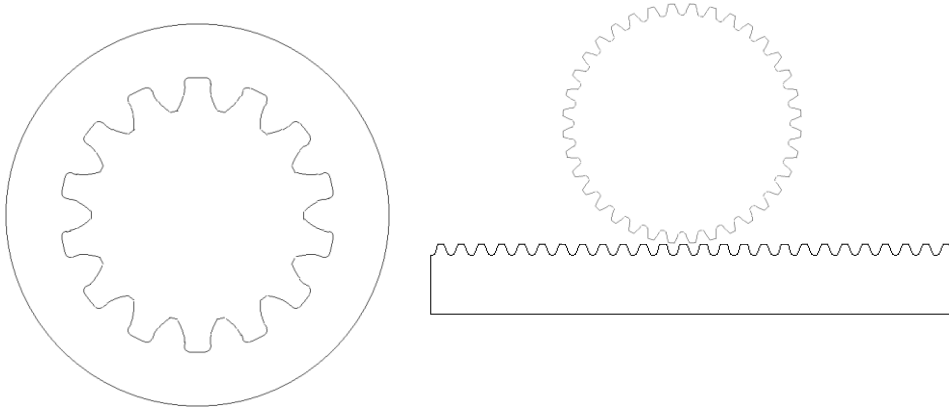
‘Sprocket+’ issues an error message ‘Working Face Length appears excessive’ when a line through the outer end of the working face and perpendicular to it (hence the inner radius of the topping curve) lies ‘outboard’ of the adjacent pitchpoint (roller center). Interference problems may also be indicated in this event. (The message is ‘Working Face too long’ when the data is drawn from either ANSI B29.15 or ASME B29.15M)

For the Type A sprocket, the whole side of the tooth is effectively the working face, so neither the working face dimension; nor the topping radius are required as data.

‘Sprocket +’ also contains the option to define the root of the tooth gap as flat or radiused. Flat is sufficient when pitchline clearance is small, in which case it is tangential to the roller pockets. Radiused (to the root radius) is normally required for large pitchline clearances.

Error messages raised by ‘Sprocket+’ aim to be as informative as possible. Nevertheless, it should be appreciated that any apparent problem could be caused by any one or more of many data parameters. Great effort has also been put into suppressing unnecessary messages, as might otherwise be raised while data is being modified. Despite our best care in these matters, it remains possible that some ultimately annoying message may persist for some time until data is brought under control. This seldom occurs with users who know what they require, but can cause frustration with the ‘boutique designer’ who is ‘just throwing a few numbers around’ to see what happens. One suggestion is to start from a standard, show its details, then modify gently.

Involute Spur Gears



The types of spur gear envisaged by FastSHAPES to be cut on NC profiling machines are more likely to be found in larger, lower speed, more ‘agricultural’ applications that in high speed power transmission applications where noise, wear and other quality factors are paramount.

Such applications can be found typically in mining, bulk materials handling and metals smelting in such devices as linear / reciprocating shuttle drives, ladle tilting mechanisms and the like. Environments are typically both noisy and dirty, and gear mechanisms of a size that frequently lie outside the economic capabilities of most standard gear cutting systems.

Nevertheless, design concepts applicable to smaller higher quality gear trains are equally applicable to the envisaged applications and are used in these FastSHAPES.

RandP produces patterns for Rack and / or Pinion applications, while **InvInt** produces patterns for Internal gears, these being spur gears in all instances, and restricted to the involute form. Each provides for gear sectors as well as full gears, and both use identical concepts as to data entry and output. Except as noted where applicable, the following applies to either or both cases.

Provision is made for three common standards:

- the 14.5 degree full depth tooth,

- 20 degree full depth tooth, and
- 20 degree stub tooth,

plus a user-definable non-standard option. It should be noted that 14.5 degree pressure angles are less common today than in the past, with the 20 degree stub tooth being perhaps the most popular. Pressure angles up to 25 degrees are also used.

Gearing is specified by, pitch circle diameter (PCD) and number of teeth (N) in a complete gear or pinion. A reference section shows:

- Circular pitch
- Diametral pitch (teeth per inch of PCD)
- Module (mm of PCD per tooth)
- Base Diameter
- Root Diameter
- ID (internal gear) or OD (pinion)

These are treated as reference material rather than specification data because teeth of the size envisaged being cut by these FastSHAPES are normally larger than are covered by normal circular pitch, diametral pitch or module based systems.

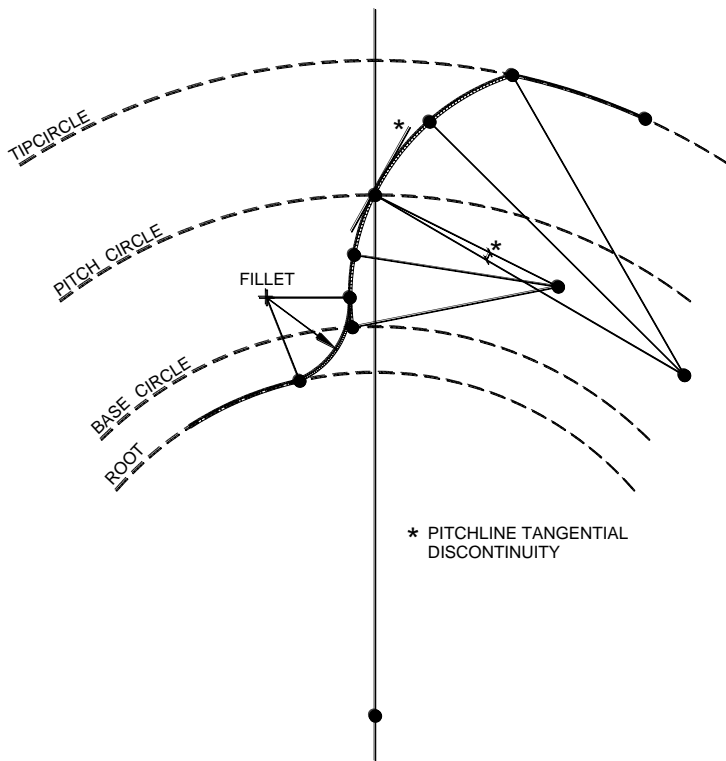
Standard addendum, dedendum and root fillet data are shown for reference together with facility to enter non-standard data and the option to switch between. Pinion or gear teeth otherwise have no provision for modification of the profile at the tip.

For the rack, there is facility to add flank relief, specified by radius and tangent point height above pitchline. No standard is offered in this case.

The involute curve, while being ideal for defining the tooth form, is not ideal for cutting on all NC profiling equipment; which is normally limited to cutting paths defined by straight lines and circular arcs. FastSHAPES output does not support any more complex cutting paths than these.

The involute could be approximated by a series of very short straight line entities, or a by series of longer circular arc entities. In either case, discontinuities in slope arise at entity match points.

The approximation adopted uses two circular arcs, one above and one below the pitch circle, each fitted through 3 points on the involute with the common point on the pitchcircle. Other points are on the base circle and tooth tip circle, plus two intermediate between these and the pitch circle.



INVOLUTE CURVE APPROXIMATION

There is a slight discontinuity in the arc tangents at the pitchline, expressed in degrees in the following table, positive indicating a convexity of the tooth surface.

Error in setting out expressed in terms of percentage of the tooth circular thickness is as follows. The Max. (+) figures indicate worst oversize, the Min. (-) figures indicate undersize, both measured normal to the tooth face.

PressureAngle/Dep th	14.5 ° FULL DEPTH	20 °full depth	20 °stub
10 TEETHDISC.	-0.08 °	+0.64 °	+1.1 °
	Max. +0 .30 %	+0.21%	+0.12 %
	Min - 0.19 %	-0.15 %	-0.09 %
100 TEETH DISC.	+0.25 °	+0.05 °	+0.04 °
	Max. +0 .05 %	+0.01 %	+0.01 %
	Min. - 0.04 %	-0.01 %	-0.01 %

Error varies over the four half-arcs in sequence (under/over/under/over) from baseline to tip, with the 5 reference points located without error, by construction.

In a typical instance of tooth circular thickness approx. 8” (200mm) the worst case error for a 20 ° stub tooth pinion with 10 teeth would be about $0.0012 \times 8 = 0.010$ ” or 0.24mm. Note that this is applicable at each face of the tooth. The error roughly doubles for full depth 20 ° teeth.

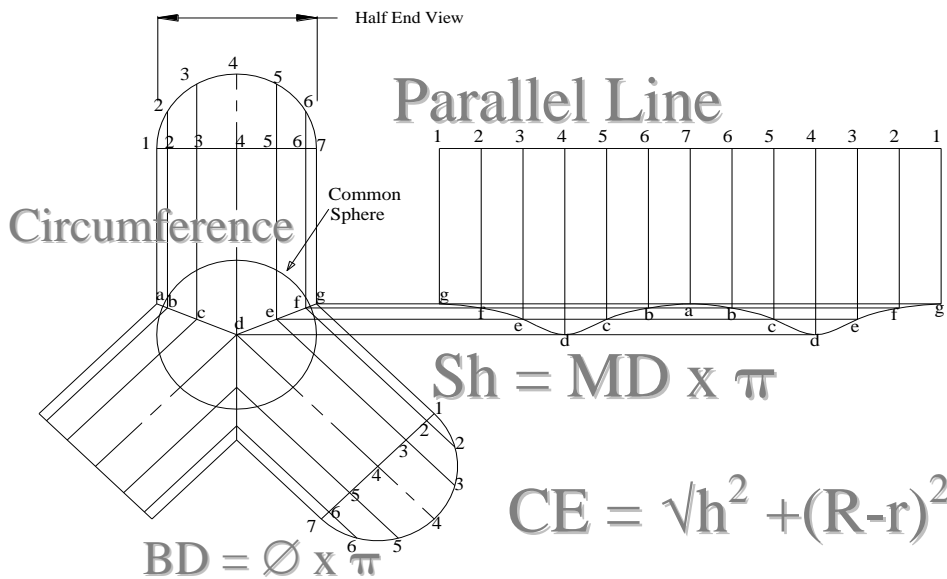
Error of this typical magnitude is considered acceptable in the context of most thermal cutting processes likely to be used, and needs to be considered also in the context of required clearances / backlash in the typical applications envisaged.

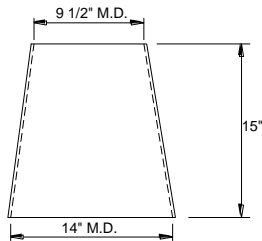
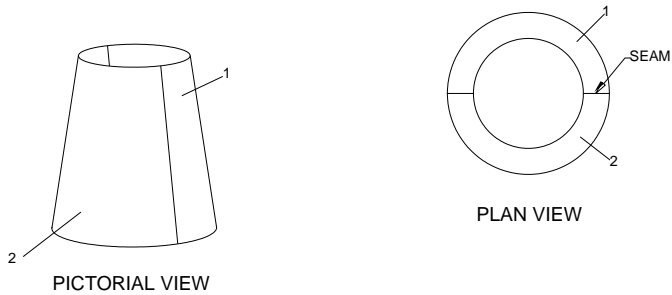
No specific provision is made for modification of the tooth forms for clearances or backlash. It is expected that for gears of the size envisioned, the installation would provide for adjustment of gear center locations and attainment of required clearances. A further option is to cheat on the specified amount of kerf compensation required when the components are cut.

FastSHAPES provides no further guidance to the user on the subject of appropriate or inappropriate applications or suitable design of gearing.

CHAPTER

3





CONE	Base Dia. = 14" MEAN	Material Thickness = 1/4"
	Top/Dia. = 9 1/2" MEAN	
Further Requirements: 2 Segments required		

CONE

1. Run the FastSHAPES® program by “double clicking” the FastSHAPES icon on your Windows desktop.
2. From the FastSHAPES® library, “double click” on the CONE icon.
3. Select the JOB data form at the top, by clicking on it.
4. Click in the “Job Number” field and type in “1”, and then press the **TAB** key.
5. You are now in the field for “Drawing Number”. Type in “**NF25-2-4**” and press the **TAB** key twice.
6. You are now in the field for “Assembly Number”. Type in “**Top Section**” and press the **TAB** key three times.
7. You are now in the field for “Prepared by”. Type in your name.



The data screen is not a set of mandatory fields. It is up to the user as to whether these fields are filled in or not. They will however help the user find and retrieve previously saved data.

8. Select the CONE data form at the top, by clicking on it.

In this exercise you will produce a single stroke, single segment elbow.

The Cone dimensions are:

Large DIA: 14” MD

Small DIA: 9 ½” MD

Height: 15”

Material Thickness: ¼”

IMPORTANT: All developments are calculated on the Mean Diameter

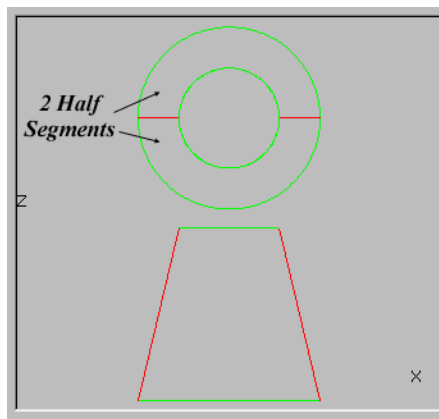
Click on the “**Material**” button.

9. Choose a Class and Specification from the Drop Down boxes.
10. Click in the “Thickness” field and type in $\frac{1}{4}$, and then click “OK”.
11. Click in the field for “Top Diam”. Type in $9\frac{1}{2}$ and press the **TAB** key.
12. You are now in the field for the “Bottom Diam”. Type in 14 and press the **TAB** key.
13. You are now in the field for the “Axis” height. Type in 15 and press the **TAB** key.
14. Select the “MEAN” option for the “Dimensions”.

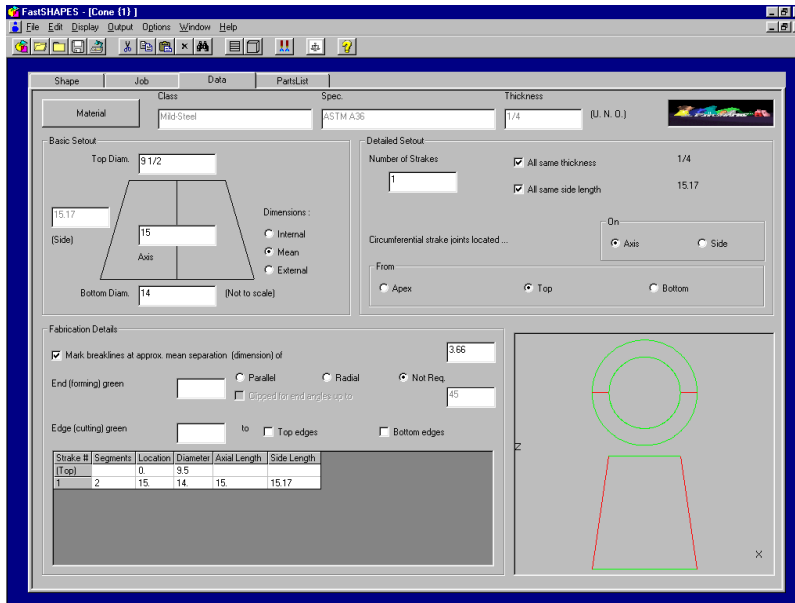


This is done because all dimensions on the drawing are specified as “MEAN”.

15. Now go to bottom left hand corner of the screen under “**Fabrication Details**” and look for “Segments” in the Table. Click on 1 in second box and change this value to 2, then press the ENTER key. The display window should now show 2 half segments.



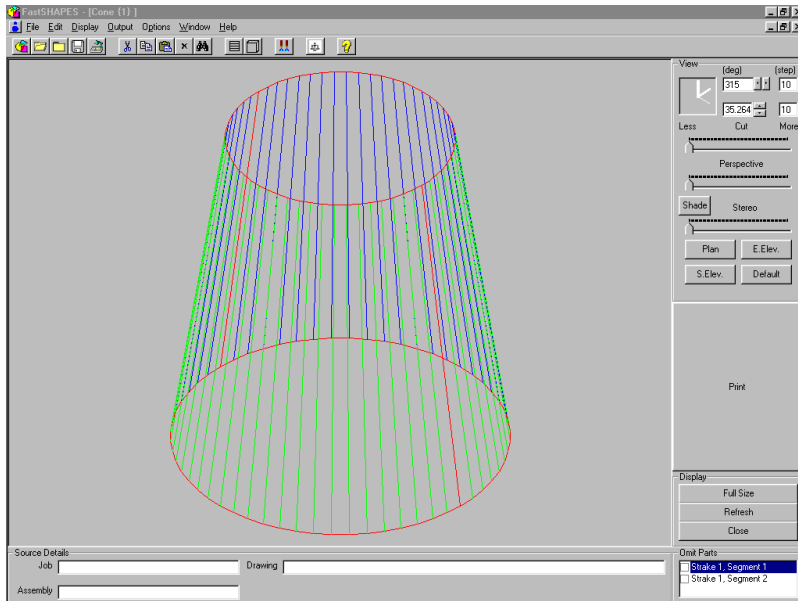
16. Check the “Mark breaklines...” option. Now move the mouse to the box opposite this and click inside with your mouse. Now type in 3.66 ($1\frac{1}{2}$ th Mean Circumference).



17. From the menu, select “Display-Part(s)”.



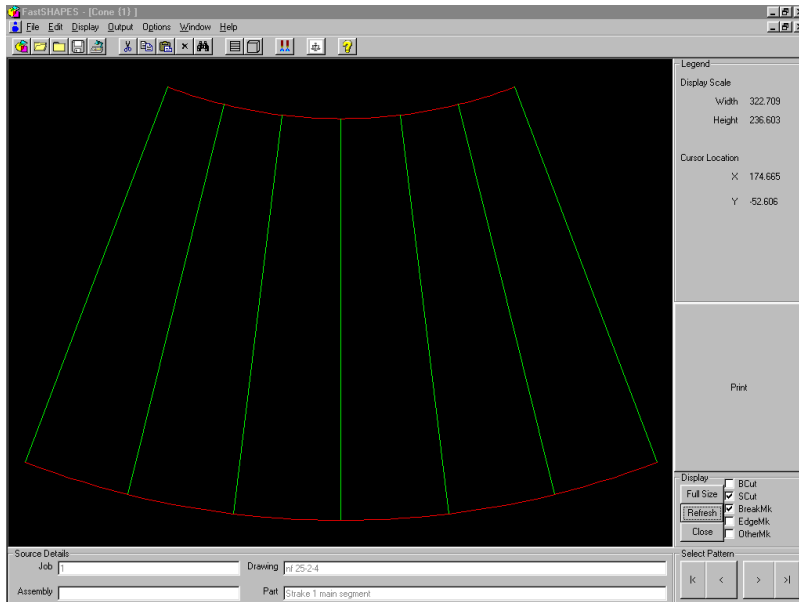
The screen will now show the Wireframe of the developed CONE.



18. Now click the “**SHADE**” button, which can be found on the right hand side of the screen under the “**VIEW**” options. The CONE will now be shaded, showing the inside of the construction as **BLUE**, and the outside of the construction as **GREEN**.
19. Click the **CLOSE** button, to return to the **DATA** form.
20. From the menu, select “**Display-Pattern**”.



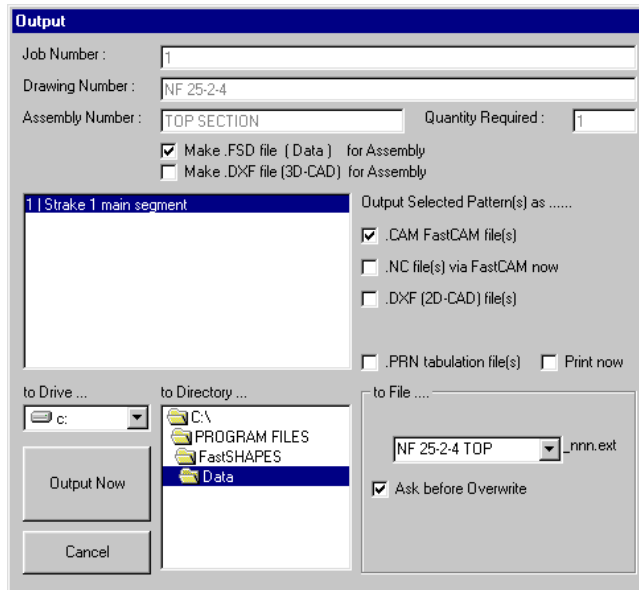
The screen will now show the developed Pattern of the CONE.



21. You should now see the pattern of one-half of the CONE development on the screen. The RED color indicating the cut boundary of the pattern, and the LIGHT-GREEN indicating the Breaklines (presslines)
22. Click the CLOSE button, to return to the DATA form.
23. From the menu, select “Output”. The Output form will now appear.
24. Check only the following options:
 - “Make .FSD file (DATA) for Assembly”, &
 - “.CAM FastCAM file(s)”



In this example we will only be generating the pattern in the FastCAM “CAM” file format. For information on producing NC Code refer to topic “NC CODE GENERATION”.

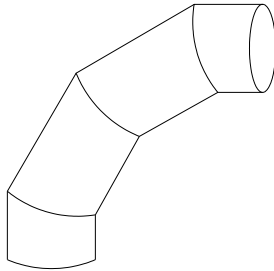


25. Click in the “To File...” field and type in “**NF 25-2-4 TOP**”. This will be the name used for the output files.
26. “Double Click” in the “To Directory...” field and change into the directory where you would like to save the files.

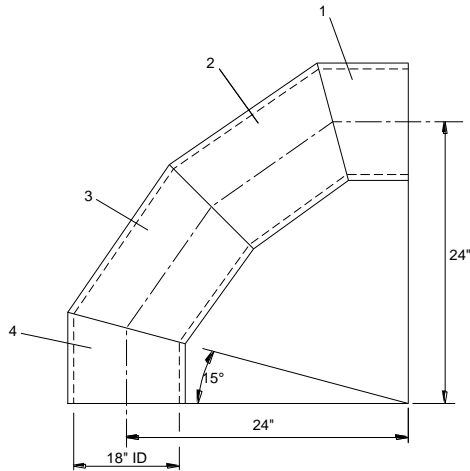


Try and avoid saving your output from FastSHAPES into the FastSHAPES directory, as this is meant to be for the programs’ internal files ONLY.

27. Click the “OUTPUT NOW” button.



PICTORIAL VIEW



ELBOW	Base/Inlet Dia. = 18"ID	Bend Angle = 90°	Material Thickness = 1/4"
	Top/Outlet Dia. = 18"ID	Bend Radius = 24" to Centerline of Bend	
Further Requirements: Seams to be 180° apart. All segments to be STITCH cut.			

ELBOW

1. Run the FastSHAPES® program by “double clicking” the FastSHAPES icon on your Windows desktop.
2. From the FastSHAPES® library, “double click” on the ELBOW icon.
3. Select the JOB data form at the top, by clicking on it.
4. Click in the “Job Number” field and type in “2”, and then press the **TAB** key.
5. You are now in the field for “Drawing Number”. Type in “**NF26-2-1**” and press the **TAB** key twice.
6. You are now in the field for “Assembly Number”. Type in “**4 GORE MID ELBOW**” and press the **TAB** key three times.
7. You are now in the field for “Prepared by”. Type in your name.



The data screen is not a set of mandatory fields. It is up to the user as to whether these fields are filled in or not. They will however help the user find and retrieve previously saved data.

8. Select the ELBOW data form at the top, by clicking on it.

In this exercise you will produce a 4 Gore, single segment elbow.

The Elbow dimensions are:

Base/Inlet DIA: 18” ID

Top/Outlet DIA: 18” ID

Bend angle: 90°

Bend Radius: 24” to Centreline of Bend

Material Thickness: ¼”

Further Requirements:

Seams to be 180° apart

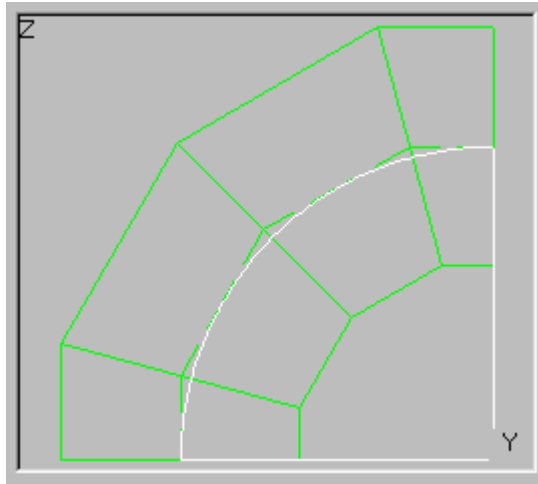
All segments to be STITCH cut

9. Click on the “**Material**” button.
10. Choose a Class and Specification from the Drop Down boxes.
11. Click in the “Thickness” field and type in $\frac{1}{4}$, and then click “OK”.
12. Click in the field for “Base Internal Diameter”. Type in 18 and press the **TAB** key.
13. You are now in the field for the “Bend Angle”. Type in 90 and press the **TAB** key.

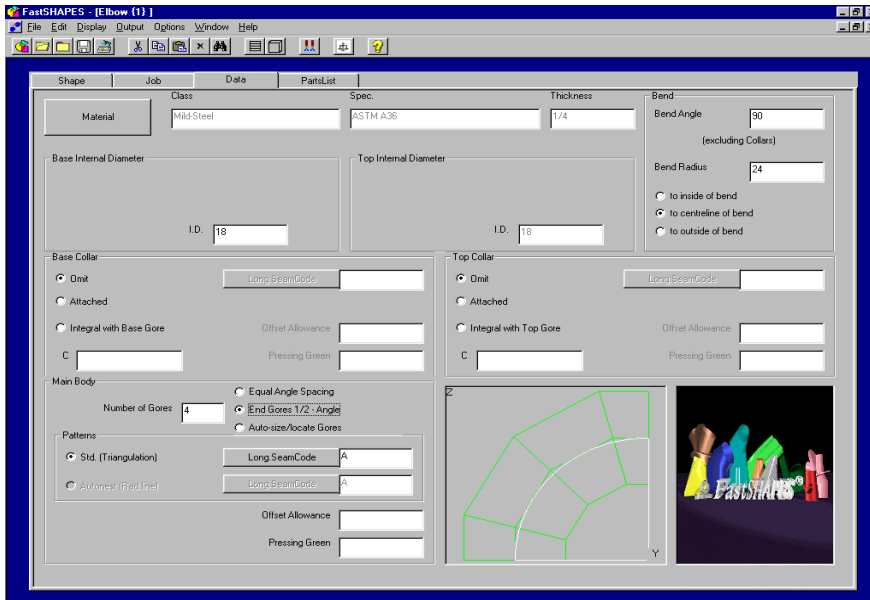


Note, how you did not have to fill in the Top Internal diameter. It was automatically done, as the Elbow program is designed to only construct Constant Diameter Elbows.

14. You are now in the field for the “Bend Radius”. Type in 24 and press the **TAB** key.
15. Select the “to centreline of bend” option for the “Bend Radius”.
16. Now go to bottom left hand corner of the screen under “**Main Body**” and click in the field for “Number of Gores”. Change the default value of 2 to 4, then press the **TAB** key. The display window should now show 4 gores.



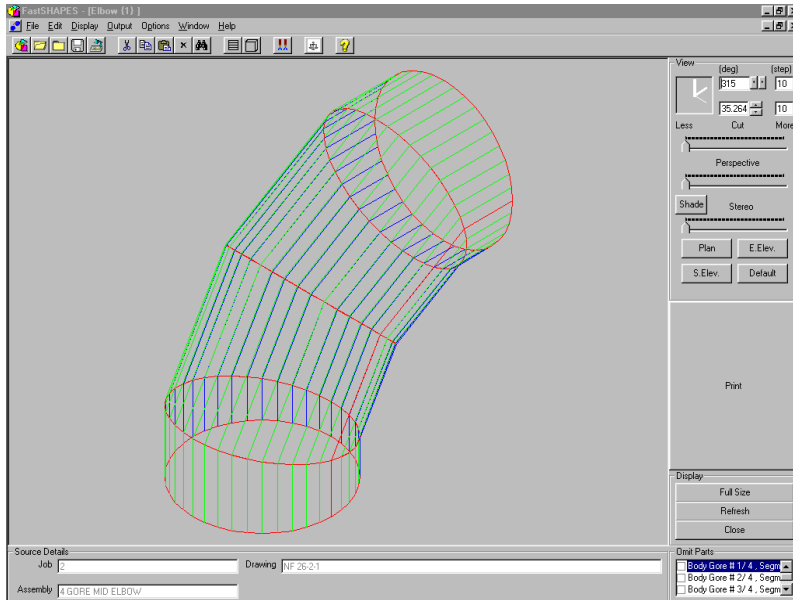
17. Move your mouse over the Display Window, and a message will appear: *“Left click for Parts, Right click for Patterns”*, Click your left mouse button.





The screen will now show the Wireframe of the developed ELBOW.

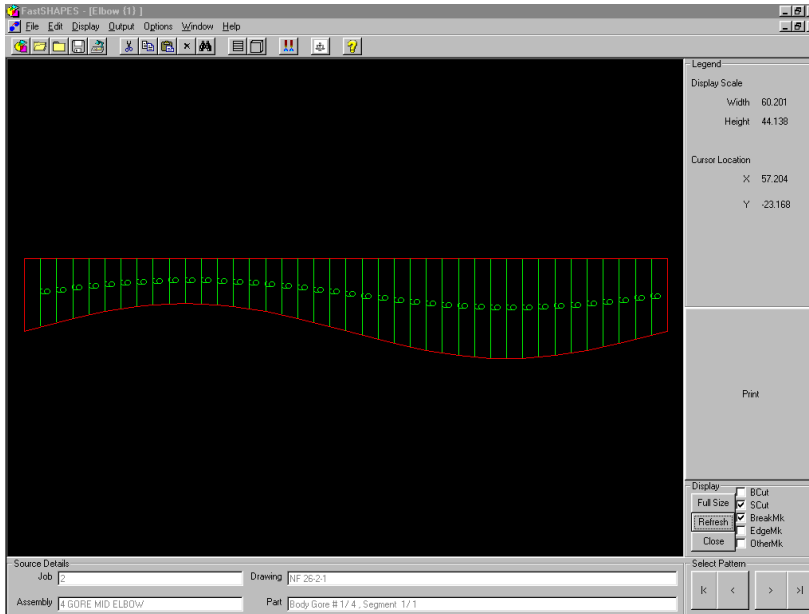
18. Click the CLOSE button, to return to the DATA form.



19. Move your mouse over the Display Window, and click your right mouse button.



The screen will now show the developed Pattern for the first gore of the ELBOW.



20. Move to the Lower right portion of the screen under the “**Select pattern**” area, and click the “right” arrow. This will now show the subsequent developed patterns, 2, 3 & 4.

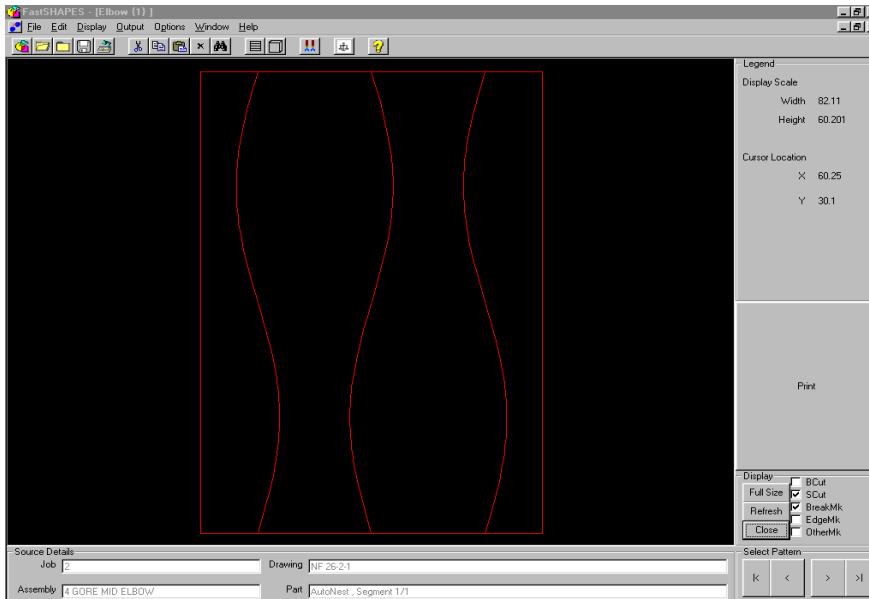
21. Click the CLOSE button, to return to the DATA form.

22. Now go to the “**Main Body**” section and select “Auto-size locate Gores”. This will now enable the “Autonest (Rad Line)” option under the “Patterns” section. Select this option, and then set the “Long Seam Code” by clicking on the “Long Seam Code” button to “A”.

23. Move your mouse over the Display Window and click with your left mouse button.

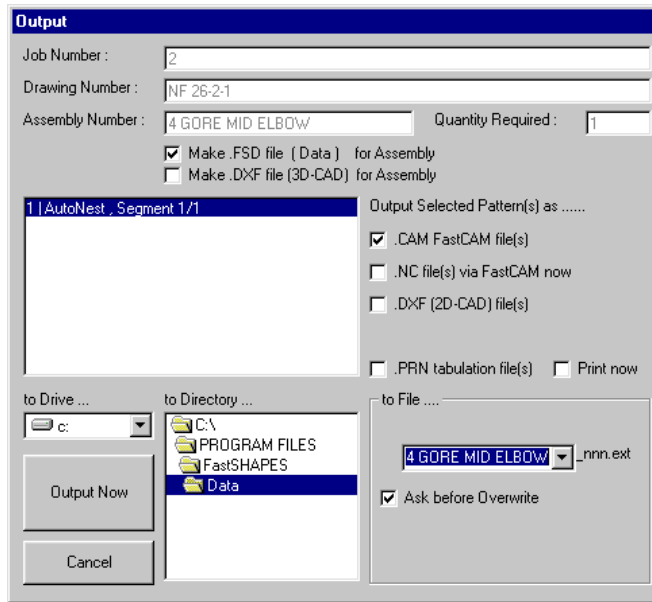


The screen will now show the developed Pattern for the ‘back to back’ development of all of the gores. This was done so that the items could be STITCH cut, internally. Note how the seams for each gore are now 180°



apart from each other.

24. Click the CLOSE button, to return to the DATA form.
25. From the menu, select “Output”. The Output form will now appear.
26. Check only the following options:
 - “Make .FSD file (DATA) for Assembly”, &
 - “.CAM FastCAM file(s)”



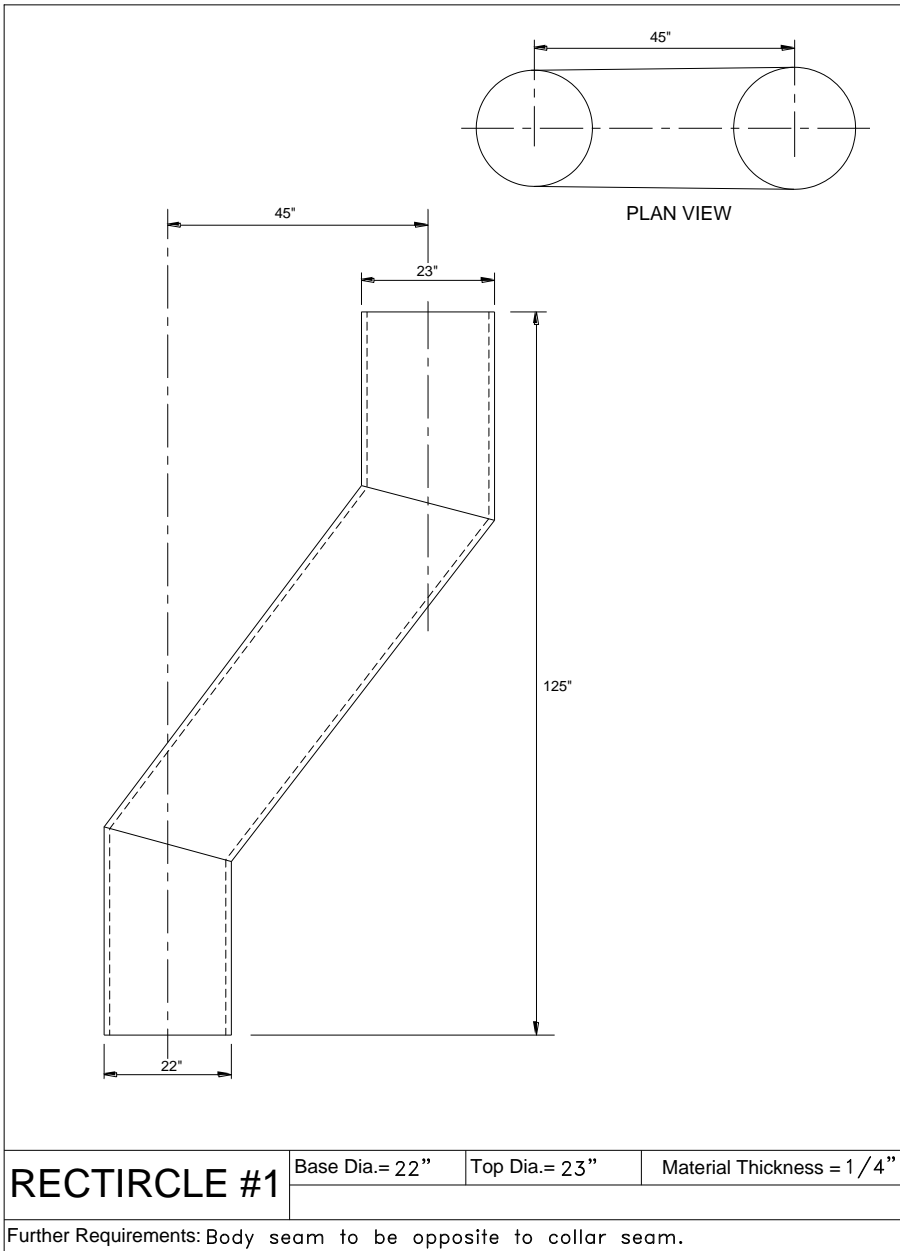
In this example we will only be generating the pattern in the FastCAM “CAM” file format. For information on producing NC Code refer to topic “NC CODE GENERATION”.

27. Click in the “To File...” field and type in “**4 GORE MID ELBOW**”. This will be the name used for the output files.
28. “Double Click” in the “To Directory...” field and change into the directory where you would like to save the files.



Try and avoid saving your output from FastSHAPES into the FastSHAPES directory, as this is meant to be for the programs’ internal files ONLY.

29. Click the “OUTPUT NOW” button.



RECTIRCLE#1

1. Run the FastSHAPES® program by “double clicking” the FastSHAPES icon on your Windows desktop.
2. From the FastSHAPES® library, “double click” on the RECTIRCLE icon.
3. Select the JOB data form at the top, by clicking on it.
4. Click in the “Job Number” field and type in “9”, and then press the **TAB** key.
5. You are now in the field for “Drawing Number”. Type in “**OFFSET PIPE**” and press the **TAB** key twice.
6. You are now in the field for “Assembly Number”. Type in “**OPIPE9**” and press the **TAB** key three times.
7. You are now in the field for “Prepared by”. Type in your name.



The data screen is not a set of mandatory fields. It is up to the user as to whether these fields are filled in or not. They will however help the user find and retrieve previously saved data.

8. Select the RECTIRCLE data form at the top, by clicking on it.

In this exercise you will produce an Offset Pipe transition.

The Rectircle dimensions are:

Base Dimensions: 21 ½” ID

Top Dimensions: 21 ½” ID

Vertical Height: 125”

Segments: 1

Min Vertical Height on Base: 32”

Min Vertical Height on Top: 24”

Material Thickness: ¼”

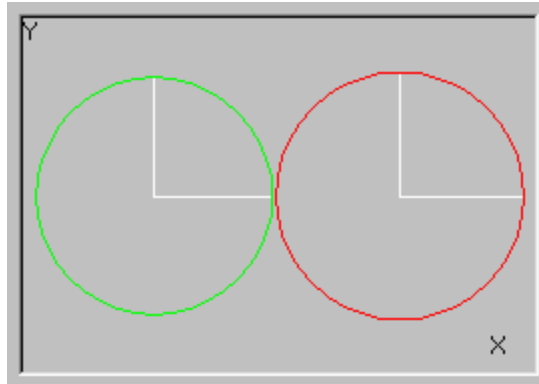
Center to Center Offset: 45”

9. Click on the “**Material**” button.

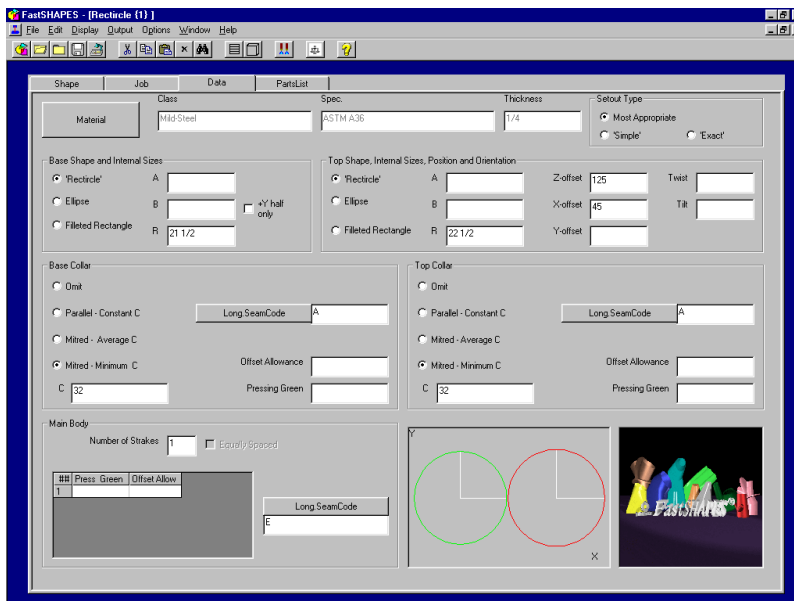
10. Choose a Class and Specification from the Drop Down boxes.
11. Click in the “Thickness” field and type in $\frac{1}{4}$, and then click “OK”.
12. Select the “Rectircle” option under the “**Base Shape and Internal Sizes**”, and press the **TAB** key three times.
13. You are now in the field for the “Internal Corner Radius”. Type in $21 \frac{1}{2}$ and press the **TAB** key twice.
14. Select the “Rectircle” option for the “**Top Shape, Internal Sizes, Position and Orientation**”, and press the **TAB** key three times.
15. You are now in field for the “Internal Corner Radius”. Type in $21 \frac{1}{2}$ and press the **TAB** key.
16. You are now in field for the “Z-offset” (Height). Type in 125 and press the **TAB** key.
17. You are now in field for the “X-offset”. Type in 45 and press the **TAB** key.
18. Select the “Mitred – Minimum C” option under the “**Base Collar**”, and press the **TAB** key.
19. You are now in the field for the “Collar Length”. Type in 32.
20. Click the “Long.SeamCode” button, and check Seam “A”
21. Select the “Mitred – Minimum C” option under the “**Top Collar**”, and press the **TAB** key.
22. You are now in the field for the “Collar Length”. Type in 24.
23. Click the “Long.SeamCode” button, and check Seam “A”
24. Now go to bottom left hand corner of the screen under “**Main Body**” and click the “Long.SeamCode” button, and check Seam “E”.



NOTE: The Seams have been chosen as A & E to keep the Collar and Body Seams opposite to each other.

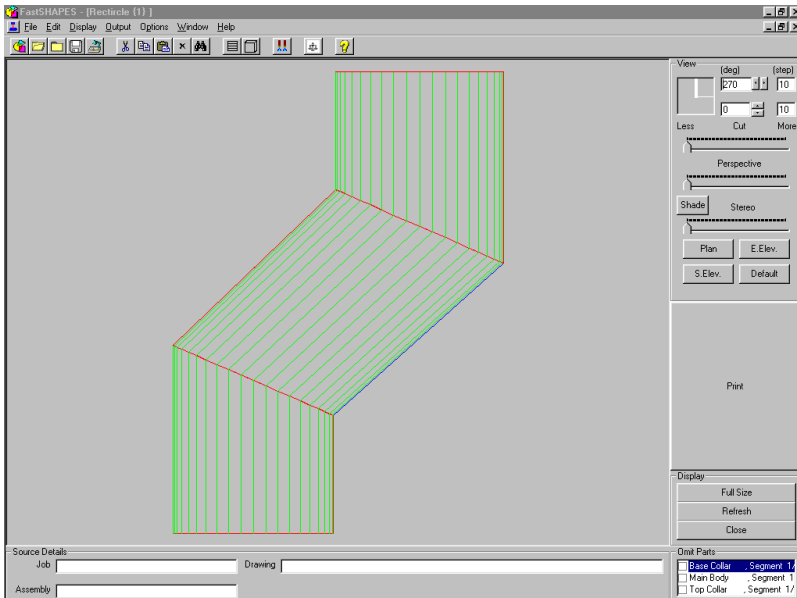
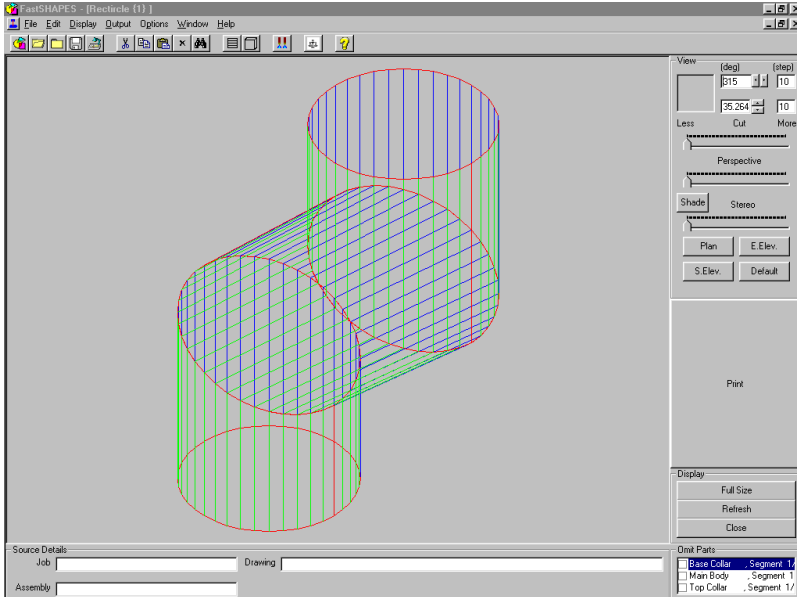


25. Move your mouse over the Display Window, and a message will appear: “Left click for Parts, Right click for Patterns”, Click your left mouse button.





The screen will now show the Wireframe of the developed RECTIRCLE.





NOTE: The seams for the Collar and Body are on opposing sides.

26. Move to the CLOSE button, and click your right mouse button.



The screen will now show the patterns for the development. You can scroll through all the patterns, by simply clicking on the “NEXT” and “PREVIOUS” buttons under the “Select Pattern” area.

27. Click the CLOSE button, to return to the DATA form.

28. From the menu, select “Output”. The Output form will now appear.

29. Check only the following options:

- “Make .FSD file (Data) for Assembly”, &
- “.CAM FastCAM file(s)

Output

Job Number : 2

Drawing Number : NF 26-2-1

Assembly Number : 4 GORE MID ELBOW Quantity Required : 1

Make .FSD file (Data) for Assembly

Make .DXF file (3D-CAD) for Assembly

1 | AutoNest, Segment 1/1

Output Selected Pattern(s) as

.CAM FastCAM file(s)

.NC file(s) via FastCAM now

.DXF (2D-CAD) file(s)

.PRN tabulation file(s) Print now

to Drive ... to Directory ... to File ...

c: C:\PROGRAM FILES
FastSHAPES
Data 4 GORE MID ELBOW _hnn.ext

Ask before Overwrite

Output Now Cancel



*In this example we will only be generating the pattern in the FastCAM “CAM” file format. For information on producing **NC Code** refer to topic “**NC CODE GENERATION**”.*

30. Click in the “To File...” field and click on the drop-down arrow. Select the “**OFFSET PIPE**” option.



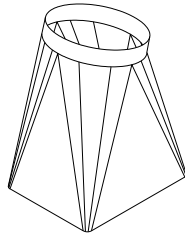
Note how under the “To File...” option, you have 3 saved options for the filename. These options are generated automatically, once you have entered in any Job Data.

31. “Double Click” in the “To Directory...” field and change into the directory where you would like to save the files.

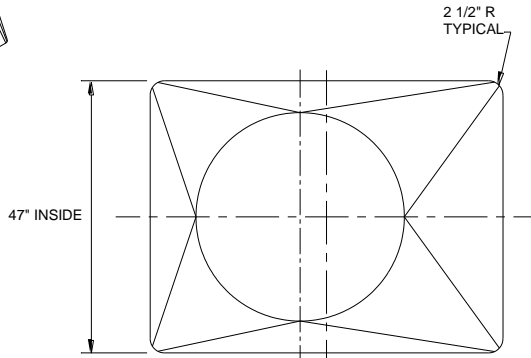


*Try and avoid saving your output from FastSHAPES into the FastSHAPES directory, as this is meant to be for the programs’ internal files **ONLY**.*

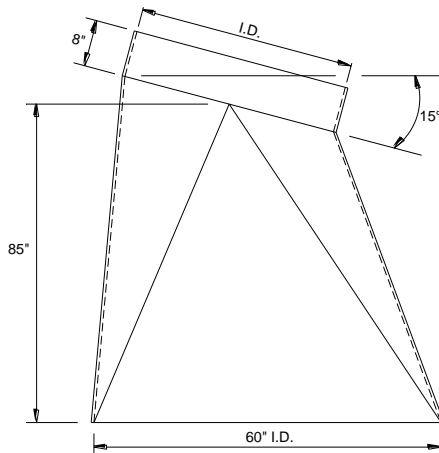
32. Click the “**OUTPUT NOW**” button



PICTORIAL VIEW



TOP VIEW



FRONT VIEW

RECTIRCLE #2	Base = 47" x 60"	Collar = 8"	Material Thickness = 1/4"
	Corner Radius = 2 1/2"		

Further Requirements: Seams to be 180° apart. Split into 2 vertical strakes.

RECTIRCLE #2

1. Run the FastSHAPES® program by “double clicking” the FastSHAPES icon on your Windows desktop.
2. From the FastSHAPES® library, “double click” on the RECTIRCLE icon.
3. Select the JOB data form at the top, by clicking on it.
4. Click in the “Job Number” field and type in “6”, and then press the **TAB** key.
5. You are now in the field for “Drawing Number”. Type in “**T375-R**” and press the **TAB** key twice.
6. You are now in the field for “Assembly Number”. Type in “**RECT375**” and press the **TAB** key three times.
7. You are now in the field for “Prepared by”. Type in your name.



The data screen is not a set of mandatory fields. It is up to the user as to whether these fields are filled in or not. They will however help the user find and retrieve previously saved data.

8. Select the RECTIRCLE data form at the top, by clicking on it.

In this exercise you will produce a Square to Round transition, with a sloped/tilted outlet.

The Rectircle dimensions are:

Base Dimensions: 47” x 60”

Outlet Diam: 24”

Vertical Height: 85”

Segments: 2

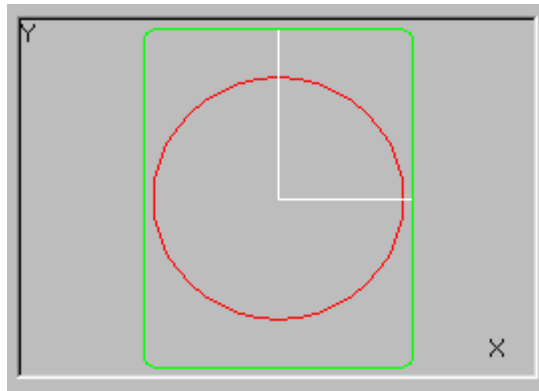
Outlet Slope: 15°

Outlet Collar: 8”

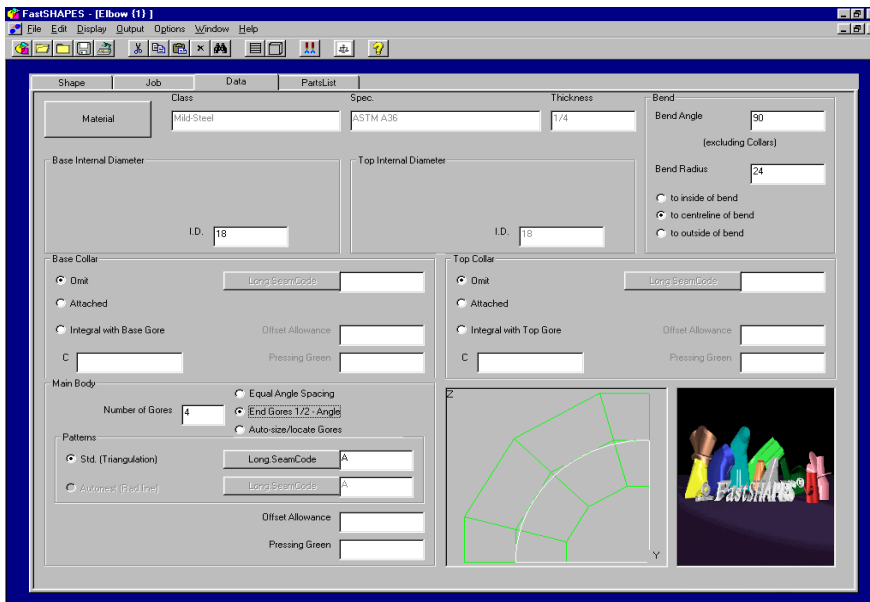
Material Thickness: ¼”

**Further Requirements:
Split vertically into 2 Strakes**

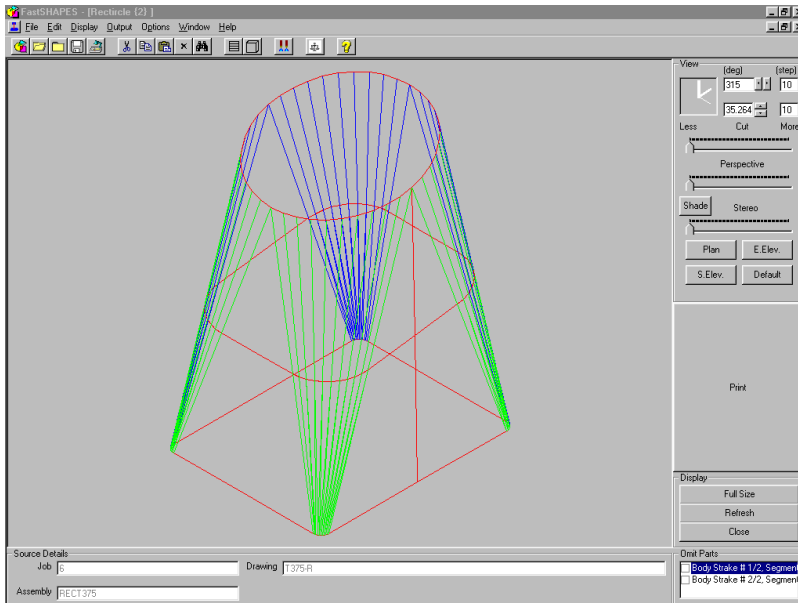
9. Click on the “**Material**” button.
10. Choose a Class and Specification from the Drop Down boxes.
11. Click in the “Thickness” field and type in $\frac{1}{4}$, and then click “OK”.
12. Select the “Rectircle” option under the “**Base Shape and Internal Sizes**”, and press the **TAB** key.
13. You are now in the field for the “A” dimension (X direction). Type in 47 and press the **TAB** key.
14. You are now in the field for the “B” dimension (X direction). Type in 60 and press the **TAB** key.
15. You are now in the field for the “Internal Corner Radius”. Type in $2\frac{1}{2}$ and press the **TAB** key twice.
16. Select the “Rectircle” option for the “**Top Shape, Internal Sizes, Position and Orientation**”, and press the **TAB** key three times.
17. You are now in field for the “Internal Corner Radius”. Type in 24 and press the **TAB** key.
18. You are now in field for the “Z-offset” (Height). Type in 85 and press the **TAB** key four times.
19. You are now in field for the “Tilt”. Type in 15.
20. Now go to bottom left hand corner of the screen under “**Main Body**” and click in the field for “Number of Strakes”. Change the default value of 1 to 2.



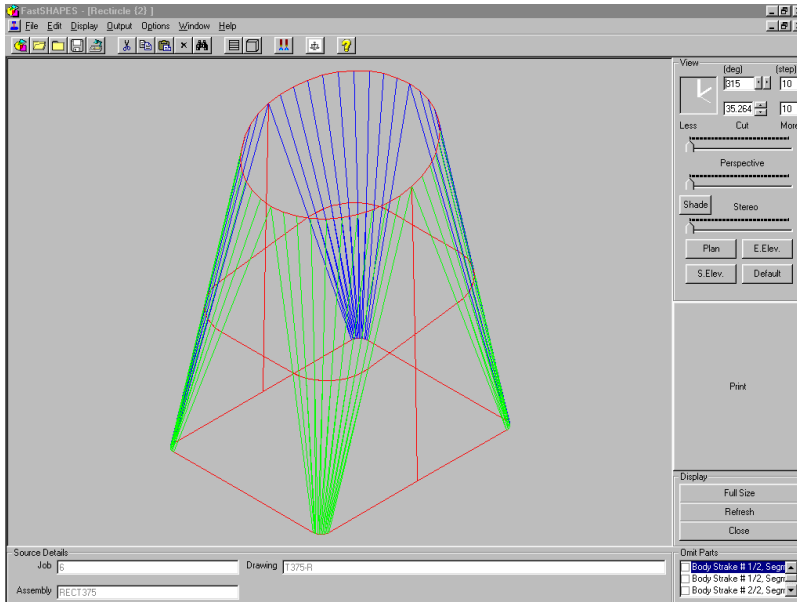
21. Move to the area for “Equally spaced” and CHECK(tick) the box.
22. Move your mouse over the Display Window, and a message will appear: *“Left click for Parts, Right click for Patterns”*, Click your left mouse button.



The screen will now show the Wireframe of the developed RECTIRCLE.



23. Click the CLOSE button, to return to the DATA form.
24. Move your mouse back into the “**Main Body**” area, and click on the “Long. Seam Code” button. Check the options for A & E, then click OK.
25. Move your mouse to the top right hand corner of the screen, into the “**Setout Type**” area, and select the “Simple” option.
26. Move your mouse back to the Display Window, and a message will appear: *“Left click for Parts, Right click for Patterns”*, Click your left mouse button.

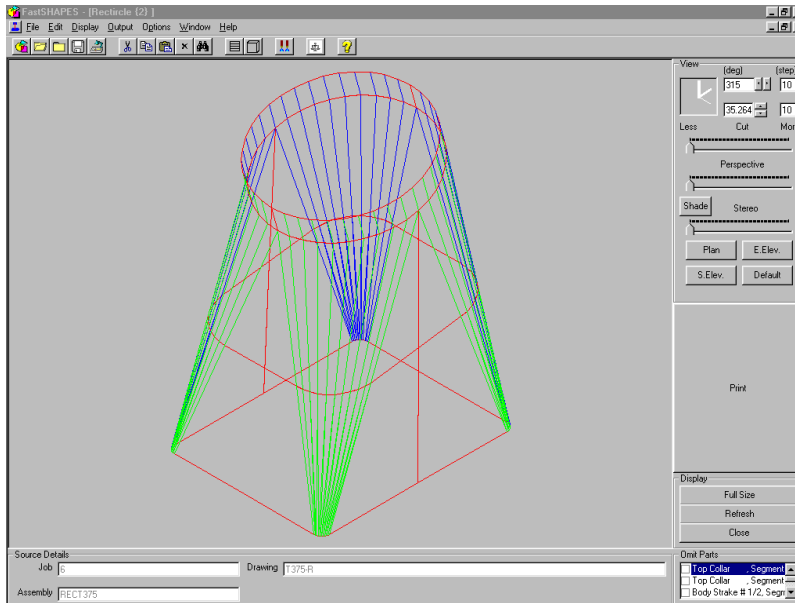


The screen will now show the Wireframe of the Rectircle, with opposing seams.

27. Click the CLOSE button, to return to the DATA form.
28. Now go to the “**Top Collar**” section and select “Parallel – Constant C”. This will now enable the “C & Long Seam Code” options for the collar.
29. Click in the field for “C” and type in 8.
30. Click on the “Long. Seam Code” button. Check the options for C & G, then click OK.
31. Move your mouse over the Display Window and click with your left mouse button.



The screen will now show the Wireframe with an 8” collar attached to the outlet.



32. Move to the CLOSE button, and click your right mouse button.



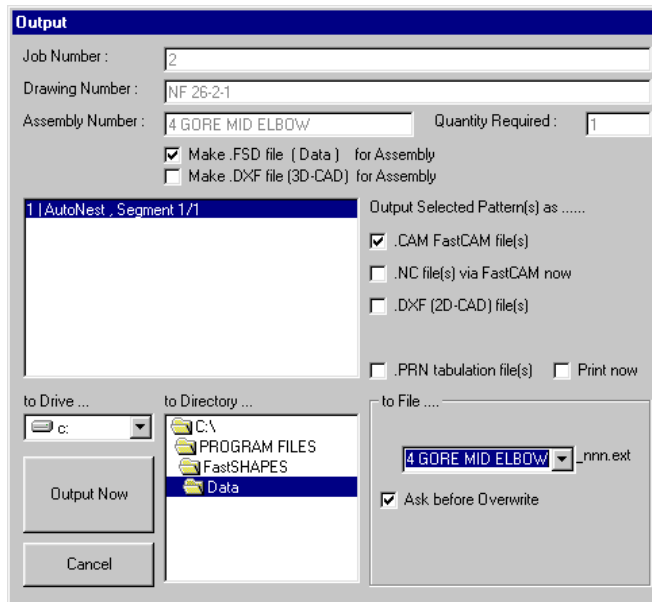
The screen will now show the patterns for the development. You can scroll through all the patterns, by simply clicking on the “NEXT” and “PREVIOUS” buttons under the “Select Pattern” area.

33. Click the CLOSE button, to return to the DATA form.

34. From the menu, select “Output”. The Output form will now appear.

35. Check only the following options:

- “Make .FSD file (DATA) for Assembly”, &
- “.CAM FastCAM file(s)



In this example we will only be generating the pattern in the FastCAM “CAM” file format. For information on producing NC Code refer to topic “NC CODE GENERATION”.

- 36.** Click in the “To File...” field and click on the drop-down arrow. Select the “RECT375” option.



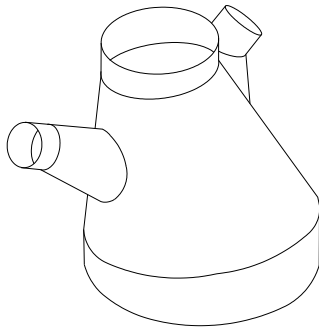
Note how under the “To File...” option, you have 3 saved options for the filename. These options are generated automatically, once you have entered in any Job Data.

- 37.** “Double Click” in the “To Directory...” field and change into the directory where you would like to save the files.

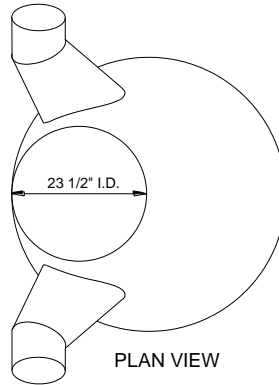


Try and avoid saving your output from FastSHAPES into the FastSHAPES directory, as this is meant to be for the programs’ internal files ONLY.

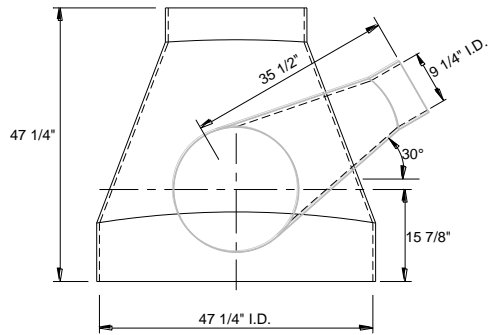
- 38.** Click the “OUTPUT NOW” button.



PICTORIAL VIEW



PLAN VIEW



CBRANCH	Base Dia. = 47 1/4"	Top/Dia. = 23 1/2"	Material Thickness = 1/4"
	Angle = 30° Off Horizontal		Collars = 5" Minimum
Further Requirements:			

CBRANCH

1. Run the FastSHAPES® program by “double clicking” the FastSHAPES icon on your Windows desktop.
2. From the FastSHAPES® library, “double click” on the CBRANCH icon.
3. Select the JOB data form at the top, by clicking on it.
4. Click in the “Job Number” field and type in “7”, and then press the **TAB** key.
5. You are now in the field for “Drawing Number”. Type in “**CONC BRANCH**” and press the **TAB** key twice.
6. You are now in the field for “Assembly Number”. Type in “**2**” and press the **TAB** key three times.
7. You are now in the field for “Prepared by”. Type in your name.



The data screen is not a set of mandatory fields. It is up to the user as to whether these fields are filled in or not. They will however help the user find and retrieve previously saved data.

8. Select the CBRANCH data form at the top, by clicking on it.

In this exercise you will produce a Cone with 2 angled Branches.

The CBranch dimensions are:

Cone Body Base Diam: 47 ¼”

Cone Body Top Diam: 23 ½”

Vertical Height: 47 ¼”

Branch Base Diam: 9 ¼”

Branch End Diam: 22”

Branch Angle: 60°

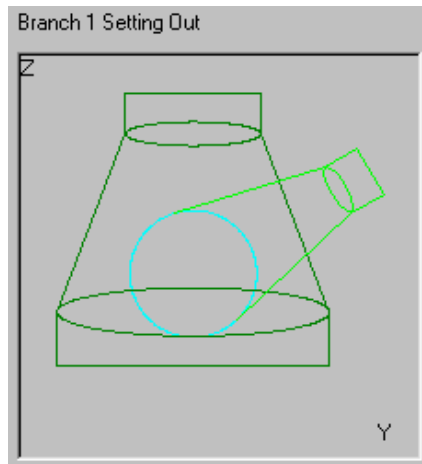
Segments: 1

Branch Collars: 5”

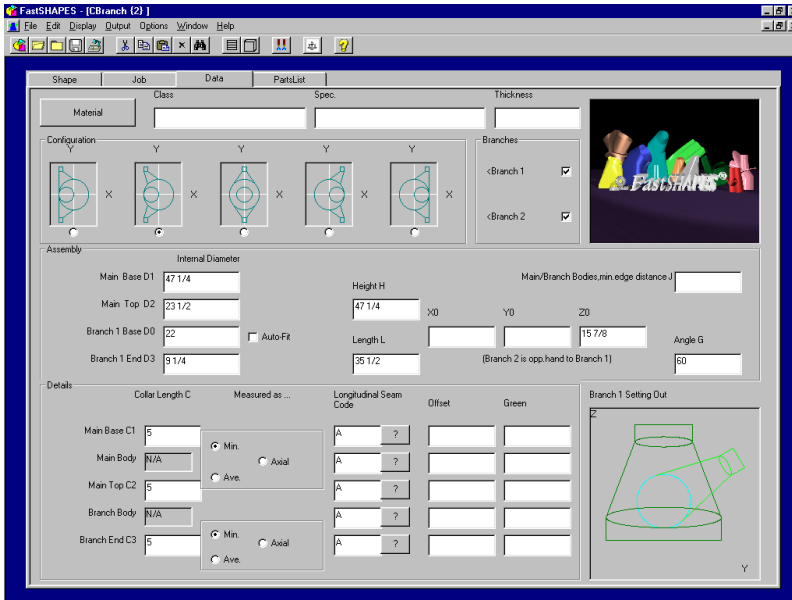
Material Thickness: ¼”

9. Click on the “**Material**” button.
10. Choose a Class and Specification from the Drop Down boxes.
11. Click in the “Thickness” field and type in $\frac{1}{4}$, and then click “OK”.
12. Check, Branch 1 & Branch 2 under the “**Branches**”, area.
13. Select the Angled Branch option in the Negative X direction (2nd option from the left), under “**Configuration**”.
14. Click in the “Main Base D1” field under “**Assembly**” Type in $47 \frac{1}{4}$ and press the **TAB** key.
15. You are now in the field for the “Main Top D2” dimension. Type in $23 \frac{1}{2}$ and press the **TAB** key.
16. You are now in the field for the “Branch 1 Base D0” dimension. Type in 22 and press the **TAB** key twice.
17. You are now in the field for the “Branch 1 End D3” dimension. Type in $9 \frac{1}{4}$ and press the **TAB** key.
18. You are now in the field for the “Height H” dimension. Type in $47 \frac{1}{4}$ and press the **TAB** key.
19. You are now in the field for the “Length L” dimension. Type in $35 \frac{1}{2}$ and press the **TAB** key three times.
20. You are now in the field for the “Z0 – (*the height from the base of the Cone body to the central point of the branch base*)” dimension. Type in $15 \frac{7}{8}$ and press the **TAB** key twice.
21. You are now in the field for the “Angle G” dimension. Type in 60 and press the **TAB** key.

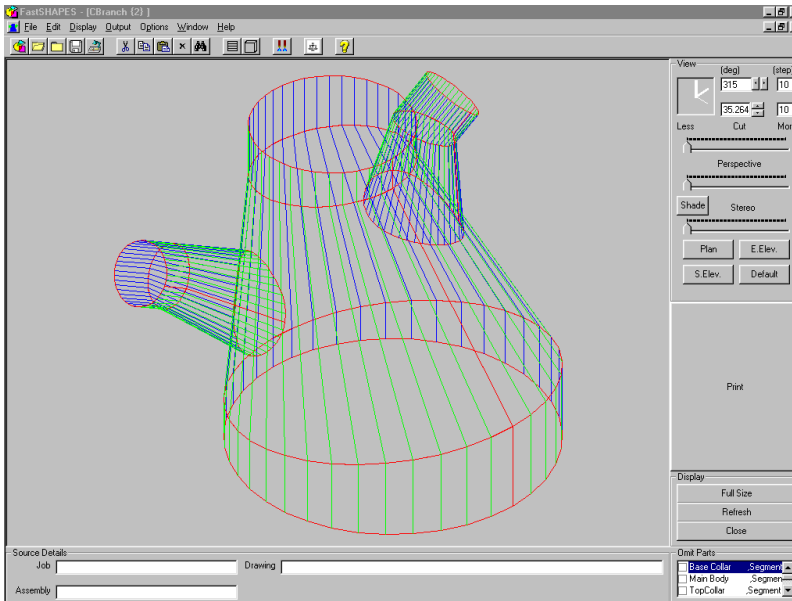
22. . You are now in the field for the “Main Base C1, Collar” dimension. Type in 5 and press the **TAB** key twice.
23. You are now in the field for the “Main Top C2, Collar” dimension. Type in 5 and press the **TAB** key.
24. You are now in the field for the “Branch End C3, Collar” dimension. Type in 5 and press the **TAB** key.



25. Move to the area for “Equally spaced” and CHECK(tick) the box.
26. Move your mouse over the Display Window, and a message will appear: *“Left click for Parts, Right click for Patterns”*, Click your left mouse button.



The screen will now show the Wireframe of the developed



CBRANCH.

27. Move to the CLOSE button, and click your right mouse button.



The screen will now show the patterns for the development. You can scroll through all the patterns, by simply clicking on the “NEXT” and “PREVIOUS” buttons under the “Select Pattern” area.

28. Click the CLOSE button, to return to the DATA form.

29. From the menu, select “Output”. The Output form will now appear.

30. Check only the following options:

- “Make .FSD file (DATA) for Assembly”, &
- “.CAM FastCAM file(s)



In this example we will only be generating the pattern in the FastCAM “CAM” file format. For information on producing NC Code refer to topic “NC CODE GENERATION”.

31. As both Branches are identical, we will only output one of them, to do this, simply choose items 1 to 5 only.

32. Click in the “To File...” field and click on the drop-down arrow. Select the “**CONC BRANCH**” option.



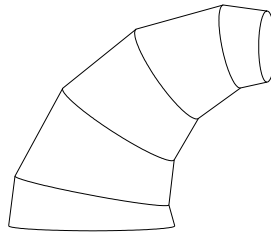
Note how under the “To File...” option, you have 3 saved options for the filename. These options are generated automatically, once you have entered in any Job Data.

33. “Double Click” in the “To Directory...” field and change into the directory where you would like to save the files.

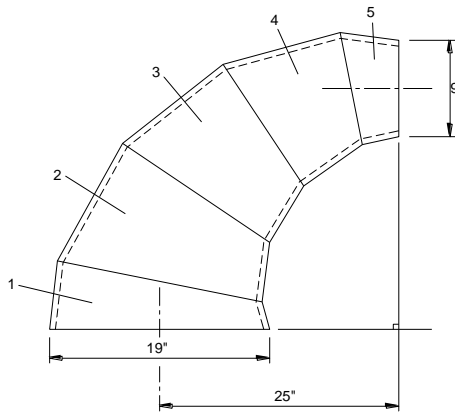


Try and avoid saving your output from FastSHAPES into the FastSHAPES directory, as this is meant to be for the programs’ internal files ONLY.

34. Click the “OUTPUT NOW” button.



PICTORIAL VIEW



LOBSTER	Base/Inlet Dia. = 19"	Bend Radius = 25"	Material Thickness = 1/4"
	Top/Outlet Dia. = 9"	4 GORES	

Further Requirements:

LOBSTER

1. Run the FastSHAPES® program by “double clicking” the FastSHAPES icon on your Windows desktop.
2. From the FastSHAPES® library, “double click” on the LOBSTER icon.
3. Select the JOB data form at the top, by clicking on it.
4. Click in the “Job Number” field and type in “3”, and then press the **TAB** key.
5. You are now in the field for “Drawing Number”. Type in “**LOB-1**” and press the **TAB** key twice.
6. You are now in the field for “Assembly Number”. Type in “**4 GORE LOBSTER**” and press the **TAB** key three times.
7. You are now in the field for “Prepared by”. Type in your name.



The data screen is not a set of mandatory fields. It is up to the user as to whether these fields are filled in or not. They will however help the user find and retrieve previously saved data.

8. Select the LOBSTER data form at the top, by clicking on it.

In this exercise you will produce a 4 Gore, single segment lobster.

The Lobster dimensions are:

Base Diam: 18 ½” ID

Top Diam: 8 ½” ID

Bend angle: 90°

Bend Radius: 25” to Centreline of Bend

Material Thickness: ¼”

Further Requirements:

Seam to be on Back

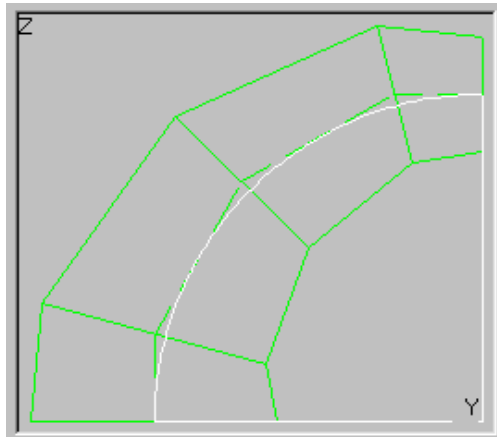
PressGreen: 1”

9. Click on the “**Material**” button.
10. Choose a Class and Specification from the Drop Down boxes.
11. Click in the “Thickness” field and type in $\frac{1}{4}$, and then click “OK”.
12. Click in the field for “Base Internal Diameter”. Type in $18\frac{1}{2}$ and press the **TAB** key.
13. You are now in the field for the “Top Internal Diameter”. Type in $8\frac{1}{2}$ and press the **TAB** key.
14. You are now in the field for the “Bend Angle”. Type in 90 and press the **TAB** key.
15. You are now in the field for the “Bend Radius”. Type in 25 and press the **TAB** key.
16. Select the “to centreline of bend” option for the “Bend Radius”.
17. Now go to bottom left hand corner of the screen under “**Main Body**” and click in the field for “Number of Gores”. Change the default value of 2 to 4, then press the **TAB** key. The display window should now show 4 gores.

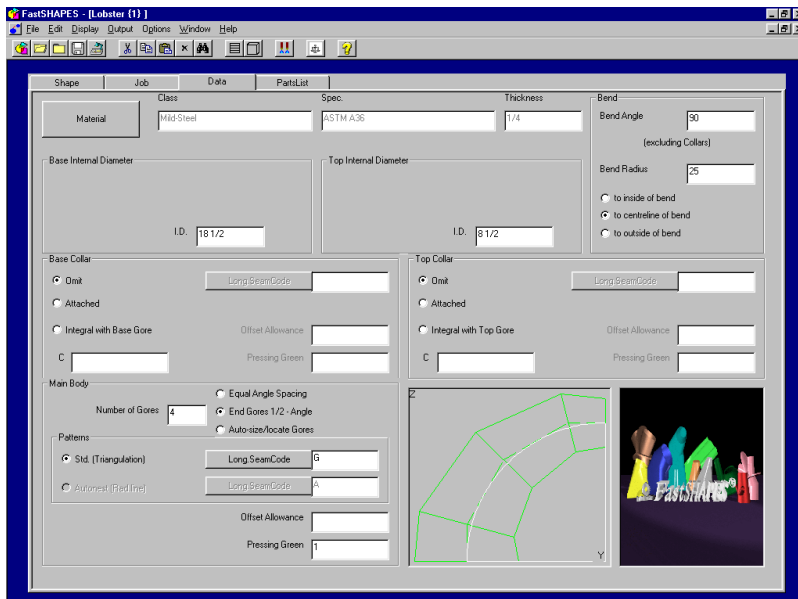
18. Move into the “Patterns” area, and Click the “Long. SeamCode” button.

19. Check Seam G, and Uncheck Seam A, then Click OK

20. Move down to “Pressing Green” and click in the field. Type in 1.



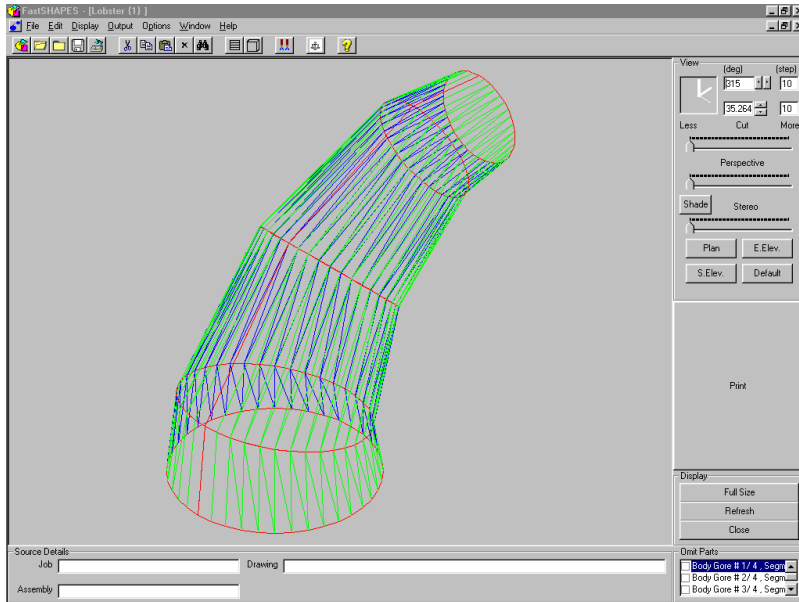
21. Move your mouse over the Display Window, and a message will appear: “Left click for Parts, Right click for Patterns”, Click your left mouse button.





LOBSTER.

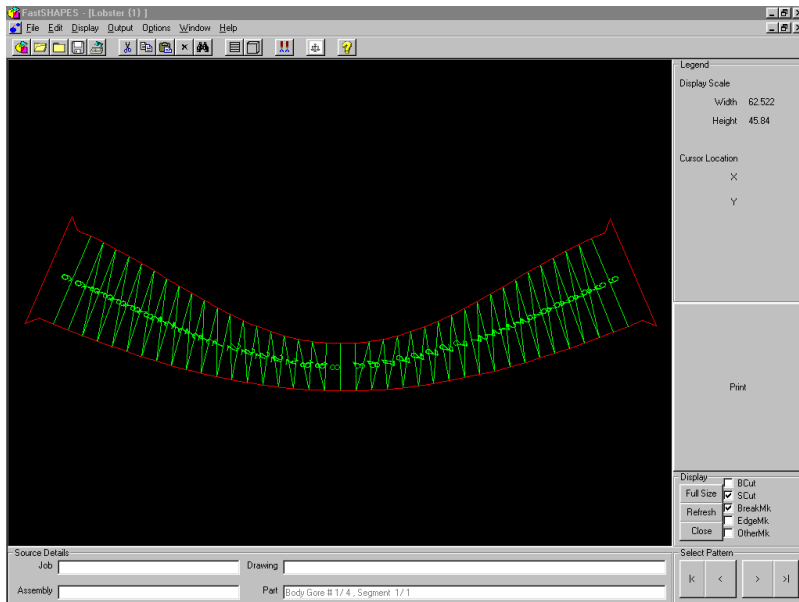
The screen will now show the Wireframe of the developed LOBSTER.



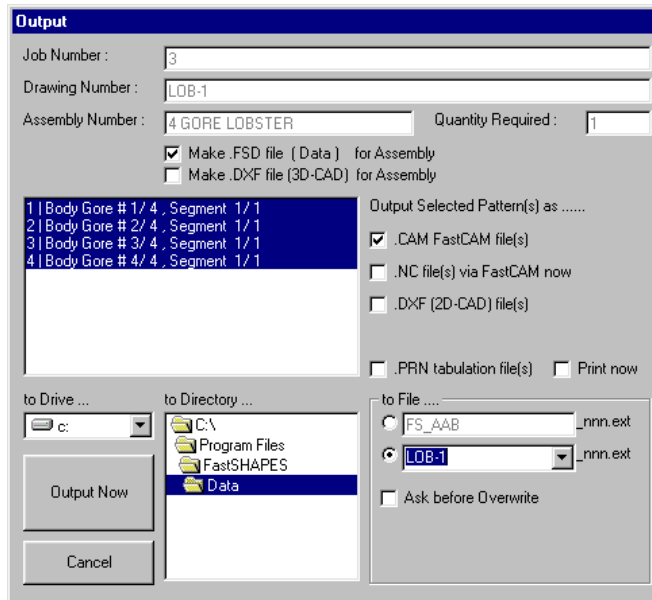
22. Move your mouse over the CLOSE button and Click your right mouse button.



The screen will now show the developed Pattern for the first gore of the LOBSTER. Note the “fishtail” shape on each end, due to the Pressing Green.



23. Move to the Lower right portion of the screen under the “**Select pattern**” area, and click the “right” arrow. This will now show the subsequent developed patterns, 2, 3 & 4.
24. Click the CLOSE button, to return to the DATA form.
25. From the menu, select “Output”. The Output form will now appear.
26. Check only the following options:
 - “Make .FSD file (DATA) for Assembly”, &
 - “.CAM FastCAM file(s)



In this example we will only be generating the pattern in the FastCAM “CAM” file format. For information on producing NC Code refer to topic “NC CODE GENERATION”.

27. Click in the “To File...” field and click on the drop-down arrow. Select “LOB-1”. This will be the name used for the output files.
28. “Double Click” in the “To Directory...” field and change into the directory where you would like to save the files.



Try and avoid saving your output from FastSHAPES into the FastSHAPES directory, as this is meant to be for the programs’ internal files ONLY.

29. Click the “OUTPUT NOW” button.

Development and Release History and Notes

The genesis of FastSHAPES was a suite of 'program-writing-programs' written during the mid. 80's whose concept was not much different from today's FastSHAPES, but whose output was restricted to NC code. A few of the suite had earlier been written informally on 8-bit machines operating under CPM, using a basic interpreter.

In the early 90's, the suite was rewritten and expanded, running under DOS, with each program in the suite existing as a standalone program. In those pre-Win95 days, the programs were written to use dual monitors to facilitate simultaneous data entry and graphic display, and included a number of features now made largely redundant under Win95.

In 1997/98, the suite of DOS programs was again rewritten, and amalgamated as a single Win95 application now known as FastSHAPES Standard Edition. Redundant features have been eliminated. A new PartsList feature has been added that provides significantly more estimating and production control data than was previously available.

A succession of informal releases was made over time as the conversion proceeded. All releases prior to August 1999 were identified as Version 1.0.0, and documentation available was limited to that prepared for the DOS antecedents.

The December 1999 release has been upgraded to Version 3.2.3 and is issued with this new documentation.

An on-line help system has yet to be prepared, and further additions to the suite of FastSHAPES are planned over the next 12 to 24 months.

The only significant change between V3.0.0 and V3.2.0 has occurred in Sprocket / Sprocket + where a new specification has been added, and the data details under Sprocket + generic Types A and B has been modified slightly. Where formulae made reference to roller diameter, this has been replaced by reference to pocket diameter. The effect of the difference on existing data is minimal at worst, and unlikely to have any detrimental effect if old data is reprocessed, nevertheless the user should be aware of the fact.

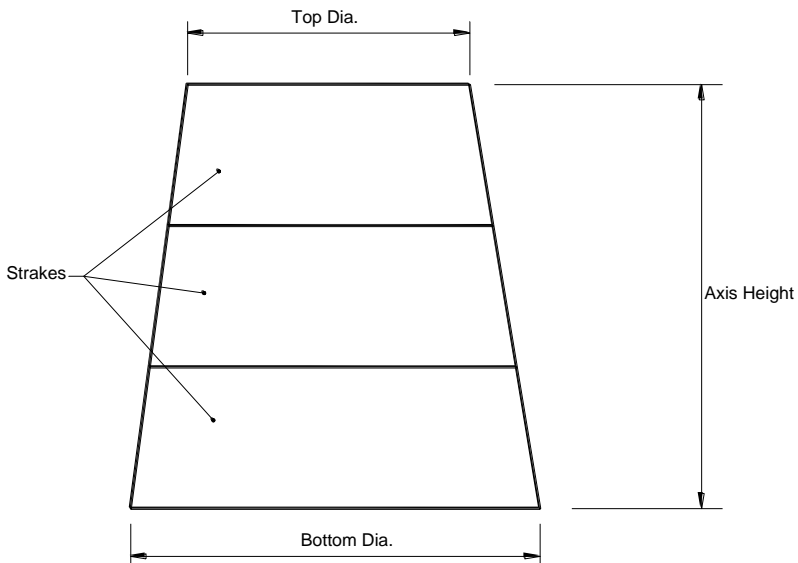
Minor corrections or changes to terminology have been made throughout the software to bring it into line with this manual, and to improve consistency between the various FastSHAPES.

A p p e n d i x



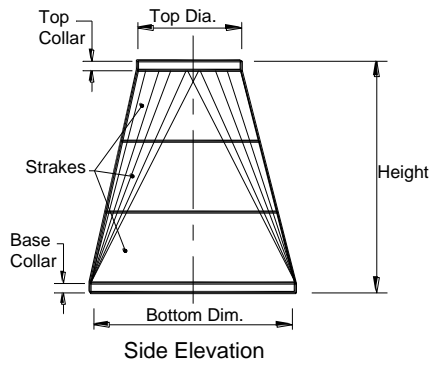
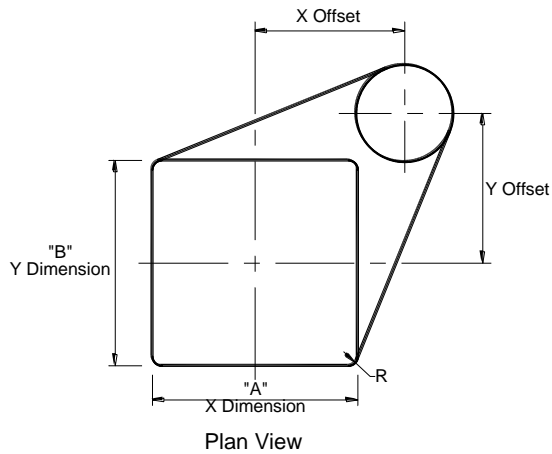
The following diagrams represent the “Data Layout” for majority of the Shapes, and are designed to aid the user in the Data Entry process.

APPENDIX A

Cone

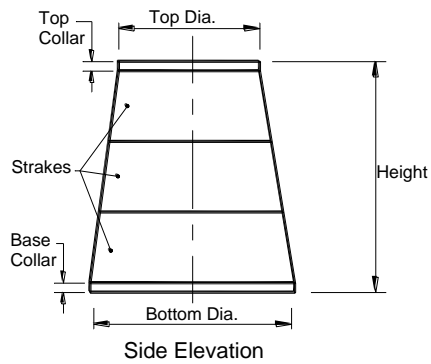
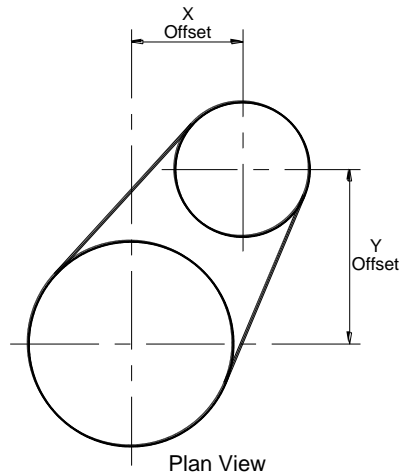
"Data Layout"

Rort



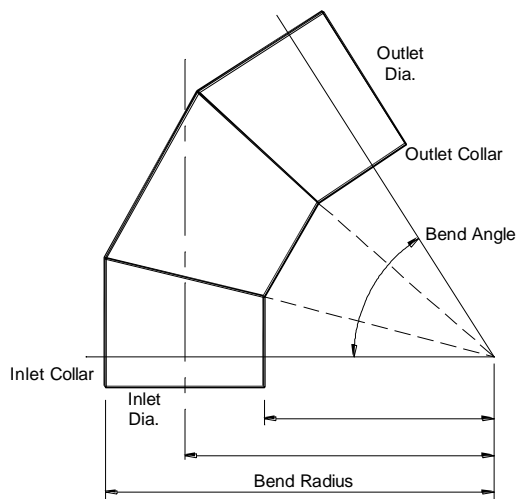
"Data Layout"

Obcone



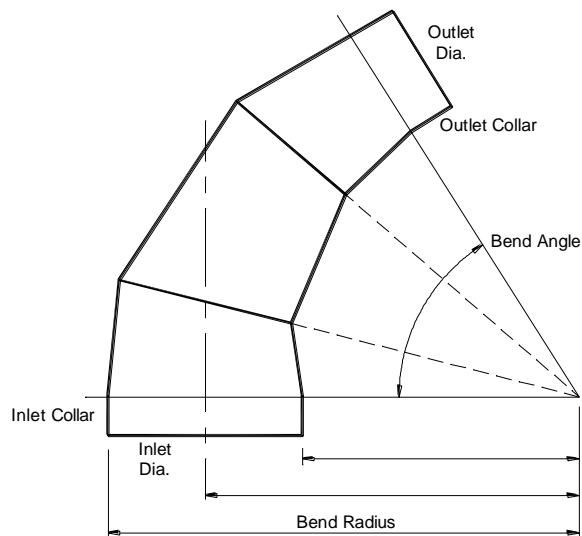
"Data Layout"

Elbow



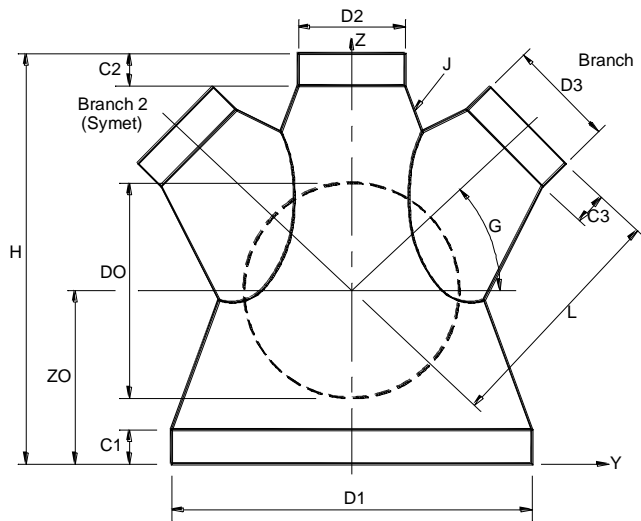
"Data Layout"

Lobster



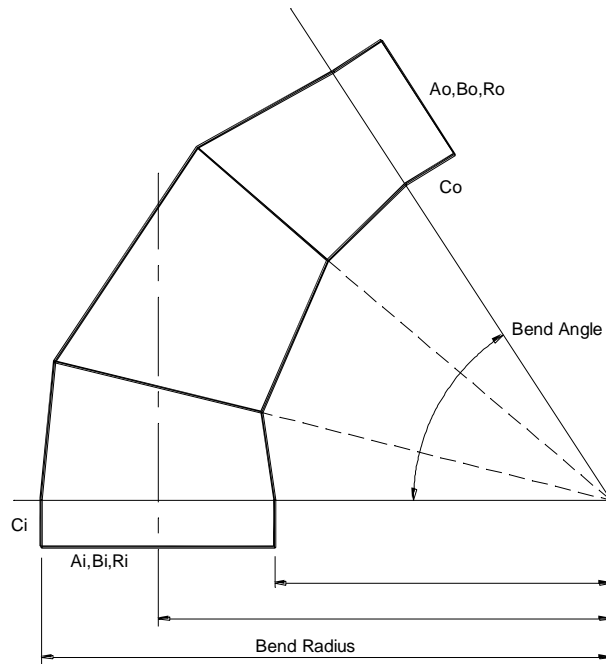
"Data Layout"

Cbranch



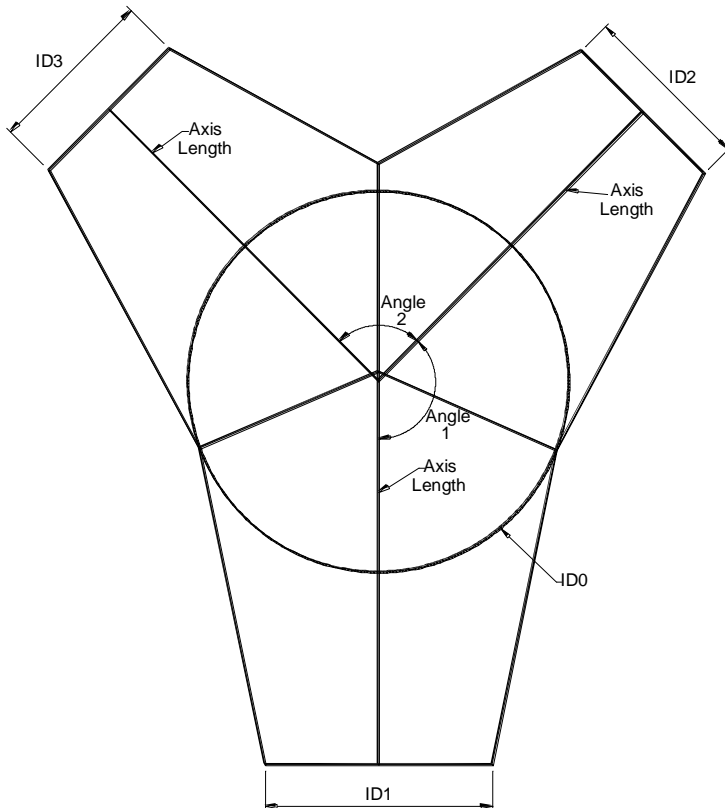
"Data Layout"

Bend



"Data Layout"

Bifurcate



"Data Layout"

A p p e n d i x



APPENDIX B

THE TOOLBAR

The following is a list of definitions for the Toolbar used in FastSHAPES.



Select a new FastSHAPE, or Display the Shape Library



Select and Open a Data file



Close the current FastSHAPE



Save the current Data to file



Close FastSHAPES and exit to the Desktop



Cut



Copy



Paste



Delete Selection



Search for text



Display a 2D Pattern



Display a part (3D wireframe)



Make Output File(s)



Change FastSHAPES Configuration



Help System

Glossary

ADDENDUM: The material ABOVE the pitch line as defined by the appropriate Standard.

ANGLE: The relation of two straight lines originating from a single point.

ARC: A Continuous portion of a circle.

CHINE: A ‘knuckle’ joint between 2 adjacent surfaces or elements of a ‘surface’ in which the curvature is smooth and continuous along the length of the joint, but may be discontinuous transversely across the joint.

CNC: Computer Numeric Control used to control flame cutting machines, milling machines, lathes and other computerized metal working equipment.

CONCENTRIC REDUCER/CIRCLES: Two circles, which have common center points, but differing dimensions.

CONE: A solid object, which has one circular end and the other end tapering towards a point

DEDENDUM: The material BELOW the pitch line as defined by the appropriate Standard.

DIAMETER: Refers to the distance across a circle, passing through the center point.

DXF Files: Files produced by Autocad and other CAD systems, which are used to transfer geometric information from one computer system to another. The DXF interface allows the geometry of the parts to be drawn/generated on the CAD system, and the information then can be transferred to another system for verification, modification, production.....etc.

ECCENTRIC REDUCER/CIRCLES: Two circles, which have no common center point.

ELLIPSE: A curved figure with its Major axis longer than its Minor axis.

GREEN: (Press/End or Edge Green) Extra material attached to the ends/edges of an object/pattern. Usually required to, facilitate the forming process.

OBCONE: Oblique Cone. Also known as an Eccentric Reducer.

OBROUND: An elongated hole. The ends are made up of half circles.

PROGRAM PATH: Found in FastCAM® and is used to set the direction of kerf, order and direction of cut line, placement of lead ins and lead outs, and defines the cutting/marking process.

QUADRANT: An arc of 90 degrees, or one quarter of a circle.

RADIUS: Distance from the center of the circle to the outer edge of the circle. Exactly one half the Diameter.

RORT: Rectangular to Offset Round Transformer

SPHERE: A three dimensional surface, where all points of which are equidistant from a fixed point.

SURFACE: The boundary of a three-dimensional figure.

TANGENT: The point at which a line blends to an arc or circle.

Additional Software Modules by FastCAM

Interactive Nesting Module®:

Simple Drag & Drop part nesting with, dynamic part rotation, interactive arraying and high speed block nesting, including plate remnant nesting.

FastNEST®:

Fully automatic true shape nesting module for bulk processing. Highly optimizable nesting system with inbuilt FastPATH® automatic pathing system.

FastPATH®:

Automatic & Intelligent Pathing Module. Including DXF/IGES interface for CAD with automatic batch processing for FastNEST®. FastPATH® costs CAD DXF nests and parts instantly. Generates NC code immediately. Eliminates hours of manual pathing. Two click positive and negative Bridging. Dramatic improvement in machine performance, saving up to half the cutting time. Dramatic savings in plasma consumables.

FastTRACK®:

Plate and remnant plate tracking system. Accurate, simple and fast tracking of all remnant by exact shape, grade and thickness. Find Plate - rapid search of remnant stocks for a specific size, material, grade and thickness - find that plate fast! Stand alone product or FastCAM® compatible for automatic updating from nesting system.

FastBEAM®:

Fast graphic programming system for Beamline equipment supporting Peddinghaus, Wagner, Ficep. Accepts output from most 3D structural CAD systems and DSTV generally. Beamlines used for the drilling, punching and cutting of rolled sections, such as Beams, Channels, Angles and Flat Bars.

FastCOPY®:

Direct NC digitizing and optical template conversion system. With FastCOPY® it takes a programmer no more than 5 to 10 minutes to digitize a complex and large profile very accurately, including all holes, fillets, bends and the like. Parts can be digitized to any chosen scale. FastCOPY® is unique, worldwide in its operation, simplicity and accuracy.

FastCUT®:

Simple to use! Designed to find the optimum layout of parts to be cut from any flat sheet material or lengths from rods and bars. Provides Clients with quick and accurate quotes. It even takes remnant into account. Anyone who has a need to shear flat SHEET or BAR material is a potential user of FastCUT®.

FastCUT® Cut to Length (CTL):

Software program that enables you to instantly calculate ideal blank sizes from coil based materials. Ideally used by large manufacturer or Steel Service Center who use large quantities of coil for processing. Results can be optimized for ideal single length blanks or mixed lengths.

FastLINK

Upload and Download NC files from Machine or PC Depending upon controller type. Background or foreground program operation under Windows. Also multiple machine support available.

FastTRAN (DOS)

Universal translator for NC Code files to NC Code. Support for over 27 Controllers and multiple process. Need to convert ESSi files into EIA code then FastTRAN is a must.

International Offices

Customers can obtain additional information on the FastCAM® product range by contacting your Local Reseller or from any of our International Offices. They offer high quality and expert service.

The address of our internet site is

www.fastcam.com

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8700 West Bryn Mawr,
Suite 800 South

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Tel: (312) 715 1535

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FASTCAM China

No 34, 377 Chenhui Rd Bldg
High Tech

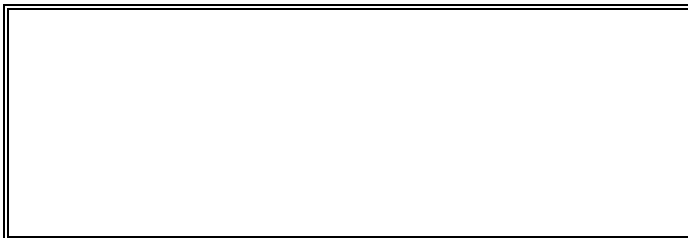
Pudong, Shanghai 201203 China

Tel: (8621) 5080 3069

Fax: (8621) 5080 3071

Email: fastcam@fastcam.cn

Or contact your local Reseller



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