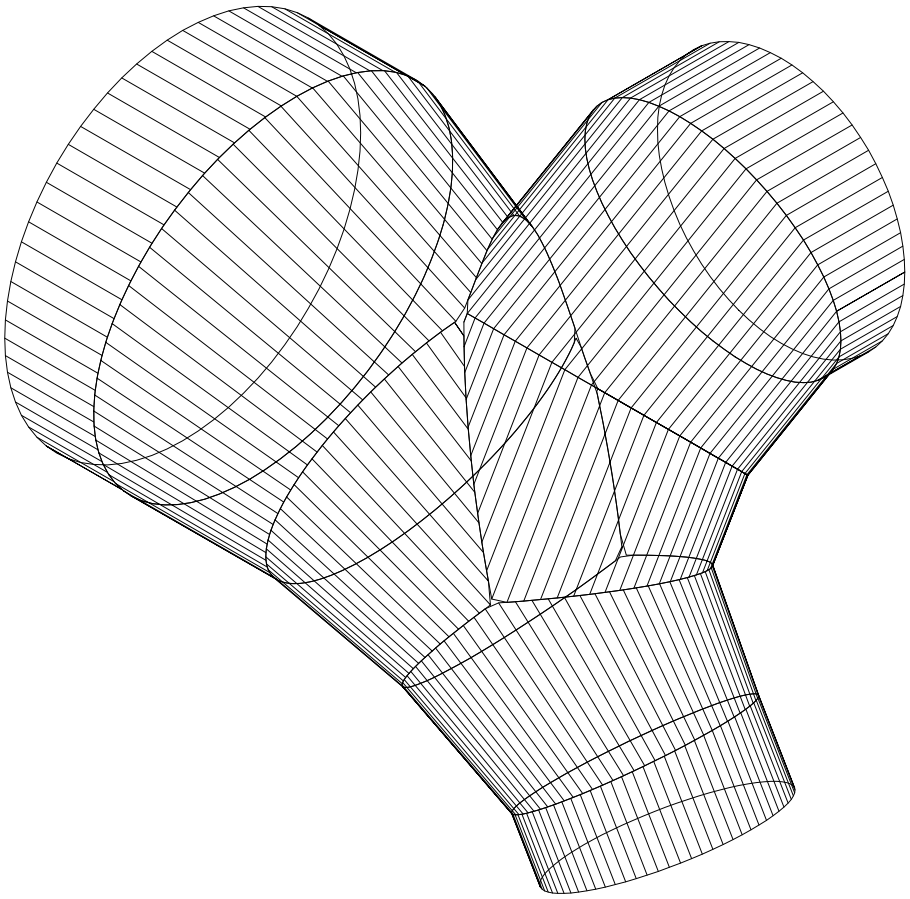
SHEET LIGHTNING TITLE PAGE



CONTENTS

1 INTRODUCTION

1.1 Sheet Lightning - The Tool

The subject of sheet pattern work is undoubtedly one of those areas of study that falls into the categories of both an exact science and, at the same time, a fundamental art. There is an exact solution to every problem and yet the techniques and method used to obtain the solution are more like an artistic phenomenon. As always there are problems and limitations which the craftsman must learn to live with but as with every other area of study the frontiers are continually being pushed back as the study advances and as the tools are developed to assist with the task.

SHEET LIGHTNING is a revolutionary tool for the sheet pattern worker. As a tool for the craftsman, it considerably widens his design scope. It combines the ability to achieve the simple pattern work quickly and easily, with the power to handle the complex with very little extra time and effort. The work can be organised and saved without a single template lying around the workshop. The one-off problem is no longer a problem.

Sheet lightning operates as an exact science using a sophisticated mathematical model, invisible to the user, which produces patterns that are, for all intents and purposes, exact. At the same time the pattern worker is given a friendly and attractive 2D or 3D environment to work with that takes him/her a long way beyond the limitations of the drawing board and opens up the scope of design, even exploiting the artistic elements of the work. Within a very short time you will begin to feel at home with the system as you discover the facilities at your finger tips just as if they were learning to use any other engineering tool for the first time.

1.2 Jargon

A common fear for the computer novice is in the apparently complicated jargon that has built up around the subject of computers. Some of this is unavoidable because things have to have names. However to operate a user-friendly software package this should not be much of a problem. Only the bare essentials should be required and a very general understanding of the way computers function. On the other hand each package or application must have some jargon of its own. What the package does and how it does it often needs words and names to be invented in order to describe it. SHEET LIGHTNING is no exception to this but remember that this jargon would be as unfamiliar to the computer expert as to the novice. In fact, in the case of SHEET LIGHTNING much of the jargon is more relevant to the study of sheet metal pattern work than to the study of computers. A little use of the system will soon bring familiarity.

1.3 Libraries

SHEET LIGHTNING is not limited to the selection of simple standard patterns from libraries of data. Its mathematical base makes the system much more sophisticated in that it can deal with an infinite variation of patterns and intersections, including multiple intersection of more than two elements. In fact the designs supplied on disc are simply given as examples.You can easily build up a library of designs although experience has proved that the process of creating a design is often too simple to even merit the storage of the design file. To repeat the creation of a design is often even quicker and simpler than locating and loading it from disc.

1.4 This Manual

The first four sections of this manual are intended to give all the introduction that is needed to each aspect of the package for you to get well and truly started. They include an introduction to relevant aspect of the computer, an introduction to sheet metal pattern work and an introduction to SHEET LIGHTNING itself. The more complicated facilities are better discovered with practice and by following the procedures described in the manual. A quick reference is provided and in many cases will be enough to give get you working with the various facilities.

1.5 What's On The Disc

The following is a the complete list of files supplied on the disc with a description of their purpose.

(a) **SHEET.EXE -** The SHEET LIGHTNING program.

(b) **INSTALL.EXE -** This is the facility for installing the SHEET LIGHTNING program on a hard disc. To run the installation when the system prompt appears (e.g A> or C>) type "INSTALL".

© ******.BGI -** These are the drivers necessary to drive various graphics adapters. Note that the EGA/VGA adapter is not included because it is permanently linked in to the software. Make sure that the required driver available in the current directory if the graphics adapter is not EGA or VGA.

(d) **README.DOC -** A document which contains any information supplemental to the publication of the manual. The file can be read by using the - TYPE filename - at the system prompt A> or C>.

(e) ******.GRD -** Example designs supplied with the software. These can be loaded, examined and used by SHEET LIGHTNING:

(f) ******.PAR -** Parametric design files defining the variable parameters to be attached to a design file. The design file it relates to has the same name as the PAR file but with a GRP file name extension.

(g) ******.GRP** - A parametric design file of the same format as GRD files. These files are associated with a PAR file of the same name. Together they define a parametric design file.

(f) ******.HGL -** Example HPGL plotter files created by SHEET LIGHTNING. These can be passed to a plotter by any suitable system or used in other appropriate ways such as loading them into a word processor for producing reports with images. Of course the system must support HPGL. SHEET LIGHTNING can be used to pass data directly to a plotter if required.

(g) ******.DXF -** Example data interchange files. These files may contain 2D or 3D data which is held in a standard format that can be imported into other CAD systems.

(h) **SHTRIAL.EXE -** This is a demonstration version of SHEET LIGHTNING which has been freely distributed for demonstration and evaluation purposes. All graphics output capabilities are removed. This version can however be used for creating design files or graphic arrangements that can be despatched later by the full system. The demonstration version is undongled and can be copied and passed on for evaluation purposes.

2 THE COMPUTER

This section is not intended to cover the general operation of computers but rather to address a few brief aspects specifically relating to SHEET LIGHTNING. For more information on the computer system, consult the computer manual.

2.1 Drivers

SHEET LIGHTNING is a DOS program. It has an EGA/VGA graphics driver linked permanently into the software. If the system you are using is not EGA or VGA then it is essential that the necessary driver exists in the current directory when the software is run. These other drivers enable SHEET LIGHTNING to operate on other systems. If the necessary driver is not found to be available, an error is reported and execution halted. Super VGA drivers are not currently supported.

2.2 Maths Co-processor\Emulation

SHEET LIGHTNING will automatically detect the presence of a maths co-processor in your computer system. If a processor is not installed the program should still run satisfactorily due to the inclusion of an emulator although at a much reduced speed. As with most graphics software, a co-processor is strongly recommended.

2.3 Security Dongle

If you get a security lock error when attempting to run the software make sure that the security dongle or disk is correctly installed. The dongle should be installed in the parallel port LPT1.

2.4 The Mouse

The mouse is the main device used to operate the software so it is essential that the MOUSE.COM utility program supplied with your DOS system or mouse driver has been executed to initialise the mouse before SHEET LIGHTNING is run. Consult your system manual for more details.

Almost every feature in SHEET LIGHTNING can be accessed without pressing a single key on the keyboard. The use of the keyboard is given rather as a short-cut option. The sensitivity of the mouse to movement can be adjusted to preference from within SHEET LIGHTNING. The procedure is given in the discussion of the “Options” menu.

2.5 Keycodes

When using SHEET LIGHTNING, pressing the left-hand button on the mouse has exactly the same effect as pressing the <Enter> button on the keyboard. Pressing the right-hand button is equivalent to pressing the <Escape> button on the keyboard. Movement of the mouse produces the same effect as pressing the directional keys on the keyboard. Various hotkeys are also provided to give quick access to menus and other facilities.

2.6 Screen Resolution

SHEET LIGHTNING will automatically detect and use the highest standard graphics mode available, giving the highest possible screen resolution. Super VGA modes are not generally supported.

2.7 Files Extensions

The main design files produced by Sheet Lightning are the GRD files from the design system and the GRP/PAR files from the parametric system. The GRD and GRP files have exactly the same format but a GRP extension indicated that it is a parametric template design fle and should therefore have an accompanying PAR file of the same name. Other files produced by the program are either output or configuration files.

Of course, the Sheet Lightning program and demonstration program are both executable program files, therefore having the extension "EXE" and the demonstration files used by the demonstration program have extension "DEM". The following summarises:

GRD : Sheet Lightning design file.
GRP : Sheet Lightning parametric template file.
PAR : Sheet Lightning parametric text file.
DXF : Data interchange file
CFG : Configuration file.
EXE : Program files.
HGL : HPGL plotter file.
MNU : Menu text file.
HLP : Menu help file.

Further discussion of these files, including methods of naming, loading, saving and their purpose are discussed further in the appropriate place.

2.8 Output Devices

The main output device for SHEET LIGHTNING is the plotter, which is used to reproduce

2D and 3D drawings and full size plots of sheet patterns. The language used is HPGL which has become a virtual standard plotter language although others do exist. Most plotters have the capacity to run on HPGL although they may also support other languages. In this case it is essential that the plotter is set to the correct language mode.

If the plotter is connected via the serial port SHEET LIGHTNING can be configured to communicate with the device thus reading the size of the plotter. Otherwise limits must be set manually (see despatch).

Numerical output may also be despatched to a printer to allow patterns to be plotted by hand. This may be particularly useful for exceptionally large patterns although SHEET LIGHTNING can partition a plot over any number of sheets if the plotter is not large enough to accommodate the whole sheet.

If problems are experienced with device interface there is a further option of despatching data in any form to a file which can in turn be passed to a device using some third party utility.

3 SHEET LIGHTNING

3.1 Description

SHEET LIGHTNING is an engineering design and production tool. Its main application is the production of flat patterns in sheet metal work which is an area of considerable interest in the manufacture of flues, ducting, chimneys, chutes and a host of other sheet metal products. SHEET LIGHTNING can be used to make the easy even simpler or to open the door to the development of complicated multi-intersection design patterns which would be a nightmare to attempt with conventional drawing techniques. Of course there is much to be said for keeping design simple where possible and using standard pieces, but at the same time there is much to be said for having the scope and capability to go beyond where necessary and to be able to address the one-off problem with ease. Either way, simple or complex, SHEET LIGHTNING is the answer to the problem.

3.2 Design

The system includes a comprehensive design facility tailored for the purpose, which allows you to work in either a two or three dimensional environment. This makes the design easy to visualise and allows quick alterations before despatching the result to an output file or device.

3.3 Operation

Using the package demands very little knowledge of the workings of the computer. In fact virtually everything can be done using the computer "mouse". The manual gives a step by step guide to discovering how to achieve any design. A little practice is the main requirement.

3.4 Hotkeys

The term "hotkey" describes a key which can be used usually as an alternative to using the mouse for performing a certain function. These keys should be regarded as shortcuts and will become more useful when you are reasonably familiar with the software

3.5 Active/InActive

Colours are often used in the software to give extra indication as to whether a function is active or not. A black or cyan background is a usual indication that the function is inactive.

Magenta or red indicates active.

3.6 Input Box

At certain times you will be "prompted" for typed input, sometimes in the form of a name or perhaps a value. This is always done in the form of an input box which pops up on screen with a title that indicates what the box is prompting for. An example of this is found in loading a file. The title given at the top of the box will be "Load File Name".

When the box pops up it may already contain a value. This is usually the present name or value of the prompted item which, if it has not previously been altered will be the default value or name. If it is not necessary to change this value or name then the left-hand mouse button can be pressed to accept it. If a change is required, then the first character entered will cause the name to disappear. Only the typed character and the flashing cursor will be left indicating that the computer is awaiting further typed input. When the new value is typed into the box, press the left-hand button to accept the new value. If the input was in any way "illegal", as for instance when letters are typed in response to a request for a numerical value, then the input will be rejected and an error reported. If the right-hand button is pressed when an input box is on screen it will disappear and the current operation will be aborted.

3.7 Numerical Input

In many situations an input box will automatically appear on screen to accept numeric input immediately when a numeric key is pressed (i.e."0..9","+","-" or "."). This will only occur when it is logical to do so depending on what procedure or operation is currently taking place in the design area. When the input box arrives on screen it will contain just the character/s that were pressed to initiate the input box. The heading to the box will indicate its purpose.

The F5 hotkey can also be used to initiate an input box in such a situation. Its effect is to invoke an input box just as though numeric typing had begun but with the existing value in the box. The advantage of this is that the F5 can be used to examine the current value before or without changing it.

3.8 Verify Box

A verify box is similar to the input box except that the title given to the box is always "Verify" and its purpose is not to request input but simply to ask for some choice to be made. This will often take the form of a Yes or No answer with the Y/N keys being indicated as the possible choices. No response is made until a "legal" key is pressed or the escape button

which will cause the current operation for which the box appeared to be aborted.

4 SHEET LIGHTNING PATTERNWORK

The whole study of sheet metal pattern work revolves around the need for the manufacturer to be efficient, practical and not unnecessarily complicated. To achieve this end a whole study has built up to address the question of how to establish the flat shape of material which must be cut so that when it is formed in one way or another it will give the desired result.

4.1 Forming

Forming is the process of taking the sheet pattern and shaping it into the product it is intended to become. A common way of achieving this is by using a press. Such a process will bend and stretch the material into its new shape. In such situations pattern work is often not required, the product is simply pressed out of the flat sheet and then the waste is cut away later. In other cases the pattern work for such a process is very complicated because it attempts to predict the way in which the material will be distorted which inevitably depends upon the process used to form it. It other cases it is very difficult to produce the object from flat sheet therefore other forming techniques have been developed, such as "extrusion" where the original material is not made from sheet at all.

It may not come as a surprise to find out that the equipment needed for such operations can be very expensive and is often very inflexible in that it cannot be adapted for a variety of products. Therefore the one-off product is often out of the question, which leads us back to the drawing board and to the problem of manufacturing the required product in the simplest and most efficient way possible. The answer, for many products at least, is in sheet metal pattern work.

4.2 Rolling and Folding

The aim of sheet metal pattern work is to accurately predict the flat sheet pattern shape of a product so that it can be cut in its flat state and then formed into the product by a simple process of rolling and folding. These forming techniques are the simplest available, the equipment needed is relatively inexpensive and can usually be applied to any product of this kind.

4.3 Geometrical Drawing Techniques

The study of sheet metal pattern work has gone a long way in developing geometrical drawing techniques to produce a great variety of products. Most draughtsmen are familiar with the basic methods such as the parallel line method, the radial line method and

triangulation. Of course not all draughtsmen are involved in a branch of engineering which demands the use of these techniques but they have become an integral part of the training program. Many draughtsmen therefore appreciate the complexity that can be encountered in some of the more difficult situations particularly in the case of awkward inter-penetrations between objects. The study of the subject is quite systematic and logical but is often strangely dependent on an intuitive sense which develops by experience in dealing with new problems.

4.4 Complexity

Whether for reasons of the complexity of the problem or just the amount of time and work needed to produce patterns, the pattern-maker is limited in the problems he can handle. For example the intersection of an oblique cone with a second offset oblique cone has been known to cause a few headaches. Imagine adding a third, and this with a centre line not necessarily in the same plane. Who can blame the draughtsmen for keeping it simple? It would hardly be worth the effort.

4.5 Templates

Because of the time and effort needed to produce patterns, sheet metal workers tend to keep an inventory of patterns. These can often be seen hanging all around the walls of the workshop or stacked in cupboards waiting to be summoned. Finding the right template often poses almost as big a problem as producing it, especially when associated pattern templates have to be identified. Of course, such a system must place limits on the variety of design unless an overburden of templates is to be kept.

4.6 Accuracy

One factor that is not always appreciated, is the inaccuracies that are introduced when using certain geometrical drawing techniques, or the inaccuracies that can arise from making an "unwise" decision. For example, the use of chord lengths over differing radii, or in choosing how to segregate a component that must be regarded as a combination of shapes or pieces in order to produce it in a single piece. These inaccuracies are sometimes inherent in the method used but are often only slight and therefore can be accepted, or they are introduced unnecessarily by the way the draughtsman handles the problem. In any case, the accuracy is dependent upon the time and care taken by the draughtsman in using the techniques.

The level of inaccuracy that can be accepted often depends on the thickness, and therefore the stiffness of the material to be used. A higher stiffness is less forgiving to inaccuracy. In addition, thicker materials will tend to distort more in the rolling process, therefore pattern work tends to be applied to the thinner materials. When the materials are thick enough to make this kind of distortion significant, cutting is often left until after the material is rolled. It is usually inconvenient but nevertheless necessary. The scope of pattern work application

is therefore narrowed a little but is still extremely wide.

4.7 Automation

Many advances have been made in the automation of sheet metal work, particularly in cutting. Therefore whilst technology is continually advancing in the automation process another problem has arisen in support:

Numerically Controlled machines require data to cut the required shape. This has led to software systems which perform different tasks that deal with some aspect of sheet metalwork. Some deal with design, others with creating a path for data between the computer and the machine so that the data that is passed is understood by the machine, others for accepting and converting data from other sources. The problem SHEET LIGHTNING particularly addresses is how to produce cutting data from a specific design which has previously been dealt with by geometrical drawing techniques.

One way in which similar problems have been solved in the past is by use of the "magic eye". The device has been used to follow the path of a line directly from a drawing and in doing so transfer the data to a cutting machine. Needless to say this method has its limitations but it has provided a means of converting drawings to data. The basis of SHEET LIGHTNING as a CAD package is to produce this data directly from a design on the computer using the mathematical model developed for the purpose rather than by drawing techniques. In this way the inaccuracies of the drawing techniques and the possibility of error through mistakes is removed.

5 USING THE PARAMETRIC SYSTEM

5.1 Concepts

The parametric system is the simplest and most convenient way of using Sheet Lightning. It consists of loading a design into the parametric environment that has been precreated in the design environment (2D/3D CAD system) and analysed to turn it into a parametric design by attaching a number of variable parameters to it. The process of taking a design from the design environment and creating a parametric design from it is covered later in this section. For the moment we are more concerned with learning to use parametric designs that already exists.

A parametric design may have a number of groups of variable parameters. There is usually one group for each object in the design and one group defining the relation between any two related elements in the design. The groups defining individual objects may define lengths, diameters, oblique offsets or other such dimensions that only have bearing on the object itself. A relative group may contain angles of centre lines to each other, the point at which centre lines cross from a given data on a centre line, or other such parameters.

A parametric design is loaded and manipulated in the parametric environment. The required values are typed into the relevant variable parameter fields and the patterns of the design are dumped out, generally in DXF format for passing to a third party software. If required, once the parameters have been set the design can be passed back into the design environment, laid out on a sheet and despatched to an output device such as a plotter. The design environment and parametric environment therefore allow designs to be fully interchanged between them.

The general purpose of the parametric environment is to allow a single design to be created that can be used to generate a whole family of similar designs, simply by typing in numerical values. It is therefore more limited in scope than the 2D/3D design environment which provides almost total flexibility, but with the ability to interact with the design environment in a way that makes it possible to expand the libraries of design to any extent.

5.2 Starting The Parametric System

You can either access the parametric environment when the program starts up or you can transfer to it from the design environment. At start-up Sheet Lightning cone up with a menu and a choice of three option: (1) Design System, (2) Parametric System or (3) Exit to DOS. Choose option 2 to access the parametric system directly.

To access the parametric system from the design system go to the “File” menu and select “PARAMETRIC”. Now choose “Parametric Mode” in this menu. You should now access the parametric system, the design system exists in the background and if a design is loaded into the design system it remains there until you come back to it or replace it with the design in the parametric system. Use ALT-B followed by ‘P’ to achieve quick access to the parametric

system.

When you access the parametric system by either of these methods the parametric system appears empty. When a design is loaded it appears in the graphics window on the right, the parameter text window appearing on the left.

5.3 Loading A Parametric Design

To load a parametric design into the parametric system you must use the “LOAD” radio button at the base of the parametric screen, below the graphics and parameter windows. This will initiate the file search dialog box, displaying the selected design in the graphics window, exactly as in the design system. Note that the system searches for PAR files, which are textual part of the parametric design files. The actual designs are held in GRP files having the same extension as the PAR files. The GRP files have exactly the same format as the GRD files used by the design system. The file search dialog box also provides a browsing facility to help locate the required design (See File Searching).

When the required file has been selected it will load directly into the parametric system, the design held in a GRP file appearing on the right in the graphics window, and the variable parameters associated with it from the PAR file on the left. You can now begin working with the design by “EXPANDing” parameter groups and typing in values.

5.4 Examining And Setting Parametric Values

The design that has been loaded becomes visible in the graphics window. The value group become visible in the parameter text window. Each of these groups begin with the field title “ELEMENT:” followed by the number of the element to which the parameters in the group belong (e.g. ELEMENT:1 mean first element), or the field title “RELATIVE:” followed by two numbers separated by a dash (e.g. RELATIVE:1-2). This means the group of variables define the relationship between the two elements in the design.

If you select the group field title with the mouse the group is expanded beneath it showing the variables that belong to it. The element/s to which this group belong are highlighted in the graphics window. Alternatively if the “EXPAND” radio button is selected all groups are expanded. If there are too many to fit in the text window then select the “PgUp” and “PgDn” text fields around the outer border to page through the parameter list. The “COMPRESS” command does the opposite of EXPAND, reducing the groups back down so that only the group field titles can be seen.

If you have expanded a group select one of the parameter fields. The text is highlighted and the edit field cursor appears flashing on the parameter value text, allowing you to type in a new value. When you have typed in a value press the ENTER key an watch the element in the graphics window change to the size you typed. Note that the design may appear unconnected, or to use the proper term, uncut. This is because updating the cut design every

time you change a value would take too much time unless the design is very simple or you had a great deal of processing power. Instead the system waits for you to specifically select the “WELD” radio button (i.e. cut and weld). Within a few seconds you should see the design updated in the graphics window.

When a parameter group is opened dimension arrows appear on the image in the graphics window. All parameter variables visible or opened in the parameter text window have a corresponding dimension arrow in the graphics window. When the parameter variable is selected in the text window the arrow is highlighted in the graphics window.

One useful facility to be aware of is the “RELOAD” radio button. If you type in values that change the appearance of the design to the point that it is no longer clear what you are trying to do it may be helpful to reload the design from disk. This simply reloads the design file and reset the parameter variable value according to it.

This is virtually all there is to specifying a design in the parametric system. The unfolded patterns can be examined using the “UNFOLD” radio button and despatched to DXF files using the “DXF-PATTNS” button.

5.5 Viewing Designs In The Parametric System

When the parametric design first appears in the graphics window it is automatically centred in the view. If you wish to examine it in more detail then first of all you may wish to produce the cut and welded elements using the “WELD” radio button (if not already). You can then use the view switching, zooming and panning facilities to examine the design more thoroughly.

The “XY”, “ZX” and “ZY” buttons allow the design to be seen in the three planes of the design “room”. The “3D” button turns it into an isometric view (i.e. 3D view). Use the “ZOOMALL” button to properly fit and centre the design in the graphics window at any time. If you wish to zoom in on the image then simply point to the part of the image you wish to view more closely and select with the left mouse button. The design will be zoomed by the factor set in the design system. If radio button “IN” is highlighted the zooming will enlarge the image, if “OUT” the zooming will reduce it. If the “PAN” button is highlighted the design will neither be zoomed nor out but the selected part of the image will be moved to the centre, in effect panning the image.

A special toggle feature is provided by the “PICK” radio button. If it is set (i.e. highlighted) then a square cursor appears rather than the normal arrow cursor. You can use this cursor to expand and compress groups or use radio buttons in exactly the same way as the arrow cursor except that when the square cursor is used in the graphics window then rather than zooming or panning the image the selection automatically highlights the nearest dimension arrow and selects it in the text window. The “PICK” button therefore provides a direct visual method of selecting the parameters you wish to alter.

5.6 Switching Between The Design And Parametric Systems

When you are working is the parametric environment the design environment continues to exist behind it. In effect the parametric system sits on top of the design system. You can move out of the parametric system into the design system using the “DESIGNER” radio button, but in doing so you will lose the design you are working on in the parametric system. Of course the program will ask you to verify that this is what you wish to do before doing it. If you wish to carry the design in the parametric system with you back into the design system, replacing the design that is already there you can do this using the “TRANSFER” radio button and the system will ask you if you wish to save the design in the design system before making the transfer.

It is also possible to transfer a design from the design system to the parametric system using the “File|PARAMETER|Transfer Design” command, but this is normally only done to create a new parametric design (see later).

5.7 Obtaining Output From Parametric Designs

The parametric system is set up so that you can very quickly obtain DXF output of unfolded patterns from a design. All you have to do is select the “DXF-PATTNS” radio button and type or select a file name. This command is equivalent to the “Output|Files Output|DXF Patterns” command in the design system. It dumps out the unfolded patterns of the design in DXF format with the patterns arranged in a column on above the other. DXF data can be imported to most general CAD systems (and many other programs) perhaps for cutting the patterns on a cutting device or perhaps for passing on to a plotter or printer. In such cases the work of arranging images would be done by the third party software. To obtain any other form of output including a sheet layout of patterns you must transfer the design through to the design system as described in the last section and use the “Arrange” and/or “Output” menu facilities there.

5.8 Creating New Parametric Designs

Creating new parametric designs is a fairly simple process provided you know what you want and can first produce it in the design environment. Once you have the “template” parametric design in the design environment simply select the “File|PARAMETRIC|Analyse Design” command from the pop-up menu system (or use ALT-B followed by “A”). This will transfer the design into the parametric system and “Analyse” it in the process. The analysis basically consists of attaching every feasible dimension to the design that has a none zero value. It is then your job to filter out the dimensions that you don’t want to be included in your final design. If the design is complex with many objects or many relationships between objects the number of dimensions produced by the analysis may escalate. Normally the design system would be the preferred tools for this level of design.

To filter out unwanted dimensions after analysing a design use the “MARK-MODE” radio

button. This button put the parametric system into a new mode. All selections of parameter variables will cause them to be highlighted in a new colour. You can highlight as many as you wish, all at the same time. You should highlight either all the dimensions you want to keep, or dimensions which you want to lose. It you wish to keep the highlighted dimensions select the “RETAIN” radio button and all the unhighlighted dimensions will be deleted. If you wish to delete the highlighted dimensions then select the “DELETE” radio button. Having reduced the list to the dimensions you require use the “SAVE” or “SAVE AS” radio button to create the parametric file (comprising a PAR file holding the text variables and a GRP file holding the design). If you accidentally delete the wrong variables you can reinitiate the analysis using the “ANALYSE” radio buttons.

5.9 Matching in The Parametric System

Matching has not yet been introduced as a concept in Sheet Lightning, but it is a powerful facility for adjusting the size of one object, or one end of an object to match, or fit another. Without matching objects some attempted intersections would be impossible. For example consider the simplest case of two intersecting pipes. A pipe of one diameter could not be properly intersected with a pipe of a different diameter to form a bend, because there would be gaps, the intersection would be incomplete. If these pipes were matched then one of the pipes would have been adjusted to fit the other. The adjustment of the pipe would mean either the diameter of the adjusted pipe is changed so that it becomes equal with that of the other pipe, or the diameter of one end of the adjusted pipe is changed to produce a match between them in which case the adjusted pipe becomes a cone. As the study of geometrical drawing techniques is quick to point out: the latter case does not mean simply adjusting the end of the pipe to the same diameter of that of the other pipe, rather a method called “common central spheres” would be used to make the match (for right cones only). Sheet Lighting handles all this for you, making matches between any kind of objects (not just right cones), with or without oblique offsets.

The design system allows objects to be matched in a number of different ways according to the setting of options. The options affect the form of adjustment that is used. The parametric system, on the other hand hides all this. The matching process still occurs as object dimensions are changed by you but only as a repeat of the original matching process, set up when the template file was created in the design system. This means that certain dimensions of the template design are under the control of the matching process. When analysing a design Sheet Lightning takes account of these dimension and eliminates them from appearing as variable parameters.

To get a fuller understanding of the matching process and the options available in making a match, consult the use of matching in the design section. This describes the options and facilities available in the “Match” menu of the design system. The matching command used by the parametric system is identical to the “Rematch” command in this menu.

5.10 Quitting The Parametric System

You can use either the “QUIT” radio button or the ALT-X hotkey at any time you wish to quit the program. Please note that this quits not only the parametric system but the underlying design system also. The system will prompt you to save you design before quitting the program.

5.11 Parametric File Format

It has already been mentioned that a parametric design consists of two files: the PAR file which holds textual parameters and their values and the GRP file which has the same format a GRD file (design file). You can examine the parametric file using a text editor because it holds readable text. The two files must have the same names with the differing file name extensions. A PAR file on its own is not enough to define a parametric design.

5.12 Using The Parametric System From The Command Line

Sheet Lightning can be used as a command line tool. This means you can run Sheet Lightning and use it to produce pattern data without any environment or graphical interface appearing or being used by the program. This feature is not really of any use to general users of Sheet Lightning but it may be of use to programmers who wish to generate pattern data in their own programs using their own front end.

In effect the command line facility provides access to the parametric system of Sheet Lightning, forcing it to load a particular template design, adjust the parameters of the design to specified values and dump out the DXF graphical data of the patterns all in one command. Programmers wishing to use or apply these facilities should contact the program vendors for details.

6 BEGINNING WITH DESIGN

6.1 Screen Layout

At startup the screen appears divided into six main areas.

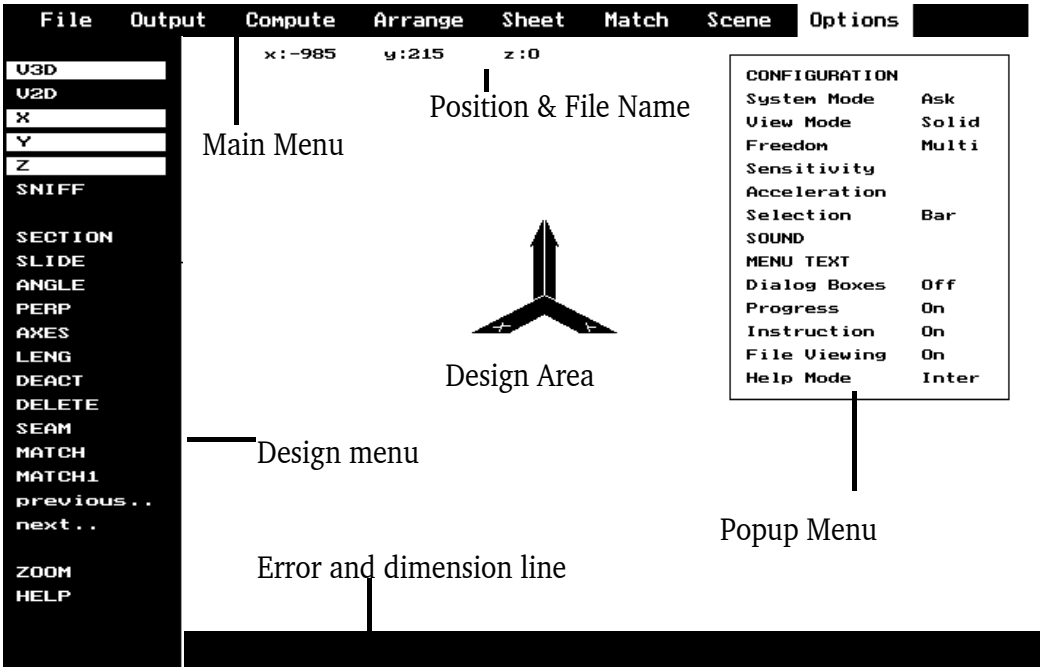


Figure 1.

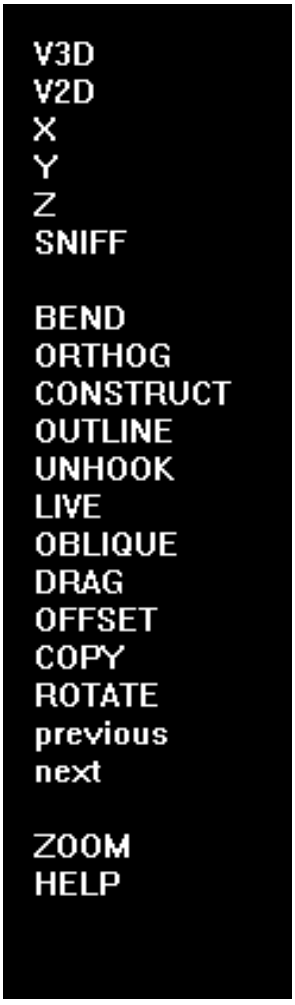
6.1.1 Main Menu

The top line of the screen contains the main menu. The main menu items are "File", "Output", "Compute", "Arrange", “Sheet”, “Match”, "Scene" and "Options". These headers are used to access other popup menus discussed later in the section.

6.1.2 Design Menu

The left hand side of the screen contains the design menu. This hold the commands and tools frequently used in creating a design and are therefore constantly available. The “previous” and “next” commands are for switching through three pages of tools as follows:

Design Menu Page 1



Design Menu Page 2

Design Menu page 3

6.1.3 Position & File Name

The second line on the screen below the main menu headers and to the right of the design menu is used for holding various items of numerical data and the name of the currently loaded file. At startup the default name "NONAME.GRD" appears which indicates no file has yet been loaded. Unless a specific name is given to a file when it is stored this (default) name will be used. The extension "GRD" is used to indicate the type of file. All design files have this extension and if not included will be automatically added when storing a file. The numerical data x,y and z are discussed later but very briefly they are simply the distances of the 3D cursor position from the walls and floor of a room. The second line is also used to report various errors which may occur or to issue a warning when either something "illegal" is attempted or if some other problem arises. Consequently this area of the screen is sometimes referred to as the "error line".

6.1.4 Dimension line

The bottom line is used to hold various dimensions when certain operations are being used. The dimensions given depend on the kind of operation in progress and the type of design object that is being worked on. These dimensions are discussed a later in section 8.

6.1.5 Design Area

The last area of the screen is the most important: the design area itself. This area occupies most of the screen and is where the design work is actually done. The design menu commands exist to support the work done here.

6.2 DESIGN ENVIRONMENT

The design facility offers the option of working in either 2D or 3D environment. The 2D facility is probably the best to begin with and is ideal for simple designs involving only two object, or more if centre lines are in the same plane. The 2D viewing can be in plan, side or end elevations. You can switch between these views quickly to examine or continue working on the design from another view. The 3D environment can be used to create anything from the simplest to the most complex design and is very useful for visualising designs.

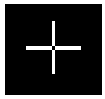
6.3 CURSORS

The cursor is the centre of attention on the screen and its position is controlled by the mouse (or by the keyboard arrow keys). The cursor has various forms. When in the design area of the screen it is normally in the form of a cross. The "hot spot" evidently being at the centre of the cross. When over the design menu the cursor changes to an arrow and the hot spot

is then at the point of the arrow. When using the zoom and scene shift facilities the arrow has the freedom of the whole screen. When using the main or pop-up menus the cursor disappears completely and the mouse controls the highlighting of the menu headers.

6.3.1 In The Design Area

At startup the design environment is in 3D mode and the cursor is in the design area of the screen at the origin. Now move the mouse and watch the corresponding cursor movement. As the cursor moves a box or square is drawn which relates the cursor position on the screen to its 3D position in the design. This box is only active in the 3D mode as it is unnecessary in the 2D where position is self evident.



6.3.2 Accessing The Design Menu

Move the cursor sharply to the left hand side of the screen into the design menu. As soon as the edge of the menu is crossed the cursor changes to an arrow. This indicates that a selection from the menu can now be made. The cursor is now said to be on "standby", which means that if it is moved back to the design area it will automatically change back to a cross and the box will reappear thus indicating you are again in design mode. Similarly by pressing the right hand button on the mouse (or escape button) the cursor will change directly to the arrow cursor which can then be used for selection of a design menu or main menu command. Having used the right hand button to directly convert to the arrow cursor it cannot be converted back into the cross cursor (i.e. placed in design mode) by simply moving back to the design area as before. In other words the cursor is not on "standby". In fact if the cursor is now moved over the design area it will remain as an arrow. In this state the arrow cursor can now be used for shifting and zooming purposes as described later in sec. 6.8.8. To return to design mode press the right-hand button again. If the arrow is over the design area the cursor will move directly into design mode. If the arrow is over the design menu it will be placed on "standby" (as described above) and can be returned to the design mode by moving back to the design area.

6.3.3 Changing The View

To change the view to a 2D view use the arrow cursor to select the “V2D” command in the design menu. The view should change to the last (or default) 2D view. Selecting the “V2D” again with a 2D view already showing will cause the 2D view to change into the next plane. Each 2D plane of view corresponds to a wall or the floor of the 3D room. Select “V3D” in the design menu to return to the 3D view.

6.4 MAIN & POPUP MENU

The main menu items appear on the top line of the screen. Each of these items hold a submenu or popup menu with a further selection of commands. To access them move the cursor to the top of the screen, point to the main menu item and press the left hand button.

Similarly to access the popup menu commands point to the command with the cursor and select with the left hand button. The submenu command selected will either hold a further submenu or execute a command. The submenus of menus are identified by the fact that they are written in capitals.

The arrow cursor mode will remain until all menus are shut down. This can be done by using the right hand button to shut menus one level at a time or by using the left hand button with the cursor pointing somewhere other than the currently open menu.

Note that a short cut exists to move directly into any of the menus at any time. These hotkeys consist of the ALT key and a letter of the pop-up menu name. For instance to quickly access the files menu press the keys ALT-F together. To gain familiarity with the menu arrangement try pressing various ALT-<character> keys. Most letters access a menu. Note that the ALT-X key is special in that it is a short-cut to exit the system. Of course a prompt in given to confirm this before the system in shut down.

The function of each pull-down and pop-up menu and the commands they offer are individually discussed in sec. 21. The mentioned hotkeys are also summarised in section 20.

The arrangement of the menu system is illustrated in fig. 3 to 5.

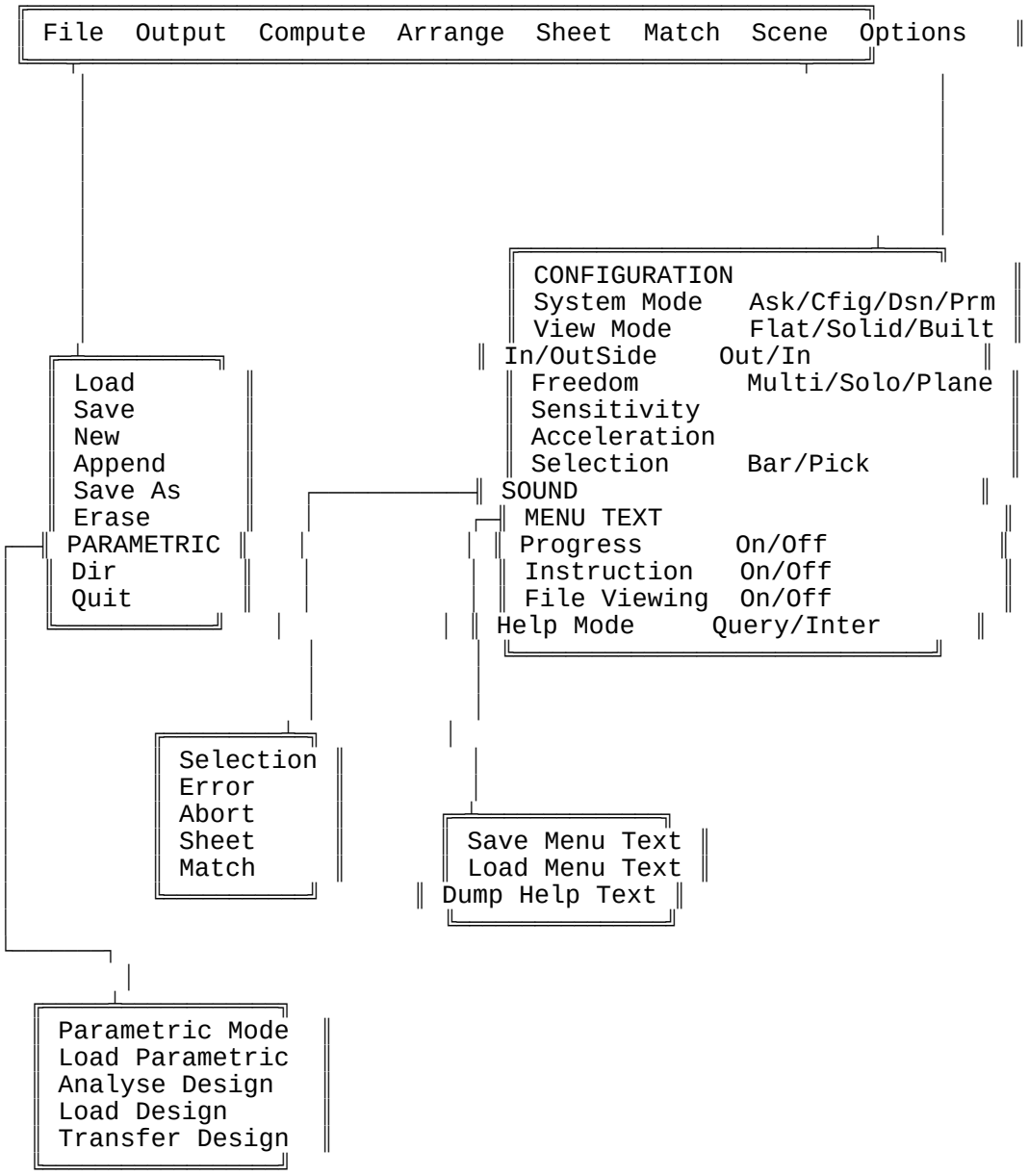


Figure 2.

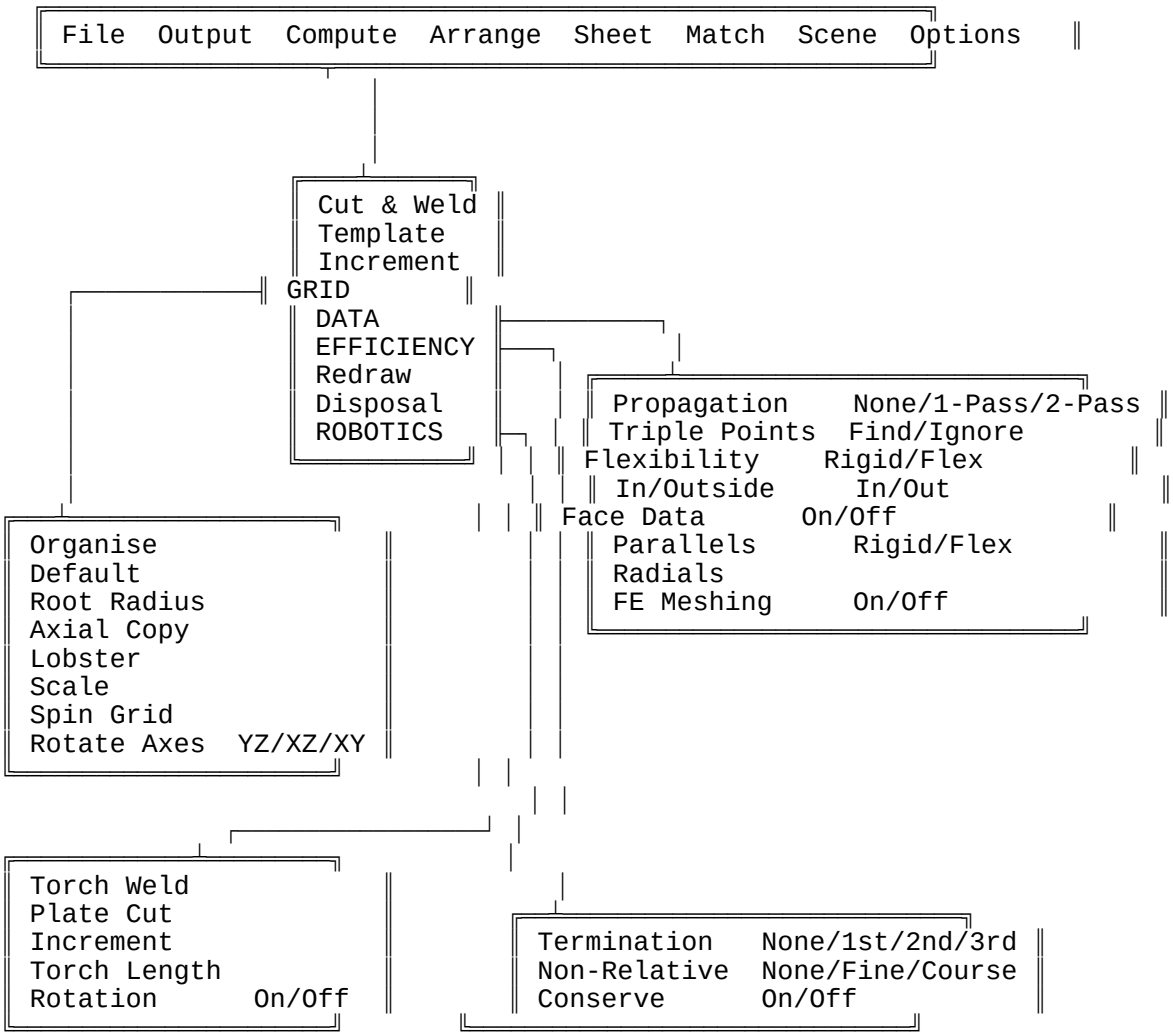


Figure 3.

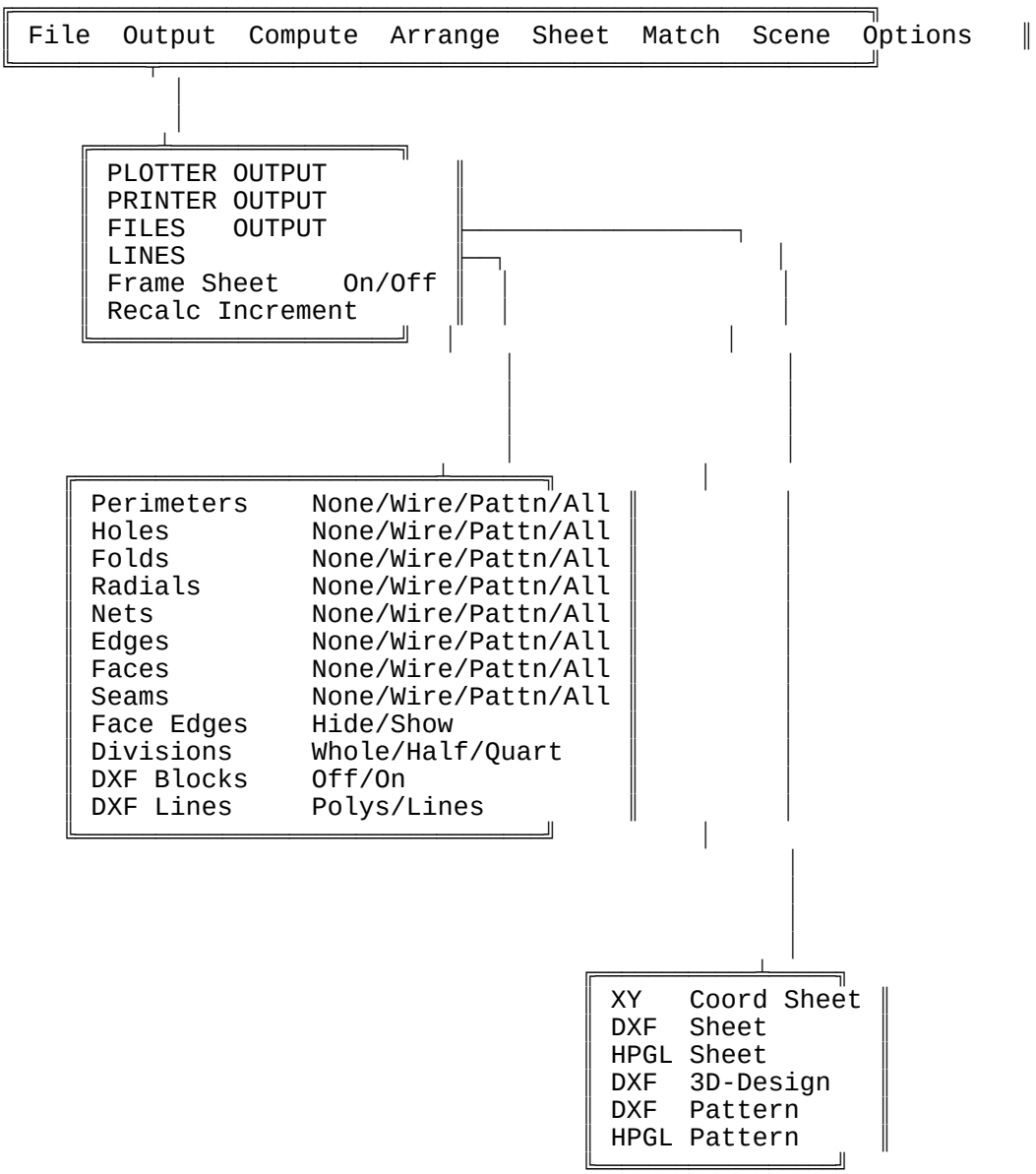


Figure 4a.

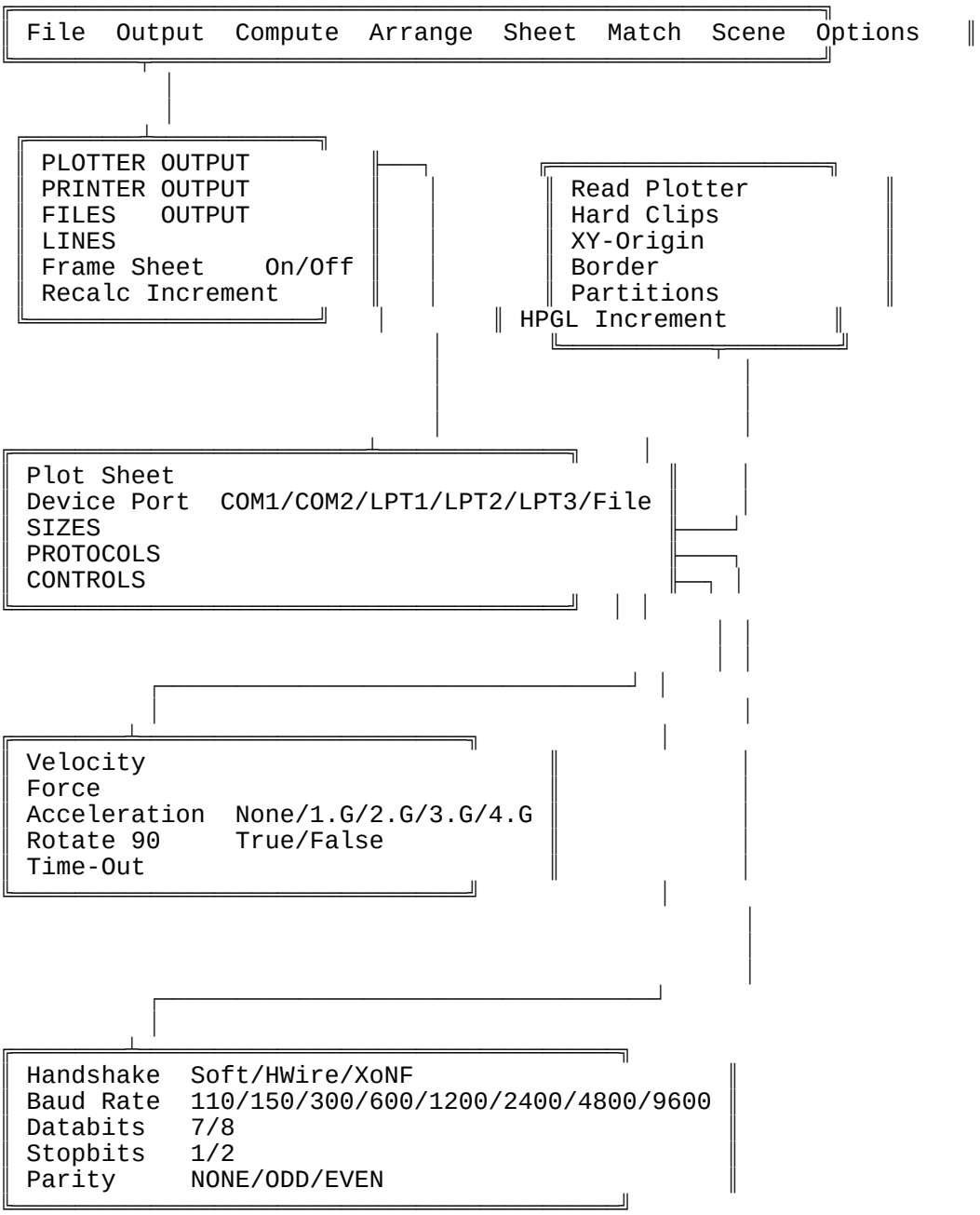


Figure 4b.

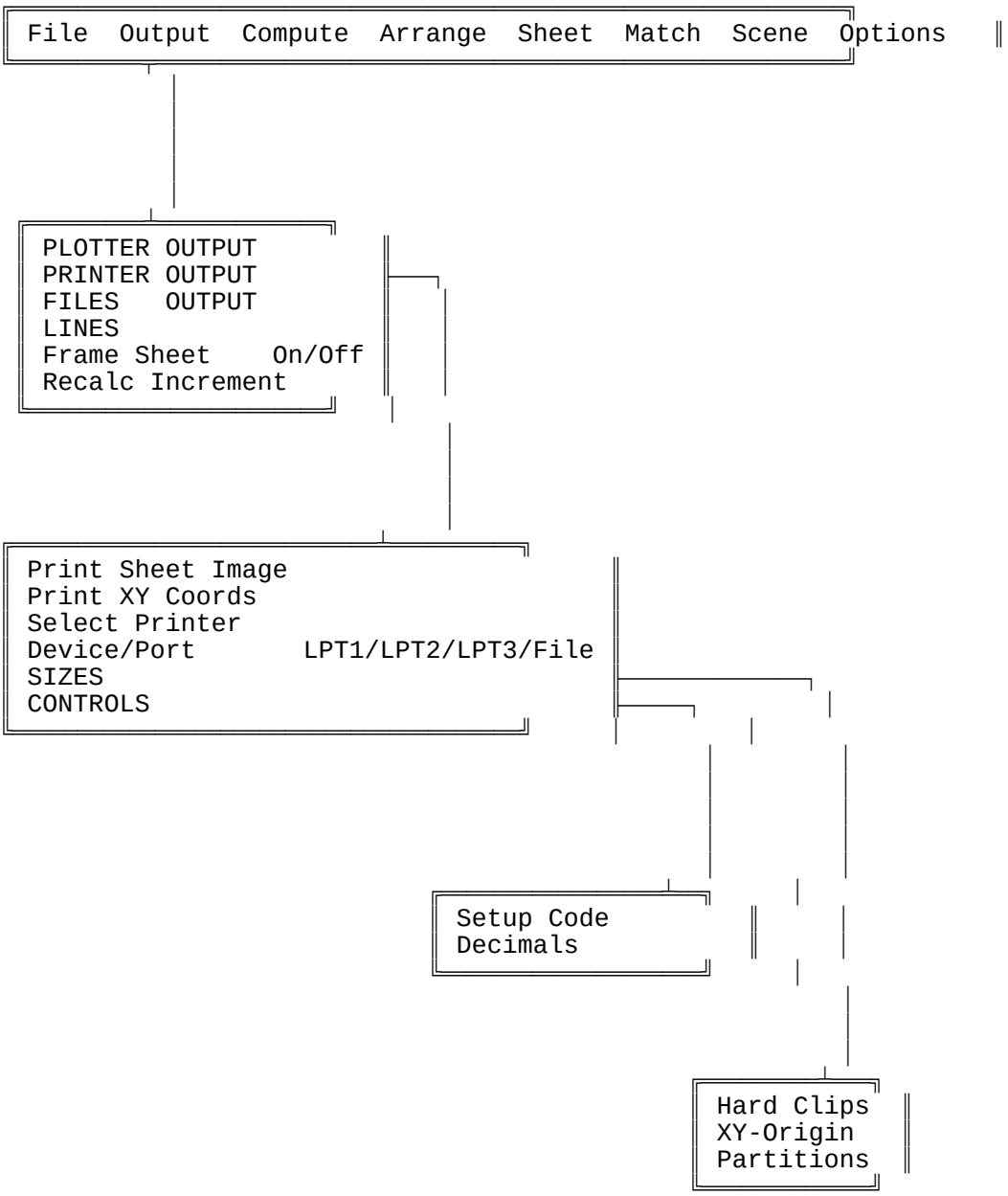


Figure 4c.

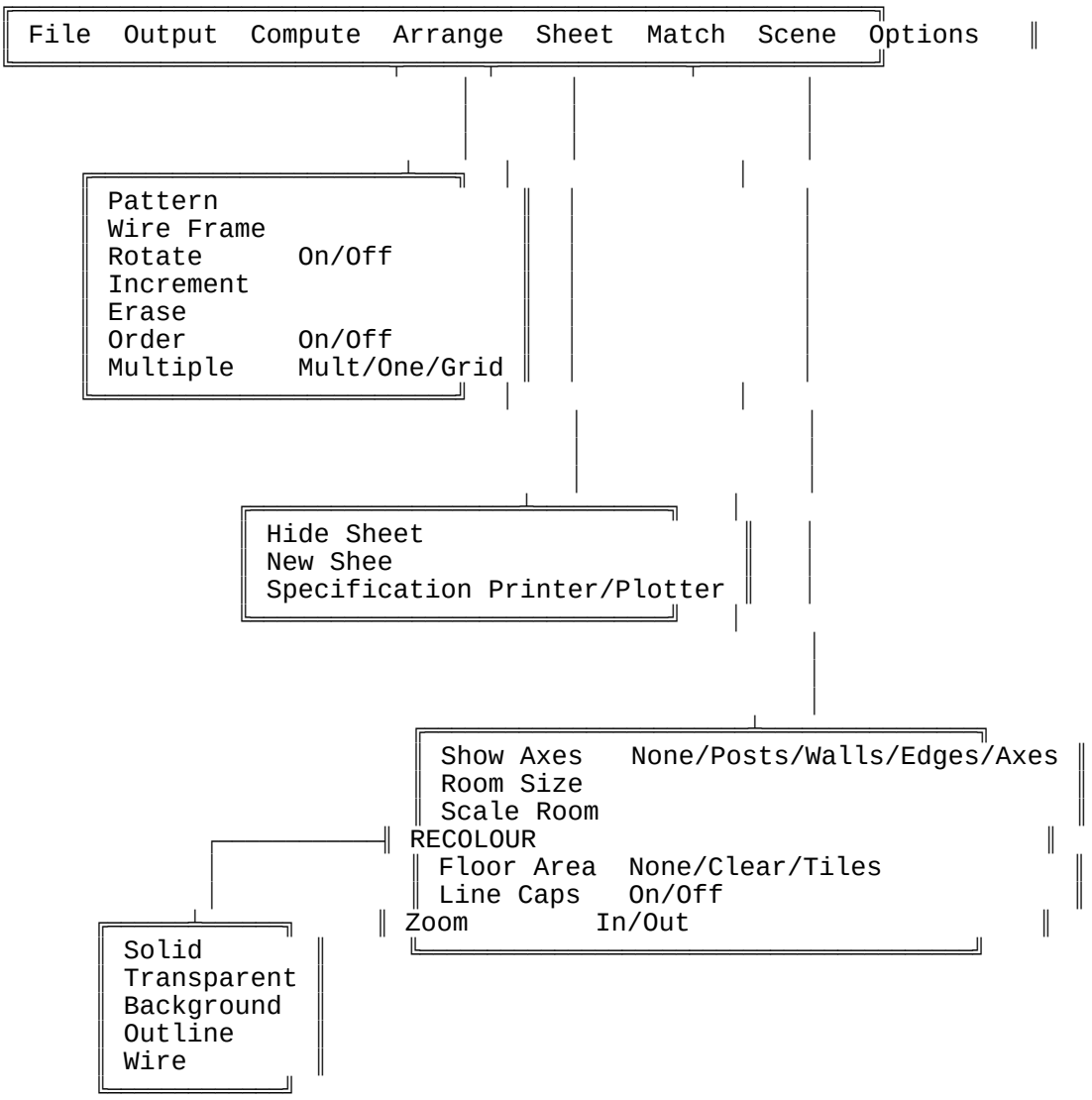


Figure 5.

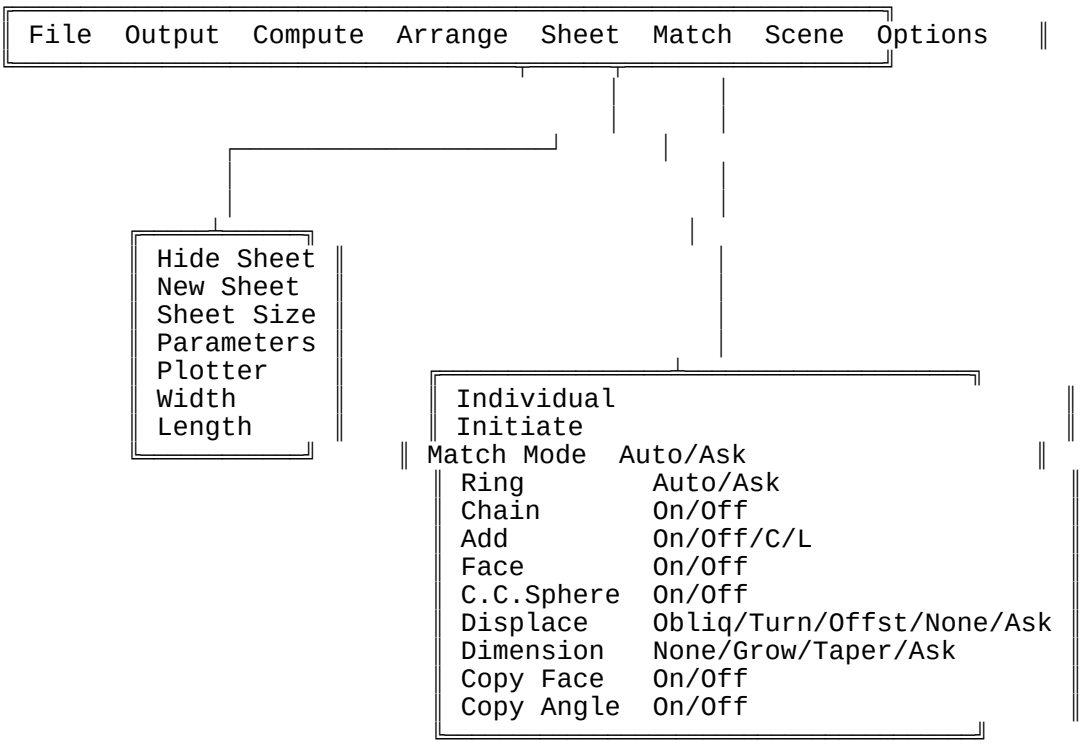


Figure 6.

6.5 FILE HANDLING

In order to get started in the design environment it is first necessary to know how to store and reload files (or designs). Each design is stored in a file with a name which is typed in at the prompt. As already mentioned the extension ".GRD" is added to any design file name to identify the type of file.

6.5.1 LOADING

A number of examples of designs are supplied on your SHEET LIGHTNING discs. To begin with we will load one of these files and resave it under another name. The following steps are required:

I. Select the "Files" command in the main menu. This will pull down the files menu (use ALT-F for a short cut).

ii. Select the "Load" command or press "L". At this point a dialog box will pop up headed "Load File". The file loading dialog box automatically searches for files with a GRD extension. You can now type in a file name into the file name edit field. If the GRD extension is omitted it is automatically assumed. Alternatively you can select a file name from the list of file names in the dialog box. The name will be automatically transferred into the file name edit field. To help you find the file you want to load the file search dialog boxes include a graphics window. Each time you select a file name the design is displayed in the window. If you still can't find the file you want you can use the file browsing facility accessed through the "BROWSE" radio button. The file browsing facility is discussed in more detail later in this section.

iii. Try selecting "EXAMPLE1.GRD". This is a simple example of a 90 degree junction between two pipes of similar sized cross-section. The pipes will appear on screen in their whole or "uncut" state. The procedure for cutting and welding these pipes is discussed later.

iv. By moving into the 2D views these pipes can now be viewed from different directions. Note that when a file is loaded certain other factors are also loaded from the same file which partly defines the environment in which the design was saved. This particularly includes the "Room Size" which may have been set previously (see sec. 6.8.7) and also data relating to the output of an arrangement of patterns to an output device (see sec. 13). To avoid this use the "APPEND" command instead of "LOAD" (see later). Append will load a file into the current environment.

6.5.2 SAVING

To resave the current design under the same name simply select the “File|Save" command. Alternatively as a short cut press the hotkey F2 which will do exactly the same thing. When the file is saved in this manner the existing file is overwritten and therefore includes any alterations that may have been made since the file was loaded.

6.5.3 NEW

Selecting "New" clears the current design and sets the current file name to "NONAME.GRD". This command is used to prepare to create a new design. The room size and environment remain unchanged.

6.5.4 APPEND

Append is used primarily to append a design file to whatever design in currently in the design environment, in effect combining the current design and the appended design into a single new design. The new objects loaded by the append command may be linked togther, as the objects in the current design may be but there cannot initially be any link between objects of the two designs. It is up to you to create the links you need in the new design.

When “Append” is used the room size remains unaltered and no sheet arrangement information is loaded from the appended design (as with the “Load” command).

6.5.5 SAVE AS

To save the design under a new name select the "File|Save As" command. As with the load command a box will pop up requesting a file name. Type in the new name and press the left-hand button. If the file already exists it will be overwritten. If not then a new file will be created under that name. Note that the file name on the second line of the screen has now changed to the new name. This indicates that the "working file" is the newly named file. Any subsequent save operations will save the current design to the file with this name. An alternative way of accessing the "Save As" facility is to point to the drawing name on the second text line of the screen and select with the left-hand button. The same input box will pop-up requesting a new name.

6.5.6 ERASE

A selected file can be erased using this command. Files to be erased are selected by

name or by a wildcard selection (i.e. '*' or '?') as with the "Load" command. You are always asked for confirmation by a verify box before the file is erased.

6.5.7 DIR

The directory command allows you to change the current directory. Unless a specific search path is given when saving or locating files the current working directory will always be assumed. The DIR command is also useful for discovering what the current drive and directory is. This is reported in an input box which is then available for alteration.

6.5.8 QUIT

The quit command exits SHEET LIGHTNING and returns to the operating system. The hotkey ALT-X provides a short cut to QUIT. If the configuration option in the options menu is set to "ON" then the current configuration will be automatically saved when "Quit" is used (though not when using ALT-X). This is discussed further in sec. 19.

6.6 Using The Design File Browser

The desgin file browser is accessed through any design file search dialog box using the “BROWSE” radio button. For example the “Load File” dialog box that pops up in response to the “File|Load” command has access to the browser for choosing and loading a design. The purpose of the browser is to allow you to quickly scan through the designs in a particular directory and pick the one you want.

When the “BROWSE” button is selected the system changes to the browse screen. This consists of a matrix of windows each containing a file name and design image and a menu of commands down the left hand side of the screen. Initially the screen displays a matrix of 36 design windows (i.e. 6x6). If more than 36 designs exist in the directory the surplus do not appear until you page through the remaining designs using the browse commands.

To select a design first select its window with the cursor. This will highlight the window. If you now select it again or press <Enter> this design will be accepted as the chosen design.

The commands available in the browser are listed below with a description of their purpose:

6.6.1 3D - Set the view of all visdible design to the 3D orientation.

6.6.2 XY - Set the view of all visible designs to the XY plane orientation.

6.6.3 XZ - Set the view of all visible designs to the XZ plane orientation.

6.6.4 YZ - Set the view of all visible designs to the YZ plane orientation.

6.6.5 PGUP

Pages the matrix of designs to the next set that were unable to fit in the current matrix. If there are no more designs in the directory this command has no effect.

6.6.6 PGDN

If the browser was previously paged up the previous designs can be retrieved by paging down.

6.6.7 “1x1"

Replaces the matrix of design windows by a single design window allowing it to be viewed at maximum size. The “PGUP” and “PGDN” commands now move through the designs one at a time.

6.6.8 “3x3"

The matrix of designs shows 9 design windows configured in a 3 wide by 3 high matrix.

6.6.9 “6x6"

The matrix of designs shows 36 design windows configured in a 6 wide by 6 high matrix. This is the initial default setting for the browse matrix.

6.6.10 WELD

The currently selected design is cut and welded showing the intersected form rather than just the uncut or outline form. This gives a clearer view of the design but may take a few seconds for the calculations to be completed.

6.6.11 UNFOLD

The patterns of the currently selected design are unfolded and displayed in its design window on at a time.

6.6.12 Directory Searching

Below the text “directory” in the commands of the browse system is the text “more..” and usually “..\”. Below this is a list of directories found in the directory currently being searched by the browser. Selecting one of these directories opens it up and commences browsing in that directory. Selecting “..\” moves back one directory towards the root. If you are already in the root directory it will not appear. A maximum of 8 subdirectories are displayed in the list. If there are more directories

than this is the directory you are currently browsing then use the “more..” command to page through them.

6.7 SPECIAL SELECTORS

At this point it will be an advantage to be aware of a number of special facilities which are frequently used for various purposes. These are known as the "Slide", the "Object Selector", the "Number Menu" and the "Sniffer".

6.7.1 The Object Selector

Some of the commands in Sheet Lightning require an object to be selected for the command to operate on. For example using the “Cut & Weld” command in the “Compute” menu requires you to identify which objects are to be cut and welded. Where such a selection is appropriate the object selector is automatically activated. It appears in the design menu and replaces the middle section of design tool text (the paging tools). Moving the mouse up and down or pressing the up/down arrow keys causes the highlight bar of the selector to move through the object numbers (or names if defined). Each number or name appears in a ladder of bars. The movement scrolls through these bars. If more objects exist in the design than there are bars the scrolling will still continue when the highlight bar reaches the lowest on the ladder.

If appropriate to the command in question it may be possible to select all objects in the design. In this case scroll the object selector upwards to the top bar of the ladder. This bar should read “ALL” meaning all objects in the design.

When the selection is made the object selector will disappear and be replaced by the design menu as before.

The “Options|Selection” toggle command provides an alternative to the object selector. Its toggle options are "Bar" or "Pick". "Bar" indicates use of the selection ladder menu as described above. "Pick" provides a square selection cursor which is used to pick the object directly from screen. When using "Pick" the selected object will flash once to indicate it has been picked.

6.7.2 The Slide

The slide tool restricts movement of the cursor in the design area to slide along an objects C/L. The slide is usually employed automatically when using some of the design tools but it can be used directly by selecting the “SLIDE” tool in the design menu. The following outlines the procedure:

I. Select “SLIDE” in the design menu. It may be necessary to page through to the right tool page using the “previous” and “next” paging commands.

ii. The object selector is automatically activated requiring you to select a single object in the design.

iii. Move the cursor back into the design area. The cursor will now be restricted to sliding along the object C/L. To switch sliding off return to the design menu and reselect the “SLIDE” again. The cursor will now have full freedom back in the design area. The position of the cursor relative to the ends of the object C/L on which it is sliding is often important. This can be monitored and fixed as required. The procedure for doing so is discussed further under Creating Designs in sec. 7.

6.7.3 The Number Menu

There are occasions when a number must be selected or used for some purpose, for example when setting mouse sensitivity, calculation increments or other angles. As with the object selector it is activated automatically when required and it appears in the middle section of the design menu. It appears as a ladder of numbers operating in a very similar way to the object selector. Up and down mouse movement and the UP/DOWN arrow keys cause the menu to scroll through a range of numbers. If the number you want is not included in the ladder, perhaps because the increment used steps over the value you want, or because the range of numbers is not wide enough to include the value you want then you can easily type in the value that you want. To do this simply begin typing numeric keys when the number menu is active, or press key F5 to force a value input box to pop up. Type in the value you require return it. The number menu therefore provides quick number selection but will accept typed values at any time.

6.7.4 The Sniffer

The "Sniffer" is often activated automatically and for various purposes. As with the object selector it is a way of selecting objects in the design for some operation. The difference is that when the sniffer is set the objects are selected from the design area and not from the design menu as with the object selector. It works by "sniffing out" the closest element to the cursor when the left button is pressed. This method is often used in preference to the object selector when it is necessary to also select a position on the centre line of the object. The procedure to use the sniffer directly is as follows:

I. Select the “SNIFF” command from the design menu. The text will be

highlighted.

ii. Move the cursor back into the design. It should now have the form of a cross (i.e it should be in design mode).

iii. Move the cursor close to the centre line of the element you wish to selected and press the left-hand mouse button. The cursor will immediately move to the closest object centre line to the cursor position.

Note that when the sniffer has snapped to the object the "SLIDE" command in the design menu is highlighted. This cursor movement is restricted to the object C/L just as if the slide command had been selected directly (see sec. 6.7.2). Also the sniffer has served its purpose and therefore the highlight is turned off. Having set the cursor "sliding" along an element C/L, depending on the purpose for which the sniffer was used it can now be used to select either one end or a central position of the element. In this case no selection is needed because the sniffer was used directly and has no other purpose. In effect it has simply been used as an alternative to selecting the slide command and using the object selector (see sec. 6.7.1).

6.7.5 Object Selector/Sniffer Collaboration

If the sniffer command has been set and for some reason it appears difficult to place the cursor to "sniff out" a particular element C/L, perhaps because another element obscures the view then an alternative is to use the object selector in "collaboration" with the sniffer. To do this just move to the object selector and select directly (providing the sniffer selector is set). When an item has been selected (as described in sec. 6.7.1) the cursor will be set to slide on the selected element when moved back into the design area.

6.7.6 Naming Elements

When using the object selector to identify a particular object initially the only identification is a number which gives the order in which it appears in the design. The object selector also offers another function; to name the objects in the design. If a character is typed when the object selector ladder is active an input box will immediately appears to accept the input as a name for the current object. On return this name appears with the number whenever the object selector is used. If the name is too long then the input is truncated but still accepted. The right hand or <E scape> button can be used to abort name input.

6.8 SETTING THE SCENE

The initial screen at startup shows a room with walls and tiled floor seen in a 3D orientation. There are a number of possible scenes arrangements possible, including a completely blank area, but this is chosen simply because it provides the designer with a good frame of reference without crowding out the screen too much. The background is purely a matter of preference and can be set up via the toggle commands in the "Scene" popup menu.

The options available are as follows:

6.8.1 Posts

The cursor movement can be monitored on several different backgrounds. The first is an arrangement of three posts laid along each axes to define the corners of the room.

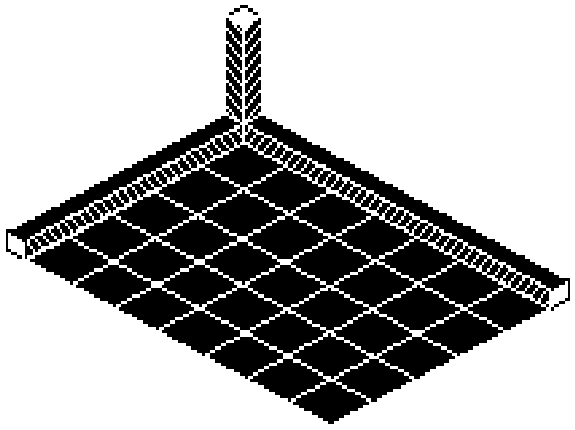


Figure 6

6.8.2 Walls

Alternatively the complete walls of the room can be drawn which provides a background in all 2-D views. This is the default setting.

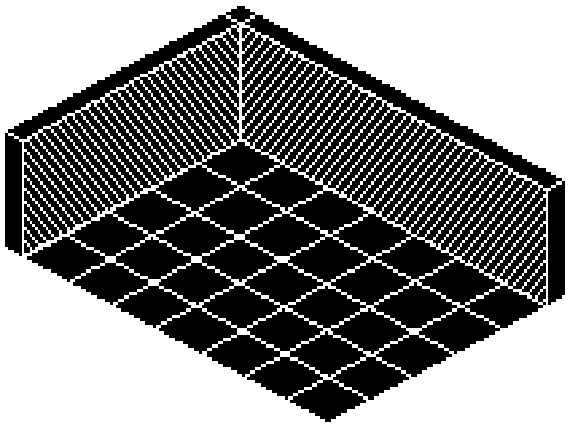


Figure 7

6.8.3 Edges

The third option offers a background which keeps distraction to a minimum by removing completely the Z-axis post or wall. The posts which represent the edges of the room remain thus giving some indication of the size of the room.

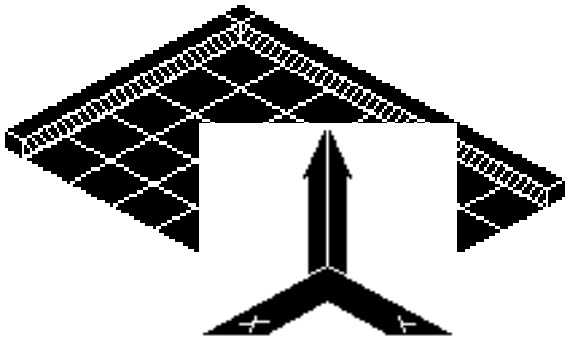


Figure 8

6.8.4 Axes

This gives minimal two or three line indication of the axes directions and labels them accordingly.

6.8.5 Floor

The floor of the room can be represented as either clear or tiled. The tiles form a useful reference grid to improve visualisation and assist with axes alignment.

6.8.6 In the dark

Another alternative is that the floor or walls/posts or both can be turned off completely leaving you to work "in the dark" with just the cursor box for reference. With practice this is useful in some cases for clarity.

6.8.7 Room Size

The length, breadth and height of the visible room can be changed using the "Scene|Room Size" command. These values are typed in response to the prompt. The size of room required should be determined by the size of design to be created. The room size settings are saved with the design in a GRD design file. It is therefore possible to load a room size of another design by loading it and then using “File|New” to clear it leaving only the room.

6.8.8 Zoom

It is possible to zoom in on any part of the screen which requires more detailed work or examination. Or if required the scene can be panned across to bring the current working area to the centre without changing the current scaling. The following steps give an example of using this facility.

i. Press the right hand button on the mouse to convert from the cross cursor to the arrow cursor. It is now free to move over the whole screen.

ii. Point to the part of the scene you wish to zoom into and press the left hand mouse button. This point will now become the centre of the design area and the scene will be scaled up by the inverse of the current zoom factor.

iii. Use the “ZOOM” command in the design menu to return the view to its normal state.

6.8.9 Zoom Factor

The “Zoom Factor” determines the degree to which a room is zoomed when the

arrow cursor is used to zoom on an image (or zomm away from it). The value of the zoom factor is set via the “Scene|Zoom Factor” command.

A zoom factor of 2 will double the current scene size when zoom is set to IN (if zoom is set to OUT the scene will be halved). If the factor is 3 then the resulting scene after a zoom will be one third larger and so on for any larger factor.

There is no restriction to the number of times a scene can be enlarged/contracted by zooming The only limitation is that with very extensive zooming the values used to draw the scene will become so large that they will lose accuracy. This is unlikely to present any restriction to the designer.

6.8.10 Panning

If it is required to pan across and shift the view to bring another point to the centre of the screen then:

i. Repeat the procedure to perform a zoom operation but this time set the zoom factor to 1.

ii. Position the arrow pointing to the position you now wish to become the centre of the screen and press the left-hand mouse button. The arrow position is moved to centre without zooming.

6.8.11 Default View

Select the “ZOOM” command in the design menu to return a room that has been zoomed to the default view.

6.9 PERFORMING AN INTERSECTION

Before looking at creating a design from the beginning, we will go through an exercise of performing one or two simple intersections to give an idea of the way SHEET LIGHTNING works.

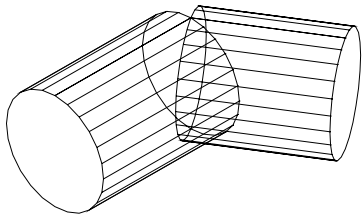
6.9.1 Intersecting

I. Load one of the example "GRD" files supplied on your SHEET LIGHTNING disc as described in sec. 6.5.1.

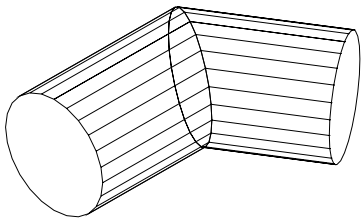
ii. Select the "Compute|Cut & Weld" command. The menu will disappear and

the "Object Selector" will be highlighted with the word “ALL". This is requesting whether an intersection should be performed for the whole design or for an individual object. If only one of the objects were to be cut, then by moving the mouse vertically the particular object can be selected. The object selector is described further in a later section.

iii. As the intersection is completed for each item, the whole object is erased leaving a shadow and is replaced by the cut object. On completion the whole design is redrawn thus erasing the shadows and leaving only the cut objects.



Outline Object



Cut Object

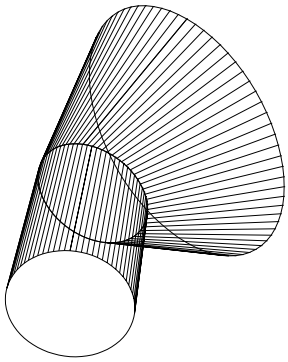
6.9.2 Examination

The cut objects can be examined more closely after a "Cut & Weld" operation using the zoom facilities. The procedure for producing a pattern is discussed in section 9.

6.9.3 Accuracy & The Net

The accuracy of an intersection is dependent upon the setting of the cut increment. If a small increment is set of perhaps 5 degrees or less, then the cut object will be

much finer that it would be for a coarse setting. The setting of this increment is evident from the "net" lines of the cut object plot. These are the longitudinal lines running along the surface of a pipe or cone. The closer together they are, the finer the plot. The means of setting these plot increments is covered more fully in a later section.



High Accuracy

7 CREATING DESIGNS

Having learned how to prepare the scene and deal with files, you are now ready create your first design..

7.1 Positioning

7.1.1 3D-Box

The position of the cursor in the 2D environment is self evident because the plane in which it moves is flat to the screen. In the 3D environment this is not the case. Therefore the cursor box is drawn to enable you to visualise the three dimensional position of the cursor in the room and the plane of movement. By moving the cursor around you will be able to see in which of the three axes the movement is fixed.

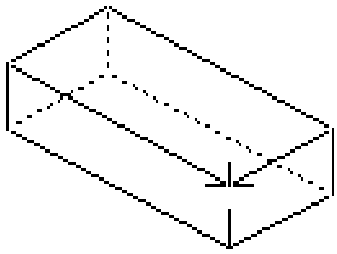


Figure 10

7.1.2 Cursor Freedom

The freedom of the cursor is always restricted to either 1 or at the most 2 of the axes at any one time. It cannot be free to move in all three axes at the same time except in certain circumstances (e.g. sliding) when it is restricted in other ways.

7.1.3 The Stabilizers

The axes stabilizers (see fig.2) are used to indicate the freedom and restriction of the cursor and, more importantly to change those restrictions to give cursor movement in the desired plane. In the 3D view the Z-axes corresponds to the vertical direction, X is the length of the room which appears to the left of the origin and Y is the breadth which appears to the right. By selection the axes stabilisers can be turned on (free/red) or off (restricted/blue). Alternatively the X,Y and Z letter

keys can be used directly from the design environment to change stabilizer settings.

7.1.4 Maintaining Position

If the cursor is used to switch stabilizers on and off, it would normally be by first escaping to the design menu in order not to disturb the current cursor position rather than by moving there directly. The previous position in the room can then be easily regained by a single press selection of the currently highlighted view command, “V2D” or “V3D”.

7.1.5 Stabilizer Freedom

The Options menu offers a facility labelled "Freedom". This refers to the way the stabilizer selections are interpreted. The possible values are “Multi”, “Axis” or “Plane”. Their mode of operation is as follows:

I. **Multi -** This is the default mode. It simply permits any individual stabilizer to be turned on or off irrespective of the setting of the others. In this way, any one or two axes can be freed for movement. Note that if all three stabilizers are free, then the movement of the cursor will default to the XY plane in the 3D view and give full freedom in the plane of any 2D view.

ii. **Axis -** When a stabilizer is selected with “Freedom” set to “Axis” then both of the other stabilizers will be turned off thus allowing movement in the room in one axis only.

iii. **Plane -** In this mode the selected stabilizer will be turned off and the other two stabilizers turned on. This allows movement in any plane to be set easily with a single selection.

7.1.6 XYZ Dimensioning

The X,Y and Z positions are indicated numerically by the dimensions on the second line of the screen. These dimensions also indicate the minimum increment that the cursor can move in the current environment. This is automatically set to correspond to the nearest acceptable round figure to the equivalent of one pixel movement. If movement is too coarse, then greater accuracy can be achieved by first zooming in on the operating zone. The exact X,Y or Z position can be determined by escaping from design mode and pressing the left hand button with the arrow cursor positioned on the relevant dimension on the dimension line. In response an input box will pop-up with the current axis value. The value can be changed which is usually useful if the axis has been stabilised and is therefore not subject to alteration

by cursor movement. The current position can also be altered by pressing key F5 when the cross cursor is active but free from restricted movement. An input box will popup holding the current postion and allowing you to type in a new position. If you type in a position and press <Enter> the cursor will be moved to the postion and a new centre line started so that further movement of the cursor will stretch out a centre line from this postion.

7.1.7 XYZ Accuracy

It is not always possible for all dimensions and positions in the design to be exact. For example, in a situation where an object's length is frozen, it is not possible for both the length of the object to be fixed at a particular value and the XYZ values to be rounded to the nearest incremental position because they are interdependant. The general rule for accepting the XYZ positions as exact, is that if the cursor is moving freely without any form of restriction then the XYZ positions given on the second line of the screen are exact.

7.1.8 Slide Dimensions

Having discussed the XYZ-dimensions this is a convenient place to mention the method of "slide dimensioning". This is a single dimension which appears on the second line of the screen and replaces the XYZ-dimension previously there. It becomes operative when the cursor is restricted to sliding along an object C/L. Its purpose is to indicate the distance of the cursor from one end of the object. This end is initially the first end that was placed in the design and can be identified by sliding the cursor along and watching whether the dimension increases or decreases. A more convenient way is to slide the cursor up to one end. This will force the dimension to be taken from that end for any further cursor movement. In this way it can be positioned at a specific distance from either end of the C/L. The F6 hotkey can be used to toggle datum ends.

7.1.9 Slide Positioning

If a particular position is required on the object C/L to which sliding is restricted, then the alternative is to type a value. As soon as numerical typing begins (i.e. pressing keys '+,-,0..9'), an input box will appear requesting the slide dimension required. This dimension is the distance from the current dimension datum, which will be one end of the object, to the present cursor position. If the typed value is valid, then the cursor will be moved to that set position along the C/L and will be rigidly fixed there until some other function frees it (such as pressing the left-hand button to begin a new object). The cursor can always be freed by escaping to the design menu and selecting the slide or bend command. The command can then be immediately turned back off if it is not required.

If the input slide value is in some way invalid as it would be, for example, if it was longer than the object C/L itself, then an error is reported and the input ignored.

7.2 Centre Line Theory

In the case of all objects of whatever type, the dimensions and position are always related to the centre line of the object. Therefore, having established how to get around the 3D environment using the stabilisers etc., we are now able to locate the cursor at any desired 3D position in the room in order to position a new centre line.

7.2.1 C/L Positioning

Positioning a centre line is simply achieved by moving the cursor to the point where the first object is to be positioned and pressing the left-hand button on the mouse. Further movement of the cursor now shows the technique known as "rubber banding" where a line is stretched between the first C/L end point and the cursor position. The C/L is now said to be "active". By finding the desired position of the second end of the object and pressing the button again, the C/L will be left fixed in position in the room. This process can be repeated to position multiple objects in the room. A "Cut & Weld" operation would then proceed to find the interference between these objects and cut them accordingly. This is adequate to establish simple "un-related" junctions between objects but other facilities are needed to establish "relationships" between the objects such as BENDs or TEEs which is a requirement of many sheet metal workers. These facilities are covered in the following sections.

7.2.2 Whole Object Manipulation

Note that when a C/L is positioned in the design as described in the previous section, as soon as the button is pressed to position the second end the object is drawn in its default state (providing the "Add Match" option is set to "ON"). This state is the simplest form; that of a cylinder, with a diameter set to the default (discussed further in sec. 8.1). Initially the object is drawn "whole" with no regard to any interference or intersection with other objects.

It is mentioned here because it is possible to manipulate an object in its "whole" drawn state exactly is the same way that the active C/L is manipulated. This is achieved very simply by selecting the "Outline" command when a C/L is active. in the following way:

I. Activate a new C/L as described previously.

ii. Move the cursor into the design menu. It is worth noting at this point that whenever an object is active and the cursor is moved into the design menu

the active object disappears together with the cursor box, if operative.

iii. Select the "outline" command in the design menu. The only response will be for the command highlight to be quickly switched on then back off, indicating that the calculation has been performed.

iv. The difference is seen when the cursor is moved back into the design area. The C/L is now replaced by its whole object. Movement of the cursor is exactly as before but now it carries the whole object with it.

v. To reclaim the object C/L, repeat the above procedure. This command can be used to toggle between the two states.

Manipulating an object in its whole state rather than as a C/L has both advantages and drawbacks. The main advantage is obviously the easier visualisation of the object's position and orientation. The orientation of a C/L, sloping towards or away from the screen, can be easily mis-read. The whole state makes this easier. The main drawback is that movement of the object is significantly more cumbersome due to the additional calculation that has to take place between each movement. With limited computer power this can be awkward. It may be worth noting here that manipulation in a 2D view is usually somewhat faster than the 3D.

7.2.3 Deactivating A Centre Line

A C/L can be de-activated (rubber banding turned off) at any time by selecting the “DEACT” command in the design menu. The command will flash on momentarily and then back off to show the operation has been performed. On returning to the design area the C/L will no longer be active.

7.2.4 Relationships

A relationship in SHEET LIGHTNING is the term used to describe special conditions which may exist in the way any two objects intersect each other in the design. If an object has no relationships, then it will always be cut off square at each end. If, on the other hand, one end of the object has a BEND relationship with another, then it will not be cut off square but the end of the object will be determined by its intersection with the other. In the case of a bend this will allow part of the material of the object to extend beyond the length of its C/L, rather than being cut off square to the end in order to complete the bend. In forming a relationship where the object TEEs onto another, the condition is similar; the object is cut off by its intersection. The difference is that the other object which the tee branch joins is not affected. It will still be cut off square. The only other type of relationship which can be formed is that of a JUNCTION. Both objects involved are still cut off square in this case but forming such a relationship does have other advantages which are discussed later.

7.2.5 Teeing

If an object is not related to any other either as a bend or a tee, then a Cut & Weld operation will cause the object to be cut off square at the end as seen when drawn whole. If a bend or tee relationship does exist then the object will be cut off by its intersection with the other object or objects. To establish a tee relationship take the following steps:

i. Position the first centre line in the room as described earlier.

ii. Next move to the design menu and select the "Slide/Tee" command (see fig. 2). This will cause the object selector to become active.

iii. Select the object to which you wish to form a tee (see sec. 6.7.1) and then move the cursor back to the design area. The cursor is now restricted to move only along the centre line of the selected object.

iv. Move to the desired teeing position on the object C/L.

v. By pressing the left hand mouse button, the first end point of the C/L of the new object is fixed on the C/L of the selected object and the slide restriction is released thus allowing the other end to be placed anywhere in the room. A tee relationship has now been formed as indicated at the bottom of the screen with the active object dimensions.

7.2.6 Bending

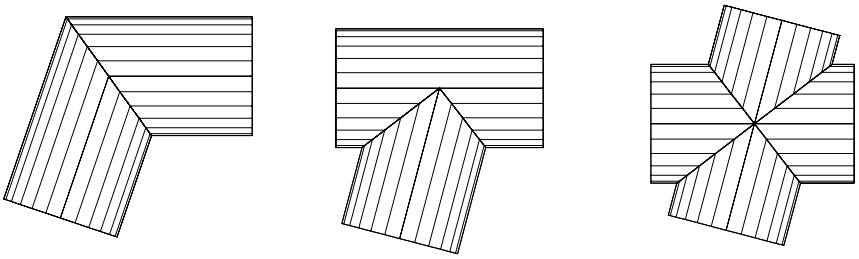
Forming a bend relationship is just as easy.

i. Follow steps i, ii and iii in the previous section.

ii. Now instead of selecting a position somewhere midway on the C/L of the object, push the sliding cursor up to one end. The slide will be restricted from moving beyond the ends of the selected object C/L.

iii. Now press the left-hand button as in (v) of the previous section. The fact that an end position was selected is sufficient to establish the bend relationship.

If a bend relationship has already been formed between two objects and a third bend object is added at the same point then each object is recorded to have a bend relationship with the other two. Any number of objects can be added this way to form a large multi-junction.



Bend/Tee/Junction

7.2.7 Chain Bending

Situations arise where a string or "chain" of objects need to be linked together end-to-end. One such case is that of the lobster back bend which may contain five or more pieces each linked end to end. This could be done using the method described in the previous section but an easier way would be to use the bend command in the design menu. In this way, one object is automatically strung onto the end of the last. The procedure is as follows:

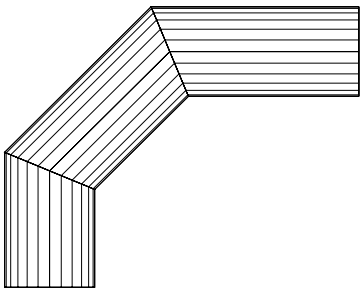
I. Select the Bend" command in the design menu. The command will be highlighted.

ii. Move to the design area and activate a C/L as described in sec. 7.2.1.

iii. Pressing the left-hand button will leave the C/L positioned in the room as before but this time the first end of a new C/L will be placed at the same position with just the single press. A bend relationship is formed between the two objects.

iv. Further cursor movement will stretch out the new C/L and the process will be repeated each time the button is pressed. The bend mode must be turned off manually. This will de-activate any currently active object.

Note that a special facility does exist for the automatic creation of lobster back or segmented bends. It is accessed through the “Lobster” command in the "GRID" menu or in the design menu.



Chain Bend

7.2.8 Junctions

Forming unrelated junctions has already been mentioned where two centre lines are simply placed in the room crossing each other. This does not form a fixed relationship between the objects. Forming such a relationship is useful for other purposes. One advantage is that you can be sure that the centre lines do cross in the 3D environment. It is also useful for "matching" and "dragging" (see later). To form a junction the dragging facility must be used.

This can be done in the following way:

I. Using the drag facility pick an object up by some mid position on its C/L.

ii. Set the slide to another object using either the slide command selection or sniffer section.

iii. Dragging the object to a mid point of the C/L to which sliding has been established and press the left hand button.

iv. A junction relationship is now established.

The relationship between the objects can be queried using the dimension command (see next section).

7.2.9 Dimensions

You will notice that in addition to the XYZ positions in the room, as displayed on the second line of the screen, when a C/L or object is active various dimensions also appear at the bottom of the screen. These include the length of the object and its default diameter. The default diameter can be altered so that any new C/L is automatically set by default. When in position each object can be dimensioned

individually as required. At this point probably the only useful dimension is the length of the object. Note that all dimensions are updated whenever the computer senses a brief pause in cursor movement. Sec. 8 is devoted to the subject of dimensioning where it is covered in more detail.

7.2.10 Length Freezing

Having activated a new C/L on screen, it may prove difficult to position the cursor aligned in a certain direction and at the same time set the object to a particular length. To overcome this problem, a facility is available in the design menu to fix the object length to a certain value (see fig.2). It can be used in the following way:

I. Having activated the new C/L move the cursor into design menu and select the "LENGTH" command. Note that if a C/L is not active then this selection will not be accepted and no highlight will appear.

ii. Now move the cursor back to the design area, into a position where the length of the object is set to the required length as indicated by the dimensions at the bottom of the screen.

iii. Press the left-hand button to freeze the object at this length. The cursor is still free to move and the object is still active as before except that its C/L length is fixed and cannot be stretched out.

iv. All that remains, is to align the C/L in the required direction and press the left hand button. The C/L is now fixed in position and the length freeze highlight is turned off.

If, having frozen the length of the object, you want to release or re-set it, then re-select the length freeze command and repeat to re-set.

7.2.11 Typed Length

When an object length is frozen and the object is still active, its length can still be altered using the facility of typed input. As soon as numerical typing begins (i.e. keys '+','-' or '0'..'9' are pressed) an input box will appear requesting the object length. On return the length of the object will be immediately adjusted but with exactly the same orientation.

7.2.12 Axes Alignment/Snap

Alignment of the C/L in a certain direction can be difficult to set exactly by hand. The design menu offers certain aids to overcome this problem, one of which is the

axes snap facility. As the name suggests this allows you to align the C/L with any of the three axes. To use it the C/L must first be active, then do the following:

I. Go to the design menu and select the "AXES" command. This will be highlighted if a C/L is active.

ii. Move the cursor back to the design area. Position the active C/L as close as possible to alignment with the desired axis and press the left hand button. The C/L will be automatically aligned with the nearest axes direction. The cursor is now restricted to move only along the axis with which the C/L is aligned. If the C/L length is also frozen at this point, then the cursor will not be free to move at all.

iii. Press the left-hand button again, the C/L position will be set in the design and the axes alignment command will be turned off.

7.2.13 Angle Setter/Snap

The angle setter command is used in a similar way. Its purpose is to allow the relative angle between two related C/Ls to be set. Again this can only be done if an object is active, otherwise the selection will not be accepted.

I. The active object must be related as a junction, tee or bend to another object. If this is the case then select the "ANGLE" command in the design menu. If an object is not active or is not related to another C/L, the selection is not accepted.

ii. If the active C/L is related to more than one other object then the object selector will become active, requesting which of the related objects the angle is to be set to. Only one object can be chosen.

iii. The number menu will now become active, this time requesting the actual angle to which the active C/L is to be set relative to the selected object. The angles range from 5 to 90 degrees and are available in increments of 5 degrees. Select the required angle. If a perpendicular is required, then select 90 degrees. Note that typed input can be given when using the number menu which will allow any angle to be chosen.

iv. Move back into the design area and position the active C/L leaning in the direction towards the end of the selected object that the angle is to be set. If the C/L is to be set perpendicular then, of course, the lean of the C/L does not matter.

v. When the active C/L is leaning in the right direction, press the left hand button and the angle will be set. As with the axes alignment, the cursor

movement will be restricted to this setting.

When the position of the cursor is adjusting as described in step (v) then it is important to understand that although the new centre line is moved, it is kept in the same relative plane. For example if the two C/Ls were in the same plane both flat or parallel to the floor of the room, then when the angle of the active C/L is adjusted they will still both remain in that plane.

7.2.14 Orthogonal Mode

The orthogonal command is identified as "ORTHOG" in the design menu. It provides a facility to ensure that all new members placed in the design are automatically aligned with the orthogonal axes. With orthogonal mode set the active centre line is not tied to the cursor directly but aligns itself with the nearest axis to the alignment of the centre line. The command is a toggle facility and can be disabled by reselecting it.

7.2.15 Construction Mode

The construction command toggle, identified as "CONSTRUCT" in the design menu provides a toggle facility which forces all newly activated C/L's to be placed in the design in their C/L form. It is often necessary in producing a design to use construction lines. This toggle facility provides a quick and easy means of changing modes.

7.2.16 Offset

Occasions arise where an offset, tee or junction is required. An offset is a situation where the centre lines of two objects do come close enough together for there to be an intersection but not close enough for the C/Ls to actually meet or cross. One way of forming a junction of this kind is simply to place two C/Ls in the room with a gap between them. As mentioned previously, this has drawbacks concerned with other design aspects like matching and dragging (see later). If a related offset tee or junction is to be formed, then the offset facility must be used. The procedure is as follows:

I. Form a normal related tee or junction without any offset as previously covered in sec. 7.2.5.

ii. Select the "OFFSET" command in the design menu. The sniffer will automatically activate.

iii. Using the sniff/snap method select the object to be offset in the design

menu. The cursor should now be tied to the objects C/L. Find a point on the centre line from which you want to drag the object and press the left mouse button. The object will now drag along a perpendicular line defining the line of offset of the two C/L s.

If you press numerical keys or key F5 at any time during the offset operation an input box will pop up allowing you to enter an offset value.

7.2.17 Relative Dimensions

It has already been mentioned that the dimensions of any active object can be seen in numerical form on the bottom line of the screen. When an object first becomes active, these dimensions only relate to the active object itself. There are times when the dimensions of a C/L relative to another C/L are required, for example when using the scissor operation. These dimensions are available for any C/L related to the active C/L and are obtained in the following way:

I. Having activated a C/L move the cursor into the design menu and select the "D" or dimension command.

ii. If the active C/L is related to more than one other then the object selector will become active allowing one of the related C/Ls or objects to be selected. On making the selection the dimensions at the bottom of the screen will immediately change to the relative dimensions.

The actual dimensions available are discussed in a later section.

7.2.18 In-line Objects

There is often need to make two C/Ls which are linked in a bend relationship to be placed in-line with each other as for example when one straight pipe meets another to form a pipeline. This is done in the following way:

I. First establish the bend but keep the new object C/L active.

ii. Select the Slide/Tee command in the design menu and, using the active object selector, choose the same object to which the fixed end of the active object is already related.

iii. Sliding will now occur on that same object exactly as before except that it is automatically detected that these objects are already related and therefore the cursor is allowed to slide beyond the C/L ends but still in-line with it. In this way, an end-on in-line bend can be established. Note that a second

relationship between any two objects cannot be established.

7.2.19 Bend Dimension Copying

When a new object is set in the design related to another object as a bend, providing that matching is enabled in the Match menu, the type and dimensions of the end of the object it is joined onto will be copied to both ends of the new object. This feature is useful when forming a chain of objects (eg. a pipe line). Note that the feature is auxiliary to the matching operation which will be applied immediately after the new object is set providing that "Add Matching" is enabled in the "Match" menu.

7.2.20 Oblique Offsets

Any object can be given an oblique offset irrespective of its type or cross section. Oblique movement can be initiated is the following way:

i. Activate a new C/L in the room.

ii. Go to the design menu and select the oblique tool, identified as “OBLIQUE” in the design menu (see fig.2). The text will be highlighted.

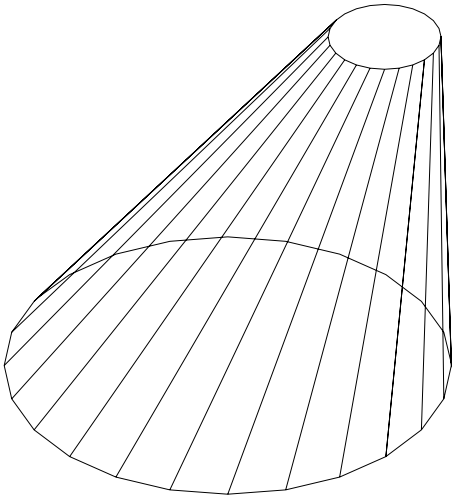
iii. Move back into the room and position the active C/L in the desired orientation of the oblique perpendicular. Press the left hand button to accept this position.

iv. Rather than placing the second end point of the active C/L as would normally have happened the C/L remains active but now as the cursor is moved a right angled triangle is formed. The hypotenuse stretches from the first point to the cursor which is the actual C/L of the active object and the other line attached to the first point is the perpendicular to the oblique. At this point the oblique command is switched off.

v. When the left hand button is pressed again the second end point of the new object is fixed and the new object outline will appear in the design. The result should now more easily be appreciated with the oblique offset obvious by the shape of the object outline.

This procedure can also be followed with an object that has been reactivated as described previously. In this case the effect of oblique offsetting is even more apparent by the distortion in shape of the object outline as the cursor is moved.

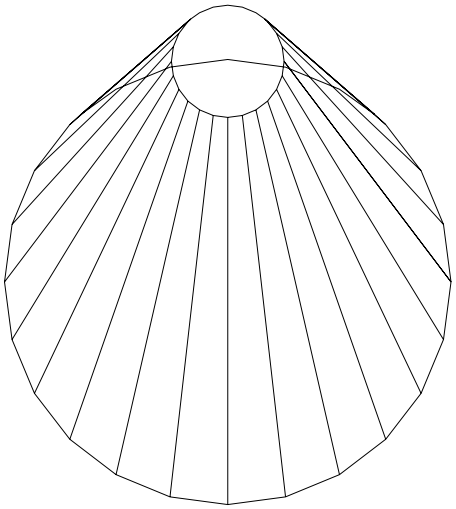
There is still another alternative and that is to reactivate an oblique object directly.



Oblique Cone (3D)

7.2.21 Oblique Reactivation

An object can be reactivated directly into oblique mode as described in the last section. The procedure is identical except that the oblique command in the design menu is selected. The sniffer is used to identify the object to be reactivated as before but when the reactivation begins it can be seen that it is in an oblique sense. Note that the perpendicular length is also frozen as before and therefore must be unfrozen to allow alteration.



Oblique Cone (2D)

It may be worth mentioning at this point that a major method of achieving a particular oblique offset is by using dummy lines and the slide facilities to obtain exact positions.

7.3 Editing Designs

7.3.1 Delete

An obvious need on occasions is to delete an object from the design. This is done by selecting the “DELETE” command in the design menu and then selecting the range of objects to be deleted. To do this, the object selector is activated twice to select the first and last object of the range to be deleted.

7.3.2 Reactivation

The majority of this section has been concerned with placing and manipulating a "new" active object in the design. A facility is also available for reactivating an object C/L which has already been placed in the design. The reactivation can be from either end of the object and once reactivated, all the operations previously discussed are then available to the newly reactivated object. The facility is accessed through the “LIVE” command in the design menu. There are certain restrictions to this it:

I. If the C/L to be reactivated is related to another object either in the form of a bend or tee, then the point at which the related C/Ls meet or cross cannot be moved. Therefore, if the object is related to another at one point only along its length, then re-activation will be allowed but the object will pivot about that point.

ii. If no relationships exist the pivot point will always be at the opposite end to the re-activated end and behave as if it were a new C/L.

iii. If the C/L or object to be re-activated is related to other objects at more than one point along its length then reactivation will not be permitted. The error report "Rigid Centre Line" will be given.

In order to reactivate an object C/L, take the following steps:

iv. Move into the design menu and select the re-activate command. If there are no objects or C/Ls in the design the selection will not be accepted. Otherwise the command will be highlighted and the "sniffer" (see sec. 6.7.4) will be turned on.

v. Return to the design area and move the cursor near to the desired C/L. Press the left-hand button. The "sniffer" will find the nearest object to the cursor and automatically set the slide to that object.

vi. Move to, or near to the end of the object you wish to re-activate and press the left-hand button again. If the operation is legal, the object is re-

activated.

Note: when the object is reactivated its length is in the frozen state. To alter this length, the "length freeze" command must be specifically selected after re-activation to unfreeze it.

7.3.3 Dragging

Having set a C/L in the design, it is possible to move the whole object or C/L to a new position using the drag command. The orientation of the C/L to the axes directions will not be changed. This is done in the following way:

I. Select the “DRAG” command from the design menu. This will turn the "sniffer" on.

ii. Using the "sniffer" select the item to be dragged (see sec. 6.7.4).

iii. Position the cursor at the point on the C/L where the object is to be dragged from and press the left hand button.

iv. The object will now be "dragged" with the cursor as the cursor is moved around the room. At this point no other object in the room will be moved.

v. When the new position for the dragged object is found, press the left-hand button and the object will be left set in position.

When the button is pressed as in (iv), all the objects that were originally related to the dragged object and the objects related to them will "catch up". That is, they will be repositioned so that they are in the same relative position to the dragged object that they were in before it was dragged. In this way a whole "chain" of related objects can be moved and repositioned in the design.

7.3.4 Drag Relating

When dragging either a single object or a chain of objects as described in the previous section, it is possible to "link in" or relate the dragged objects to another section of the design. This is done by setting up a drag as described and then selecting the slide command in the design menu. As when the cursor is sliding normally, it is restricted to move only along the object C/L, the difference in this case is that the object is dragged with it. If the left-hand button is pressed in this situation, then a relationship is formed between the dragged object and the object to which the slide is set. If the object was dragged from one end and the slide position is midway, then a tee will be formed with the slide object. If, on the other hand, the slide position was also at one end, then a bend relationship will be

formed. If the object was dragged from a midway position and the slide was also set midway, then a junction relationship is formed.

7.3.5 Drag Unhooking

Occasionally it is necessary to drag an object to another part of the design and disassociate it from any related objects so that they do not "catch-up" when the dragged object is put in a new position. To do this, use the unhook facility in the design menu in the following way:

I. Select an object in the design using the drag command and sniffer, as outlined previously in sec. 7.3.3. On this occasion, ensure that when the object is selected that a midway point on the C/L is used as the dragging position.

ii. With the object dragging move into the design menu. This should cause the dragged object to disappear.

iii. Select the un-hook command. The command should momentarily be high-lighted to show that the un-hooking process has taken place.

iv. Move back into the design area. The dragged object will reappear. Drag the object to a new position and set it down with a press of the left-button. None of the other objects to which this one was related should move (i.e. they should not "catch-up") thus showing that they are no longer related.

The reason why a midway point was selected to drag the object in this case was in order to ensure that all object relationships were severed by the un-hooking operation. If an end point had been selected, then only the tee and bend relationships associated with that end would have been severed. Therefore by selecting an end of the C/L it is possible to drag away a branch of the design rather than just the one object.

7.3.6 Direct Unhooking

The procedure for unhooking an object from its relations using the drag command has already been described. The unhook command can also be used without dragging by first setting the slide command to the desired object. If the slide is pushed up to the end of an object C/L and then the “UNHOOK” command selected the text will be highlighted momentarily and that end will be unhooked from all its relations (bends and tees). If the cursor is placed in a mid-way position and the command selected then all relations with that object will be broken.

A word of caution should be added here in using the unhooking facility in this way.

Once the relationships are broken the design will still look exactly as it did before. The difference is only seen when the new design without the relationships is cut and welded. In general it is better to break relationships during dragging so that the unhooked objects are physically separated by the drag operation.

7.3.7 Duplicating

The procedure for duplicating objects is exactly the same as that for dragging (see sec. 7.3.3) except that the “COPY” command is initially selected instead of the drag command. The difference is that when the object is selected with the sniffer and dragged away the original object is left behind and a new but identical object is dragged from it. This procedure can be repeated if multiples of the same object are required in the design.

7.4 Special Features

7.4.1 Lobster Back Bends (Swept/Segmented Bends)

The design menu and the "GRID" menu hold a special utility for the quick and easy creation of a swept or lobster back bend. To creation such a bend certain conditions must be met. These are discussed in the following procedure.

The bend is achieved by replacing a simple two piece bend with the lobster back bend comprising of a specified number of pieces. Therefore to create the lobster a two piece bend must exist in the design.

To create the bend use the following procedure:

I. Select “LOBSTER” in the design menu or "Lobster" in the “GRID” menu. The object selector will become active requesting one of the two piece to be replaced with the lobster back bend. For the procedure to continue the selected piece must be related to another in a bend relationship. Otherwise the operation is aborted and reported on the error line.

ii. The object selector will be activated for a second time, this time only allowing a limited number of objects to be selected (i.e. those that form a bend relationship with the first piece). These pieces are those which could prospectively be substituted by the lobster back bend. When the second piece is selected certain other validity checks are carried out.

iii. First of all neither piece must form any other relationship with any other piece in either a tee, junction or bend at any point except at the opposite end to that which the two selected pieces are related. The reason for this is obviously that the two selected pieces are to be replaced with only their opposite ends unaffected.

Secondly the two selected object C/L's must not be in line with each other. They must form a positive angle. If any of these conditions are not met the operation will be aborted and the error reported on the error line.

iv. A simple "Yes/No" prompt is given requesting whether the bend should have half angle ends. If "Yes" then the ends of the bend will sweep only half of the angle swept by central pieces.

v. Another "Yes/No" prompt is given requesting whether the bend should have straight ends. If "No" then the taper of the bends should be the same as all other pieces. Note that the taper is decided by the difference in size of the opposite ends of the two pieces. These two pieces will always become straight, therefore the taper from one size to the other is spread evenly over

the bend.

vi. At this point the number menu becomes active allowing the number of pieces required in the lobster back bend to be selected.

vii. An input box now pops up with the maximum possible radius of the bend. If this is returned one of the initially selected pieces will completely disappear as it is "eaten up" by the bend. Alternatively a smaller radius can be returned which means that both of the initial objects are simply shortened.

Note that in forming the being each piece is automatically matched to the joining pieces at either end.

7.4.2 Scaling

A design can be scaled by any given factor. The “SCALE” command responds with a prompt for the scale factor. It the cursor is currently sliding on some object in the design the cursor position is taken as the scaling centre. Otherwise a mid point of the design is chosen as the scaling centre.

7.4.3 Rotation

The whole or part of the design can be rotated in the plane of any given axis. Selecting the “ROTATE” command initiates a prompt for the X, Y or Z axis. The rotation angle is then input via the number menu. If the cursor is currently sliding on an object C/L the rotation only applies to that object and all other objects that are chain linked. The centre of rotation is taken to be the cursor position. If the cursor is not sliding the rotation is applied to the whole design and a mid 3D point of the design is chosen for the centre of rotation.

8 SIZING THE DESIGN

8.1 The Default Object

The default object has already been mentioned briefly. It is simply the type and size or dimensions of any new object whose dimensions have not been specifically set. When you first position a centre line in the room it automatically assumes the default form:

8.1.1 Default Type

The default object is of the simplest type, a pipe or cylinder. The length of the object is set when it is placed in the design. Therefore only two other dimensions are required to fully define it; the diameter of the cylinder and the position of its seam.

8.1.2 Default Diameter

The diameter is indicated at the bottom of the screen when a new C/L is active in the room. This size is simply chosen to give a good visible image before specific dimensions are set.

8.1.3 Default Seam Position

The seam position of the default object is set at 0 degrees. This is the position vertically above the C/L. The angle is measured clockwise looking from the first end of the object to the second end. This is easier to visualise by using the seam position command (discussed later) which allows it to be reset using the cursor and in doing so, shown as a thick line.

8.1.4 User Default

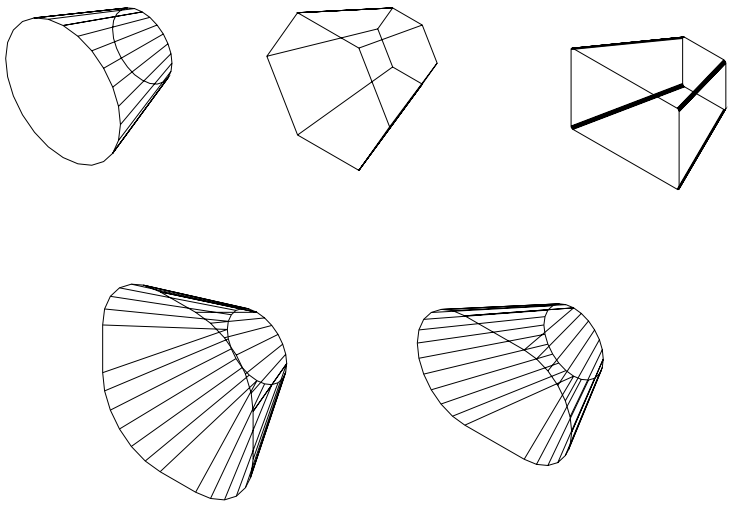
A default diameter of new objects can be set using the default command in the "Data" menu. This size will be used for any new objects provided they are not altered by the "Add Matching" facility.

8.2 Dimensioning An Object

8.2.1 Cross-section Selector

The cross-section selector is a name given to the two columns of selectors in the

design menu that appear on selection of the “SECTION” command which is itself always available. When “SECTION” is selected the list of available cross-sections appears in the middle section of the design menu, temporarily replacing the current tool page. The available cross sections are circular, regular sided, rectangular, adapter, height-width-diameter, proportional, oval, root radius rectangle, diameter, height, width and root. At this point the cursor is removed and the top selector is highlighted (i.e. the circular). If the mouse is moved up and down the highlight will jump from one section to another. A second press of the button will select the cross-section currently highlighted. The procedure for setting an object's cross-section and dimensions comes under the subject of "growing".



Basic Cross Sections

8.2.2 Pipe Growing

To setup an object as a pipe, assuming that it has already been set by default to be a pipe or is still in the form of a C/L, take the following steps:

I. Select the “CIRC” cross-section operator from the design menu as described in the previous section. This will initiate the "sniffer" (see sec. 6.7.4).

ii. Using the sniffer, select the object to be dimensioned. The cursor should now be restricted to sliding along the object C/L.

iii. Place the cursor somewhere midway along the C/L and press the left hand button. The cursor will disappear and if the C/L was visible, then it will be erased and replaced with the uncut object at its current dimensions.

iv. Moving the mouse up or down will "grow" the diameter of the selected object. At this point the dimensions of the object are written to the bottom of the screen. These numerical values can be used to set the desired dimension. If the diameter cannot be set exactly to the desired dimension due to the coarse increment with which the pipe is "grown", then it may be necessary to first zoom in to give a finer increment.

v. When the desired diameter is reached, the left-hand button should be pressed to fix the object dimensions at these values. The cursor will re-appear.

8.2.3 Cone/Reducer

The procedure is similar to dimension an object as a cone or reducer. Select the circular cross-section command as before and use the sniffer to identify the object to be dimensioned. Now instead of sliding the cursor to some midway position, as in (iii) of the last section, this time move the cursor to the end where dimensions are to be altered. This end can now be "grown" to the required size without any alteration to the diameter at the other end. As before when the diameter is set, press the left-hand button to accept it.

8.2.4 Cone Included Angle

It is often the case that a particular included angle of cone is required rather than, or as well as, particular diameters at the object ends. One of the dimensions written to the bottom of the screen is the included angle, therefore this angle can be set to the required value. If it is also necessary to set the diameters, then the cross-section operator can be re-selected and by following the procedure given in sec. 8.2.2, the cone can be grown in a manner which alters the size of both ends without altering the included angle of the cone.

8.2.5 Regular Object (with taper)

Other cross-sections can be produced from sheet by folding the material. The first of these is the regular sided cross-section which can be anything ranging from a triangle to a regular twelve sided cross-section (i.e. a dodecagon).This command may have fewer practical uses, except perhaps in the case of a square, but it forms a natural bridge between the previously discussed circular cross-sectioned objects and the more practical objects discussed later.

The regular objects are treated in a very similar way to the pipes and cones. The differences in the procedure are:

I. Instead of selecting the “CIRC” cross-section operator as in sec. 8.2.1, the “REG” cross-section should be selected.

ii. When the regular cross-section is selected the sniffer will not be activated immediately as before. First the number menu will become active. This is requesting the number of sides the regular sided object is to have. As already mentioned, any number can be selected from 3 to 12.

Having selected the number of sides the object is to have, the procedure is followed as that of a pipe or cone. One or both ends can be "grown" to produce a straight or tapered object. The dimensions of the object are similar to that of a cone with the diameters and angle now representing that of the outside cone of the object. Or in other words, the dimensions of the cone on whose surface lie all the object fold lines. This is the most convenient way of dimensioning a regular object with any number of sides.

8.2.6 The Z-Angle

An additional dimension is written to the bottom of the screen. It is referred to as the "Z-angle" and is the angle of one side of the object to the horizontal plane (also referred to as the Z-plane or XY-plane). To begin with, the uppermost side is the side to which the dimension refers and the angle is originally set to 0 degrees. Therefore, this side of the object can be seen to be level with, or flat to the horizontal. Having grown the object to the required size, this dimension can be set using the seam position operator in the design menu. The procedure is identical to that of setting the seam position on a pipe or cone although the effect is a little different.

I. Select the “SEAM” command. The sniffer is now set.

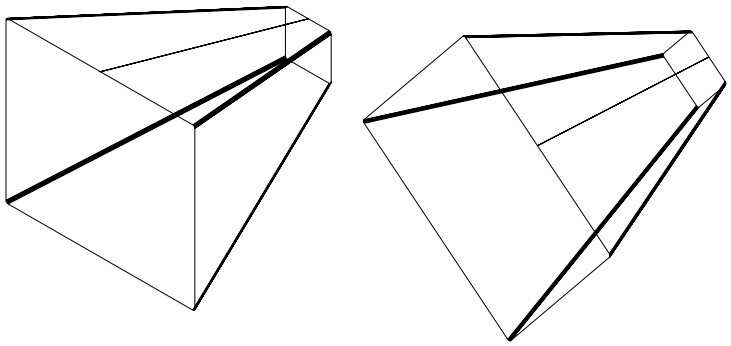
ii. Select the desired object using the sniffer. This will restrict the cursor to its C/L.

iii. Move the cursor to one end of the C/L and press the left-hand button. If the cursor was at a midway position when the button was pressed, there will be no response. If the cursor was at one end, then if the object cross-section is circular, the seam position is set as described earlier. If the object is of any folded or sided type, the cursor will immediately move to the first fold line in an anti-clockwise direction from the centre of the object side to which the Z-angle refers. Movement of the cursor is now restricted to "orbit" around the selected end of the object and in doing so, the object is rotated around its own C/L.

iv. The "Z-angle" dimension replaces the otherwise seam position dimension at the bottom of the screen. This can be used to set the Z-angle to the desired

value.

v. Press the left-hand button to accept a setting.



Object Zangle

8.2.7 Folded/Sided Object Seam Position

The seam position on any folded or sided object is kept to the centre of the side to which the Z-angle dimension refers. Therefore, by making the seam visible the "Z-side" can easily be identified.

8.2.8 Rectangular Object (Ducting)

A rectangular cross-sectioned object is often referred to as a "duct". If the “RECT” cross-section operator is selected, the number menu is not needed as in the case of the regular sided sections. The sniffer is used as before to identify the object to become a rectangle. As before, either end can be dimensioned separately or both ends together if required. There major differences in this case are as follows:

I. When the object is being grown to size, there are two dimensions to set. These are the height and width. At first, the mouse can be used to alter both dimensions at the same time. Vertical movement will alter one dimension and sideways movement will alter the other. The object height is set first using the vertical mouse movement. When the left-hand button is pressed, this dimension will be permanently set and only the width will be active.

ii. A second press of the button will set the width.

iii. At the bottom of the screen the diameter dimensions are replaced by the width and breadth dimensions which can be used to set the required values. The Z-angle dimension is set as before.

The rectangular cross-section can be used to form simple straight pieces of ducting or to adapt from one size of square or rectangle to another.

8.2.9 Complex Cross-Sections/Adapters

The cross section of an object can be made to combine the attributes of height width and diameter. If all of these dimensions are none zero the result is a rectangular shape with rounded corners. If the height is set to zero we have an oval. If the diameter is set to zero we have a rectangle. If both height and width are zero we have a circular section. Of course this provides a infinite variety of cross section shapes which are further enhanced by the fact the opposite ends of the objects can have totally different shapes. We can thus easily adapt from one section shape to another.

8.2.10 Square-To-Round Adapter

A common object in ducting work which can be very time consuming to produce by drawing techniques, is a square-to-round adapter. Although usually referred to as a square-to-round, the term often includes adapters which go from rectangle to round. This is best created using the fourth cross-section command available in the cross section list, identified by the “RECT” cross section operator..

Other commands are given to individually adjust the component height, width and diameter of sections. This command differs in that is makes the object infinitely variable in that height, width and diameter are prompted for sequentially, first for one end and then for the other.

A similar method to the previous cases is used to set the object cross-section although there are again some significant differences. The procedure is as follows:

I. Select the adapter command.

ii. Select the relevant object using the sniffer.

iii. Slide up to one end and press the button. A midway position cannot in this case be accepted.

iv. The end selected becomes the rectangular end of the object. The width and breadth are set exactly as described in the previous discussion.

v. When both width and breadth have been set the "growing" operation moves automatically to the other end of the object which becomes the round or circular end. This is set in exactly the same way described earlier for a pipe or cone.

The adapter is produced in just one piece of material by treating it as a combination of flat sides and segments of oblique cones. The oblique cone is described in sec 7.2.20. The point at which a flat side joins a cone segment is drawn as a fold line although, in actual fact, the transition is smooth and no fold is necessary.

8.2.11 Width-Height-Diameter

The cross section of one end of an object can be altered in both width height and diameter using the “ADAPTEND” cross section operator. This operator may be more useful than the adapter operator if one end is to remain the same.

8.2.12 Proportional Growing

Using the “PROPORT” cross section operator the cross section of an object can be made to grow proportionally. This means that the height, width and diameter attributes are of the object remain proportionally the same so that the shape of the section is simply enlarged. The procedure is similar to that described previously.

8.2.13 Oval

Using the “OVAL” cross section operator the selected object is grown but the height dimension is forced to zero at the selected end, or at both ends if a mid-point was selected. An oval is therefore a shape consisting of two round ends and two flat sides.

8.2.14 Root Radius Rectangle

The “ROOT” cross section operator forms an object that is basically rectangular but has a root radius in the folds. This radius is set from the "Root Radius" command in the "DATA" menu. Only the height and width of the rectangle are set. The root radius is usually very small but is useful to account for folds in thicker materials.

8.2.15 Adapter Diameter

Using the “DIAM” cross section operator the selected object end diameter is adjusted without any alteration to the current setting of the width and height dimensions.

8.2.16 Adapter Height

Using the “HEIGHT” cross section operator the selected object end height is adjusted without any alteration to the current setting of the width and diameter dimensions.

8.2.17 Adapter Width

Using the “WIDTH” cross section operator the selected object end width is adjusted without any alteration to the current setting of the height and diameter dimensions.

8.2.18 Root

Using the “ROOTEND” cross section operator the selected object end can be defined to have a rectangular shape with a root radius in the corner of the folds. This command differs from the root radius rectangle in that the other end of the object is not affected. You must set width and height in this case.

8.3 Seam Positioning

The position of a seam has already been briefly mentioned. This feature is often needed to ensure that the pattern produced is not in more pieces than is necessary, or to position the seam in a convenient position for the seam joint. Initially the seam is not visible but, unless its position has been reset in the default object, then it has a default position which is vertically above the object C/L. If the C/L is itself vertical then the seam is at the nearest point to one wall. The seam becomes visible as a thick line once the "seam" command has been selected from the design menu. The procedure for setting a seam position is as follows:

I. Select the "SEAM” command in the design menu (see fig.2). The sniffer will become active.

ii. Using the sniffer select the desired object.

iii. Slide the cursor to one end of the object and press the left-hand button. The end chosen should be the best end for visibility. Usually the largest or the nearest end. If the button is pressed with the cursor midway, there will be no response. If the cursor was at one end, then the seam will become visible and the cursor will be positioned at the point where the seam meets the selected end of the object.

iv. Movement of the mouse will now cause the cursor to "orbit" around the C/L end point, rotating the object and therefore moving the seam position with it.

v. When the desired seam position is reached as indicated by the seam dimension written to the bottom of the screen, press the left-hand button to accept. The seam will remain visible indicating that it has been set.

Seam positioning is another function which can receive numerical typed input. If any numerical key is pressed (i.e. '+,- or 0..9) during the seam positioning process, then an input box will appear to receive the value which should be the angular position of the seam in degrees.

9 CUT & WELD

9.1 Cut & Weld Definition

Initially, at the design stage the objects in the design are shown in their whole or "outline" state. These objects can be manipulated quickly and easily in "real time" (i.e. not in any pre-recorded way). The outline is not in any way affected by the other objects in close proximity but each object is cut off squarely at each end. Net lines (see later) are drawn at a fixed increment of 15 degrees for circular objects in order to give a good, visible image without excessive drawing or image calculation compute time.

The operation of cutting and welding an object is a more lengthy process. The result is that the object is cut to fit all other objects in close proximity and is allowed to extend beyond its end to form tees, bends and junctions as required.

Depending on the setting of the plotting increment there can be much more data involved. This cannot be manipulated in the same way as the object outline but is rigidly fixed. If changes are made to the object then the plot data will be disposed of first. In most cases this is essential anyway because a change in position or object dimensions will usually produce a different cut.

If the example of using the cut and weld operation given in sec. 6.9.1 was followed, then its function will be easily appreciated. In this section, some of the more detailed aspects of the cut and weld operation are covered.

9.2 The Process

The process of cutting and welding involves three main stages. These stages are also involved in producing a flat pattern. They are reported to the bottom line of the screen as they occur and are as follows:

I. **Analysing:** This is the process of comparing two particular objects in the design to prepare for the intersection calculations. During a cut and weld or pattern calculation operation, this will be the first part of the process and may reoccur intermittently during the calculations.

ii. **Calculating:** An inter-section calculation is being made to compare the object with one other. The resulting data are kept in memory until all comparisons have been made.

iii. **Sorting:** The calculation data from all the inter-sections are sorted to give a single pattern plot. When the process is complete, only the sorted data is retained in memory. The calculation data is disposed of.

iv. Converting: The plot data is converted into the required form for plotting the result to screen. The type of conversion depends on whether a pattern or a cut object is being plotted and in which view the cut is to be displayed.

To produce an object cut and weld plot, the process will begin with the calculations as reported to the bottom of the screen. When the calculations are complete, the sorting and converting procedures will follow being repeated twice alternately.

9.3 Unfold

The compute menu holds another command called "Unfold" which as the name suggest is for examining unfolded patterns produced by the objects in the design. It works in much the same way as the cut and weld command. The process of "Analysing", "Calculating", "Sorting" and "Converting" is indicated as before but the difference is in the resultant display.

As each pattern is calculated it will be drawn full size in the design area. If "ALL" was selected for the whole range of patterns in the design to be calculated then they will be displayed in turn as the calculations for each pattern are completed. The final pattern will remain on screen until a key is pressed.

Selecting “Unfold” will always cause the selected objects patterns to be recalculated, therefore if the design has been changed since the last pattern calculation the Unfold command can be used to examine the new patterns. If the pattern is to be examined without recalculation then the flat viewer utility in the design menu can be used instead (see sec. 9.13).

The ability to calculate the whole range of patterns in a single operation is useful to save time when creating a sheet arrangement. The data for each pattern is immediately available therefore saving time in possible lengthy calculations during the operation.

One further consideration in calculating a pattern is that the pattern can be produced from either the inside or outside. This is set by the "In/Outside" command in the "Options" menu.

9.4 Object Surface Lines

Object surface lines are lines running longitudinally along the object surface. If the object is not tapered they are parallel with the object centre line. If the object is tapered they are set at the taper angle to the centre line of the object. In Sheet Lightning there are several types of these longitudinal lines: “radial” lines, “fold” lines, “net” lines and “propagated” lines. The last type of line - “propagated” lines (discussed later in this section) are always invisible. Any of the others may be visible but this depends on whether the line type has been turned off in the “Output|LINES” menu. By default “radial” lines and “fold” lines are visible, but “net” lines are invisible. When intersection calculations are performed a single

calculation is made for each line around the circumference of the object, irrespective of what type of line it is. Therefore the more lines the finer the calculation of the intersection is. Also the length of time taken for the intersection calculation to be performed is directly proportional to the number of lines calculated. Each type of line is discussed in the following subsections.

9.5 Radial Lines

An object, such as a pipe, has a set number of radial lines. They appear at equal angular increments around the objects circumference. By default each object has 24 radial lines, therefore giving a radial line at every 15 degrees for a circular section object. This number can be reset via the “Compute|DATA|Radials” command. A new value can is chosen from the number menu or can be typed in when the number menu is active. Normally the radial lines are the only visible lines for the circular sectioned objects giving an equally spaced surface line to display the contours of the surface.

9.6 Net Lines

Net lines are additional invisible lines inserted into the circumference of the object to ensure the intersection calculations are made to the required level of accuracy. The number of net lines inserted depends on the setting of the angular calculation increment *for each object*. This calculation increment is set via the “Compute|Increment” command. If the increment is set to a angle lower than that imposed by the number of radial lines then net lines are inserted at equal angular spacing between the radial lines to ensure the gap between any one line and the next does not exceed that set for the calculation increment of the object. The default calculation increment is 15 degrees, therefore no net lines should be inserted in a circular sectioned object unless the calculation increment is given a lower value.

9.7 Increment Setting

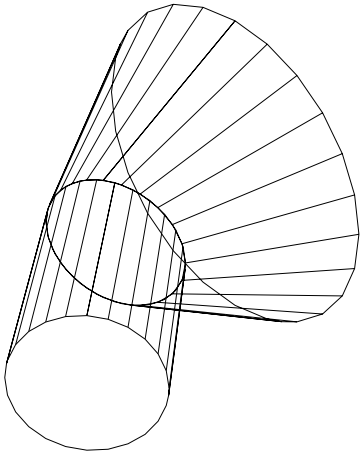
Setting the calculation increment for the “Cut & Weld” operation is achieved by selecting the "Compute|Increment” and then choosing the objects to be set using the object selector. The procedure is as follows:

I. Pull-down the "Compute" menu and select the "Increment" command. This will activate the object selector.

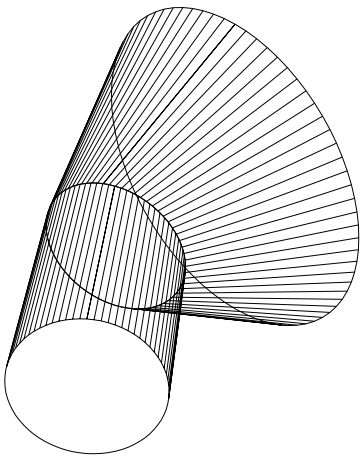
ii. The choice is given to set every object in the current design to a single increment value by selecting “ALL", or to set an individual object using the object selector.

iii. Once set, this increment is always used in any intersection calculation until reset by the same means. Also, if the design is saved, the increment for each object is saved with it at its current setting.

The minimum increment which can be set by the number menu for cut is 1 degree. This should always be more than sufficient for visibility purposes. Smaller increments can be set by typing a value when the number menu is active. It is also possible to force recalculation of objects at a smaller increment for pattern data prior to despatching output. In this situation, a very fine increment may be required to give an extremely accurate result. This is covered in a later section.



Draft Accuracy Increment



Output Accuracy Increment

9.8 Fold Lines

Fold lines are lines that divide one section of an object from another. This only occurs on objects of a more complex cross section that circular, such as regular sided, rectangular or adapter objects. For example at the point around the circumference of an adapter object that the surface changes from flat to circular a fold line would occur. Any lines actually dividing up the circular part of an object are radial or net lines, not fold lines.

9.9 Propagation Lines

The principles of propagation are important to understand to be fully aware of exactly what takes place when objects are cut and welded or when patterns are generated. There are a number of options available in the way propagation occurs including the option to turn propagation off completely. These are controlled from the "DATA|Propagation" toggle command.

The process of propagation occurs while an objects intersection with other objects is being calculated. What happens is: when it is determined that one of the existing lines (a radial, net or fold line) intersects with another object a propagation line is inserted in the other object at the point around the object circumference where the intersection occurs. This is a normal object surface line that is always invisible in both 3D image views and patterns. The line is only inserted in the intersected object if there is not already a radial, net or fold line in the object at that angular position already (within a small tolerance). If a line already exists the propagation line is unecessary and is not inserted.

The implications of a radial line being inserted is that when the system comes to do the reverse calculation (i.e. the intersection of the object receiving the propagation line with the object that caused it) then there will be a better fit of object to object because the propagated lines ensure the objects have the same points of intersection with each other along these lines. Please note, however, that to obtain a perfect fit the latter object should also propagate its own net, radial and fold line back to the first object. If this is done then the intersection of the first object must be recalculated to include the propagated lines. If the "DATA|Propagation" toggle command is set to "2-Pass" that is exactly what happens. The cut and weld operation must be for "ALL" objects and the propagation passes all the way through the design and then back down to the first object calculation. It should also be understood that if an object receives a propagation line, that line may then propagate itself on to still other objects intersecting with it. For a complex design this can result in a mass of invisible data but a very fine and accurate plot. It is recommended that you use it when creating 3D output. On the other hand it can be unnecessary for pattern data and may therefore be turned off completely by setting the "DATA|Propagation" toggle to "None". Each object is then considered individually to the accuracy set by the object increment without propagating any lines to objects it intersects with. This can result in a perfectly acceptable unfolded pattern plot. Finally the "DATA|Propagation" toggle can be set to "1-Pass" which means it does propagate lines but does not propagate back and perform a second calculation on the object. The "1-Pass" setting is the default and is perfectly adequate

for the vast majority of cases.

Finally we may make the point that when working with pipe of the same size propagation tends to have no effect. This is because the object surface lines of these objects meet anyway so there is no need for any propagation lines to be inserted. Propagation only really comes into play when the objects are different in some way. For example the intersection of two cones with different angles of taper. The normal radial and net lines of these objects would not naturally meet, so propagated lines become useful in enhancing the intersection accuracy.

9.10 Time And Memory

As always, the cost of reducing the calculation increment is that the number of calculations to be done is increased and therefore the time required to perform the intersection is proportionately increased. Also the results of the calculations have to be stored in memory and therefore the greater number of results will require more computer memory. These are common problems to the computer user. The computer system you are using and the amount of available memory will obviously determine the maximum level of accuracy that can be achieved but in most cases sufficient memory should be comfortably available to give the desired level of accuracy.

9.11 Automatic Memory Disposal

Sheet Lightning contains facilities to automatically manage the allocation and deallocation of memory. If a memory shortage occurs, it will automatically try to find or obtain memory in order to avoid aborting the operation. This may mean that some cut and weld data will have to be disposed of in order to free memory for the required purpose. In doing so, both the pattern and the cut and weld data associated with any particular object will be lost and the object replaced by its uncut image at the next redraw.

Memory is allocated for the following purposes:

(a) To store cut and weld data.

(b) To store pattern data.

(c) To store blocks of screen memory when using popup menus, dialog boxes, input boxes or verify boxes.

Automatic disposal will occur if necessary to make room for any of the above requirements. When it occurs, it is indicated by the word "Disposing" which is reported to the bottom line of the screen accompanied by a figure giving the current memory availability. This figure also appears during any other reported operation and therefore gives an indication when memory availability is low.

Of course the probability of memory shortage depends upon the system being used and the accuracy of drawings being produced but for most users memory availability is likely to be more than adequate to prevent any such situation arising in the normal operation of the software.

9.12 Object Visualisation

Drawing objects with all edges and corners visible on all sides is known as "wire frame" drawing. This is the way all objects described so far have been drawn but it is not the only way they can be visualised. It is also possible to visualise any individual object in a "solid" form providing the images on screen were calculated with the "DATA|Flexibility" toggle command set to "Rigid". This facility is available by using the "viewer" commands in the design menu. The "viewer" also includes the facility to view the flat pattern together with the solid and/or wire frame object. The viewer commands are the “WIRE”, “SOLID” and “FLAT” commands.

Note that the appearance of outline or cut objects in the design environment are enhanced by the removal of object surface lines on any inside view of the object. Only outside views of an object surface are shown with these lines, except in the case of fold lines on folded objects such as those with regular sided or rectangular cross sections. The fold lines are always visible both on the front and rear side of the object. In these cases, the fold lines give sufficient visibility therefore no radial lines are included, only net lines. The same is true for adapter or square-to-round objects although as previously discussed the fold lines shown are not in actual fact folds but rather a transition from one part of the cross section to another.

9.13 The Viewer

The purpose of the viewer is to enable the design to be visualised more thoroughly before accepting it, often in preparation to despatch. To activate the viewer simply move the cursor to one of the view selectors in the design menu and select. The cursor will disappear and left-right movement of the mouse will cause a highlight to move across the three view selectors The following sections give the three commands available:

9.13.1 Wire Frame

If the wire frame view is selected the object selector will become active which gives the command of viewing an individual object or the whole design. The main purpose of the viewer is to allow a cut object to be visualised alone, thus removing any other object that may obscure it in the "assembled" view. When the object is selected it is drawn in its original position on the screen but this time with the background and other objects removed.

9.13.2 Solid

If the solid view is selected the object selector again becomes active but this time the option of viewing the whole design is not given because solid objects can normally only be viewed individually. The selected object is redrawn as a solid view in exactly the same position as before with the background removed.

9.13.3 Flat Pattern

If the flat pattern viewer is selected the flat pattern of any object can be viewed alongside its solid and/or wire frame view. The object selector is activated as in the first section where any individual object or the whole design can be selected. Various view modes exist to support this function. The basic view is always produced. This includes the flat pattern which is drawn in the top left-hand quarter of the design area and the solid view which is drawn in the lower right-hand quarter. Both drawings are scaled as required to fit the area quarter with the maximum sized view.

9.13.4 Multi-View Modes

Using the options menu, various view modes can be chosen.

FLAT gives a view of the flat pattern only, sized to fit the design area.

SOLID gives the flat pattern sized to fit the top left quarter of the design area along with the solid drawing in the lower right quarter.

BUILT gives two additional views. In the lower left hand quarter the wire frame of the object is drawn and in the upper right quarter, the whole of the design is drawn.

All views are scaled to fit the quarter of the screen in which they are placed. To change the view mode use the toggle in the "Options" menu under the title "View Mode".

9.13.5 Multiple Solid View

You may have noticed that during the discussion of the solid view, that it was stated that multiple solid views are not "normally" possible. The fact is that the facility only exists to draw a single solid at a time. There is a way, however, of causing all of the solid objects to be drawn to the design, one after the other without erasing the previous as the viewer normally does. This is simply done by pressing the hotkey F8 when the viewer is active with a solid object drawn. It should be noted here that not

all views will draw correctly in this way, the reason being that one object can easily overdraw the other. Often it is possible to obtain a properly drawn solid object view by placing the objects in a set order in the design to prevent overdrawing. This can be achieved using the organise command in the “GRID” menu.

9.13.6 Multiple Solid And Wire View

Another facility exists which is subject to the same restrictions as the previous one. It is possible to add individual objects into a design one by one, in either a wire or solid form. This is done by pressing the hotkey F7 when the viewer is active which prevents the erasing process between viewer selections. A complete wire and solid view picture can be built in this way.

9.14 Flexible Data

The above discussion of the viewer and the ability to visualise solid objects in addition to wire frames is dependant upon the setting of a toggle command in the "Data" menu called "Flexibility". This is a toggle option which can be set to either "Rigid" or "Flex". The solid object can only be viewed if all intersection calculations have been performed with this command set to "Rigid" (for further discussion of the flexibility command see later).

10 MATCHING OBJECTS

10.1 What is "Matching"

When two objects are related in the form of a tee, bend or junction then a "matching" operation can be performed. The operation of matching involves automatically altering the dimensions of an object to fit the dimensions of a related object in a particular manner. The various matching commands available are accessed through the "Match" pull-down menu or through the match commands in the design menu.

10.2 Pipe Bends

To illustrate a simple case where matching may be required, consider a pipe bend relationship. If two objects are related together as a bend, then they must be of such a size to ensure that they cut each other off completely when put through a cut and weld operation. If, for example, the two objects are pipes (or cylinders), it would be impossible to produce a bend where the pipes were of different diameters. They would no doubt intersect each other but they would not properly cut each other off to form the bend. Therefore, a bend involving pipes can only be successful if the pipes are of the same size. The operation of matching alters the object dimensions to ensure that this is in fact the case.

10.3 Cone Bends

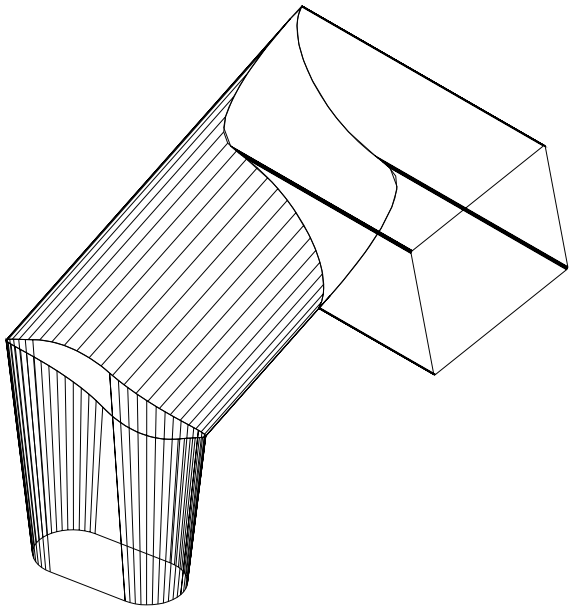
In the case of cones with different angles of taper, the calculation is not quite as simple. A bend could not be formed by simply matching the same diameters. The sizes have to be calculated to perform the match correctly.

10.4 Irregular Bends

In the case of bends formed between irregular objects, the situation becomes a little more complicated. Any draftsman familiar with the drawing techniques used to intersect pipes and cones will know of the "common central sphere" technique. This technique in effect makes use of the regularity of the pipe or cone as is in fact used in the system to achieve a straight cut for inline intersection (see later). An irregular bend will not usually conform to such criteria.

Consider a bend formed between oblique cones with their oblique offsets in different planes. Or the formation of a bend between an oblique cone and a rectangular tapered duct. In these cases, the common central sphere technique would not suffice. SHEET LIGHTNING takes care of the operation but there are various options available which alter the way in

which the objects are matched.



Matched Irregular Intersections

These options fall into two categories:

(a) The position of the C/Ls of the objects must be adjusted in some way to place them in a central position relative to each other to form an exact fit. For irregular objects, this will often mean that their centre lines do not in fact meet but cross at an offset distance. C/Ls that do meet, will sometimes have to be parted to form the required bend.

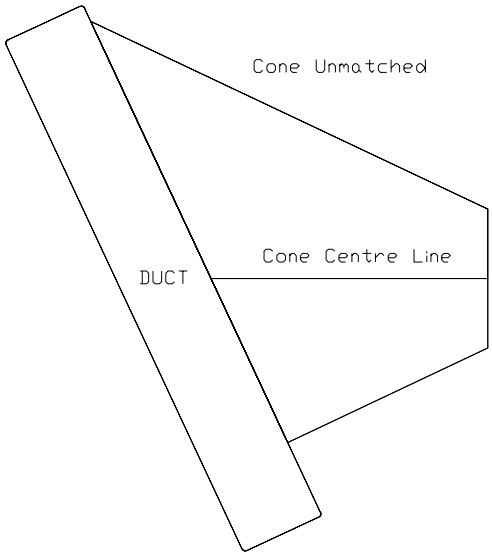
(b) The dimensions of the objects may have to be adjusted to complete the fit so that in a cut and weld operation both objects are cut off completely.

Note that these adjustments are often inter-dependent which means that adjustment of one will necessitate re-adjustment of the other to maintain the fit.

There are alternative methods for achieving both of these requirements that are set using the match menu commands. These are as follows:

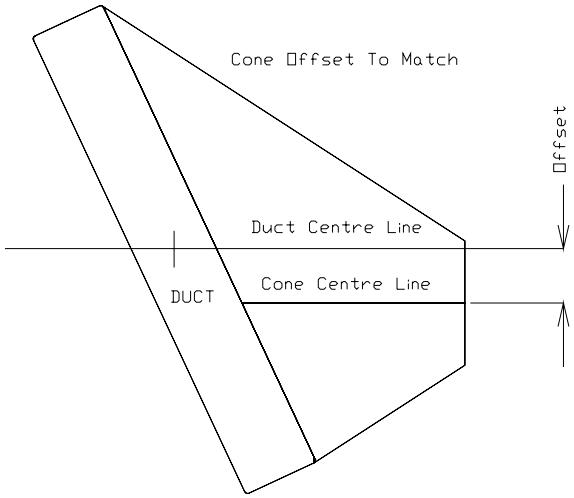
10.5 C/L Displacement

The mode of C/L displacement is controlled by the displace option in the match menu. The examples demonstrate the various settings and their effect on a matching operation. An irregular match between a cone and a duct is used illustrate all cases:



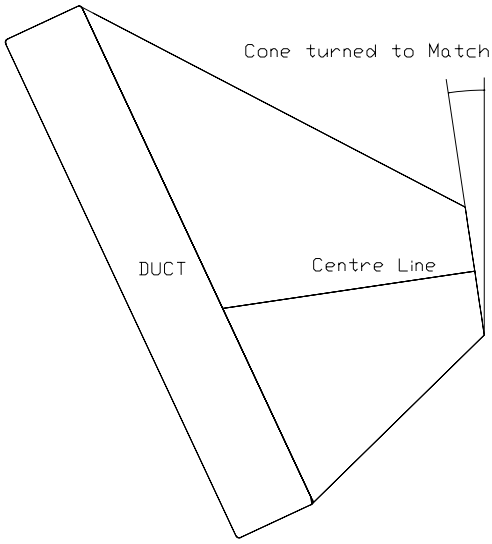
Before Matching

OFFST - The offset option causes any adjustment of the object C/L to move both ends in the same direction by the same amount, thus preserving orientation of the line.



After Offset Taper Matching

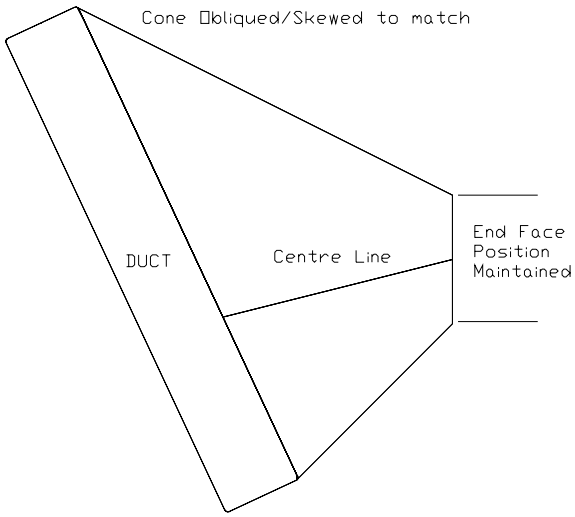
TURN - This option permits adjustment of the C/L position by moving one end only. This in effect pivots the C/L about its other end.



After Turn Taper Matching

OBLIQ - This option only allows the oblique offset of the related end to be changed with the other end remaining in exactly the same position. The advantage of this is that the opposite end of the object remains exactly as it was before adjustment which is a feature

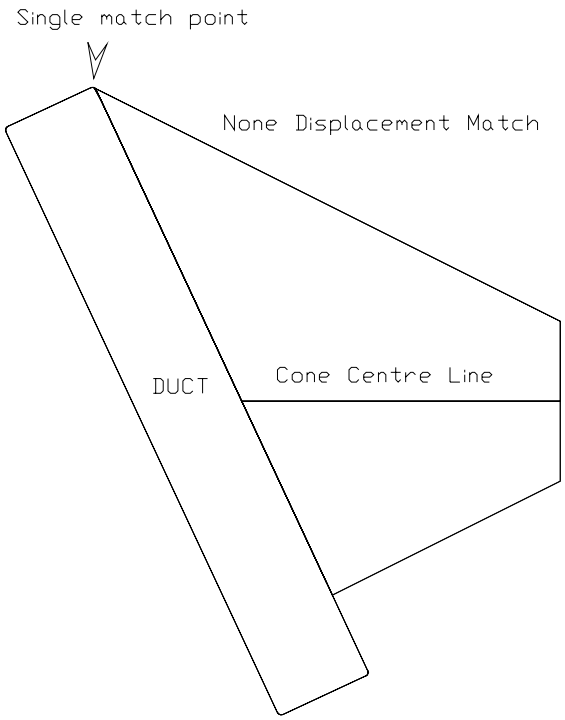
that may be useful when that end of the object is related to another.



After Oblique Taper Matching

ASK - If the toggle is set to ASK, then you will be prompted with a verify box to select which of the options is to be used. The box will appear before each matching operation takes place.

NONE - No displacement of the object is permitted. This feature is intended for use with tees and junctions. Of course, if a bend relationship is to be established, then the objects must be matched. With tees and junctions, that is not always the case, therefore the option is useful when an offset tee is to be formed where only one edge of the tee is matched to the other object. The offset can only be maintained if the C/L centralisation is disabled. In the case of bends, the NONE setting is over-ridden and behaves instead as though the toggle were set to ASK.

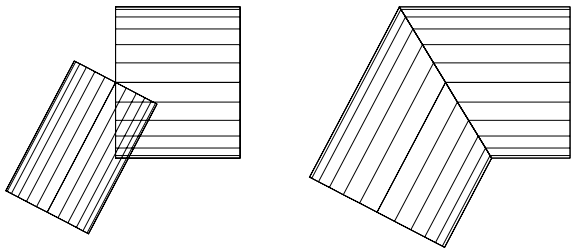


After None Displacement Taper Match

10.6 Object Dimension Adjustment

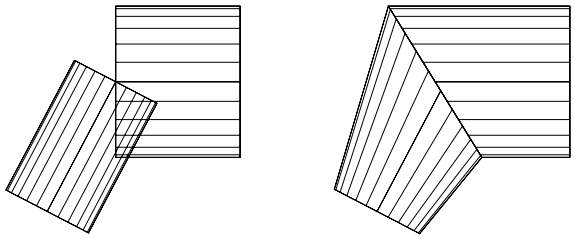
The object dimensions adjustment is controlled by the "Dimension" command in the “Match” menu. There are four settings as follows:

GROW - This option causes the whole of the object to be "grown" in order to achieve a matching size. The grow option is extremely useful to propagate a size adjustment right through a complete design. This will become clearer when the chain matching is described later.



Grow Matched Bend

TAPER - The taper adjustment will only allow adjustment of the object dimensions at the related end. Therefore the dimensions of the other end are preserved.



Taper Matched Bend

ASK - When the dimension option is set to ask, then a verify box will request which type of matching is to be used before each matching operation is undertaken.

NONE - Prevents any adjustment being made to the object dimensions. This is often used to centralise a tee object to the C/L of a related object without adjusting any dimension. As with the displace option in the case of bends, this setting is over-ridden and causes the operation to behave as if it were set to ASK.

One final point to note when using the dimension option, is that in the case of a rectangular end of an object, any adjustment will always preserve the relative proportions of width and height.

10.7 Tees

The calculation for tee relationships is similar to that of bends but less restrictive. In this case, it is only necessary that the one object is cut off fully by the other rather than for them to cut each other off. The diameter of teeing object therefore has a maximum size but there is no minimum. The teeing object can be smaller than its maximum size and still be cut off. In this respect there is more freedom of choice in the sizing of an object. Despite this, the matching operation always works to the greatest size possible in such a tee. Smaller sizes must be set manually and the matching of these objects should be disabled or refused if the maximum size adjustment is not required.

10.8 Junctions

Junctions are even less restricted and can practically be sized as required. In this case the operation of matching will still fit one size exactly to the other in the same manner as the tee. Differing sizes can be set manually.

10.9 Copy Angle

The copy angle option is an additional feature. With this option enabled, the taper angle of one object can be passed on to another. The default is for the option to be disabled and the taper angle of each object is treated individually. The feature is useful however in such cases as forming a lobster-back bend. A constant angle of taper can be transmitted throughout the bend using a chain matching operation (see later).

10.10 Add Matching

The "Add Matching" command is a toggle which can be set the "On", "Off" or "C/L". When "Add Matching" is enabled, any new object placed in the design will immediately be matched to any related object as soon as it is de-activated by placing the second end in position. By default, this feature is enabled but can be switched off from the match menu. If any of the matching options are set to ASK the relevant verify boxes will appear requesting the type of matching to be employed by the new object.

The third value "C/L" will prevent a new C/L being transformed into the default object when placed in the design. Instead it remains as a C/L and can therefore be used as a dummy line. If a dummy line is to be changed into an object use the "Outline" command in the design menu.

10.11 Chain Matching

Chain matching is a feature which allows a complete design of related objects to be re-

matched in one sweep. There are two ways of initiating such an operation.

I. When an object's dimensions are changed using the growing facility, provided the match command in the match menu is set to "ON", a chain matching operation will be automatically initiated. If matching is not required, then the command should be set to "OFF" or "ASK". If ask is set and the verify box requests whether matching should go ahead then either NO or the right-hand escape button will prevent the match going ahead.

ii. Chain matching can also be initiated directly from the match menu. To do this select the "Initiate" command. The object selector will then be activated requesting where the matching should be initiated from. Alternatively the chain match command in the design menu can be used.

10.12 The Chain Matching Process

Once the process of chain matching has been initiated, the effect is as follows:

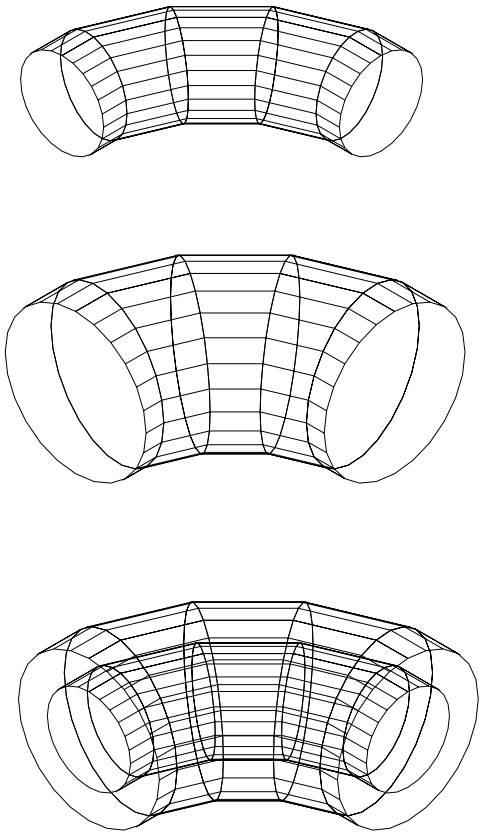
I. The selected object is searched for related objects. These are entered into a queue and then each one in turn is matched to its source.

ii. As each of the objects is matched, it too is searched for related objects which have not already been matched in the current operation. If any are found, they are entered in the queue and matched in turn.

iii. If several objects are found to be related and are added into the queue, the effect is a fanning out along the separate branches as the relationships are detected and the matching is performed. If the "Match Mode" option is set to ASK, then by choosing NO at the verify prompt the matching of the object is prevented and any related objects are not entered in the queue, thus ending all matching along that particular branch.

10.13 Twin Wall Using Grow Matching

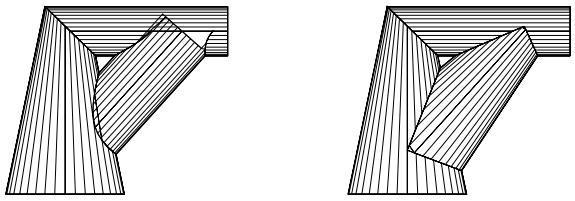
A useful and powerful facility of creating multiple walled designs is provided by the grow matching operation. A design can be copied to disk, "grown" to ac different overall size and then the original design appended over the top of the resized objects. Make sure however the "Non-Relative" command in the "Efficiency" menu is set to none before cut & welding the design.



Twin Wall Using Grow Matching

10.14 Ring Matching

When a matching operation is performed, a situation can easily be encountered where the sequence of matching one object to another encounters an object which has already been set or matched in the same operation. Take for example a situation where three pipes are joined together with bend relationships in the form of a triangle. The third pipe will try to continue the chain back to the one that was originally initiated. In other words a "ring" is encountered. In such a case a special kind of matching is available called "Ring Matching" which will adjust both ends of the object to fit their respectively related pre-matched objects. Because this can alter the dimensions of an object completely, the match menu holds an option called "Ring Matching" which can be used to prevent, or cause a verify request before ring matching is permitted.



Ring Matched Bend

10.15 In-line Objects

The situation is often encountered where objects inter-sect each other in-line or close to in-line with each other. It is clear to see that in this case, irrespective of the type of relationships that exist or the relative sizes of the objects, they will always cut each other off completely. In such cases the objects are copy matched where the dimensions of the related faces are set equal.

In the in-line situation, another problem arises in cases where the two objects are not in any way related. The problem is that the cut can be made in two different ways. The portions of the objects that are cut away and the portions which are left can be reversed to give an equally satisfactory intersection. SHEET LIGHTNING uses simple criteria to decide which ends of the object should in fact be cut away and which should be left.

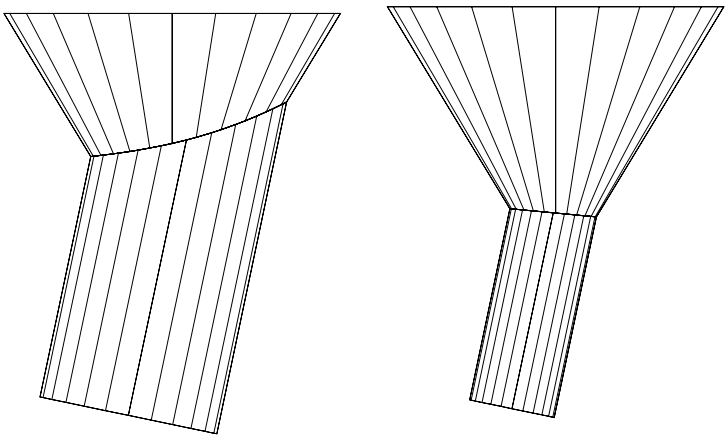
The criteria depends entirely on the relative positions of the C/Ls as follows:

I. The two C/Ls are projected onto a single line at half their relative angle.

ii. The midway point of each projected C/L is taken. The relative position of these two points decides which end of the object is cut away. The end of the object nearest to its projected midway point is kept. The other end is cut away.

A few experiments are recommended here in taking the examples given in the figures, moving their relative positions and performing a cut and weld operation. This can be done easily using the "dragging" facility described in sec. 7.3.3.

If a junction relationship is found to exist between two in-line objects, then the same criterion is applied. If on the other hand a relationship exists involving the end of one or both objects such as a tee or bend relationship, then that end will inevitably be the cut away end.



Inline Object Matching Scope

Note that an in-line situation does not necessarily imply that the C/Ls of the objects are exactly in-line. The dimensions of the object can cause an in-line situation to arise even when the objects are at a significant angle. This is detected in a cut and weld operation and the above criterion is applied.

10.16 Common Central Sphere

The common central sphere technique is used in the geometrical drawing procedures. It is used to get a correct match between intersecting right cones. The matching of right cones using the techniques described so far would in fact give exactly the same result, except in the in-line match case. In this case there may be may different sizes that perform a legal match. However if the common central sphere technique is applied to these cases it ensures that the objects are cut by a single cutting plane. In other words in some 3D view of the intersection the cut line is seen to be straight. The C.C.Sphere option in the match menu is therefore provided to deal with these as a special case. If the facility is disabled then normal face matching takes over.

10.17 Copy Matching

When two objects meet approximately in line with each other (two cones for example) it is often possible for them to have a range of relative sizes and still be adequately matched. In this situation normal matching would simply accept whatever sizes already exist. In such a situation it is often required that the faces off the two objects are the same size as each other. To ensure this face matching should be turned on. Whether the object is grown or tapered is determined as with other matching by the setting of the "Dimension" option.

10.18 Individual Matching

The individual matching process is much the same as the add match. The difference is that add matching will automatically match in any new object that is added to the design. Individual matching will allow any individual object already in the design to be matched to its relations. If the conditions exist where the chosen object is related to more than one other then a ring match will be attempted. To perform the operation select "Individual" in the “Match” menu and select the desired object using the object selector. Alternatively use the MATCH1 command in the design menu.

11 DESIGN MANIPULATION

The purpose of the “GRID” menu is to allow you to manipulate the whole design in various ways. These facilities are as follows:

11.1 Organise

The organise command is simply used to alter the order in which the objects appear in the design. This is necessary if the order of a cut & weld or a re-draw or some other operation matters. It is done by using the organise command which will then automatically activate the object selector. The procedure is as follows:

I. Select the organise command in the “GRID” menu. The object selector will become active.

ii. Select the first item to be moved. The object selector will flash off and then on again.

iii. Select the position that the object is to be moved to. The position of the first object will now be relocated immediately before the second.

iv. The object selector will now be activated again in order to allow the process to be repeated. Use the right-hand button to escape when no more objects are to be moved.

11.2 User Default

A default diameter of new objects can be set using the default command in the "Data" menu. This size will be used for any new objects provided they are not altered by the "Add Matching" facility.

11.3 Root Radius

There are various cross section available in creating a design. On of these is the Root Radius Rectangle. This is simply a rectangle with a small radius on the corners of the object. This value of the radius depends on the value set by this command. The normal purpose of this radius is to allow for folding of material and may be more significant in thicker material.

11.4 Lobster

The lobster back bend facility is described in detail in the main design section under special feature. It can be accessed both from the "GRID" menu and from the design menu.

11.5 Scale

The scale command employs an input box which prompts for a scale factor. This factor is used to increase or reduce the size of the whole design. The scaling centre is automatically selected as a central position of the objects in the design unless the cursor is currently sliding on a object C/L. If so then the current cursor position becomes the centre of rotation and only the that object and those linked chain linked to it are rotated.

11.6 Spin Grid

The whole design can be rotated in any of the three planes which make up the floor and walls of the room. As with the scaling command, a central position is automatically selected about which rotation is to occur. The procedure for performing a rotation is as follows:

I. First make sure that the toggle command "Rotate Axes" in the same menu is set to the required plane. This is a simple toggle selector. The "XY" selection will effectively cause rotation in the plane of the floor. The "XZ" and "YX" selectors will cause rotation in the plane of the left and right walls respectively when looking in the 3D view.

ii. Select the "Spin Grid" command. This will activate the number menu to allow an angle of rotation to be selected.

iii. Having selected the incremental angle the rotation will be re-cursively performed at the selected increment. The rotation is ended when the Left-hand button is pressed.

If the design is only to be rotated through a single incremental step then when the incremental angle is selected from the number menu the left-hand button should be pressed twice in quick succession. If further steps are required then the process can be repeated. Note also that as always with the number menu, typed input can be given.

11.7 Spin Direction

When using the “Spin Grid” command the number menu is used to select the incremental angle at which the objects are to be spun. The number menu selector can in this case be moved through 0 degrees into a negative angle in order to spin the design in the opposite direction. It should be remembered that downward movement of the mouse moves to a higher angle selection. Upward movement gives a lower angle and moves into the negative

values.

The positive direction of rotation can be easily identified as being an anti-clockwise direction when looking at the view in whose plane the rotation is set to take place (i.e. XY,XZ or ZY). Consequently a negative angle is a clockwise rotation.

11.8 Rotate Axes

As previously mentioned the "Rotate Axes" toggle can be set to XY, XZ or ZY. This indicates the plane in which the rotation will take place. The 3D command provides a quick indication of which plane is which.

12 DATA

The "Compute|DATA" menu holds commands which affect the way that wire frame images and patterns are calculated and/or represented. The reason for the different commands are given in the discussion of each case.

12.1 Removing The Net

A toggle command "Netting" is available under the "Data" menu to turn off all "Net" drawing. This will only leave the outline of the objects drawn to screen whether in the cut or un-cut state. This facility is available for two purposes:

I. In certain cases, particularly where the inter-sections are multiple or complex, the net lines may tend to obscure the view. By turning them off in this situation, it is possible to improve the visualisation of cutting lines.

ii. Performing an inter-section calculation with the net lines turned off will give a significant time and memory saving. Therefore, if there is a problem with memory availability, the net toggle can be turned off to improve the situation.

When the net toggle command is changed, the response is delayed until leaving the menu. This allows other settings to be changing before the redraw with or without net lines is executed.

One thing to note is that if the Cut & Weld intersection is performed with net lines switched off, no calculations for the net lines are performed at all. Therefore, for the cut object they cannot be switched on without re-calculating the intersection.

12.2 Show Seam

The Show Seam toggle has two settings, "Show" and "Hide". The default is set to hide, which prevents the seams of the design objects being shown. If the toggle is changed to "Show", there is no immediate response and the difference will not be seen until the design is re-calculated and redrawn. This can be forced by either changing the view or by using the redraw command in the "Compute" menu. The seam will then become visible as a thick line distinct from the net lines.

12.3 Propagation

This function is disabled in the standard version of Sheet Lightning. It's purpose is to enable the generation of 3D data output of a finite object mesh for stress analysis of SHEET LIGHTNING intersections. The data calculated for this purpose has an additional

requirement that the intersection net lines meet at precisely the same point rather than using the uniform incremental divisions normally employed for existing patterns and images. These facilities are of no consequence to pattern work.

12.4 Flexibility

The setting of the "Flexibility" command affects the ability to visualise solid objects in addition to wire frames and the manipulation of objects. The command is a toggle which can be set to either "Rigid" or "Flex". The solid object can be viewed only if all intersection calculations have been performed with this command set to "Rigid".

12.4.1 Rigid Data

The main disadvantage of using the rigid option is that when the view is changed or the design is spun the cut data will be lost and the intersections will have to be recalculated to regain their cut shape. The setting does however permit solid images to be viewed as previously mentioned.

12.4.2 Flex Data

This mode of data calculation has the advantage that the cut data can be transferred from one view to another without any data loss and the design can be spun and viewed from any angle in its cut state. The main disadvantages are that the seam is always visible and the edges of a circular cross sectioned object are not quite as distinct, especially with a coarse calculation increment.

If the setting is changed the existing cut data will remain in its original form with the corresponding restrictions until the object is recalculated. The setting has no effect on the flat pattern data.

12.5 Face Data

When a wire frame image is calculated each net line represents a single line of calculation. Rather than considering the object as a series of lines it is possible to consider it as a series of strips between the lines. These strips are better known as "faces". Calculation of the objects in this form can be useful in passing the data to a conventional CAD system. The 3D image can be output in DXF format which most CAD packages support. The faces can be used by the system to perform a hidden line image; something that would not have been possible with wire frame data.

Note that calculations within sheet lightning are normally kept to wire frame because the system has a partial hidden line removal facility which operates with the wire frames, it saves calculation and redraw time and it saves memory. Wire frame data is often twice as large as the normal hidden line data. The option is best used immediately prior to DXF

output.

12.6 Irregulars

This command is important when dealing with objects other that circular cross sectioned. It enables the intersection of any combination the irregular members. The command is not permanently enabled because there are a number of special cases still to be dealt with by future versions of the system. However the current capabilities in this respect are highly sophisticated but still simple to use.

It is possible, for example, to produce intersections between two or more square-to-round adapters, or a duct and a cone. The possibilities are endless.

All matching facilities work with the complex intersections in exactly the same way as circular objects.

If the facility is turned off all such object intersections are ignored and the objects cut square to its ends.

By default the facility is disabled and will remain so in future versions until all special cases have been dealt with.

Note that the setting of the command in no way affects the development of individual object pieces such as a single square-to-round.

12.7 Tee-Apex

It cases where a matched tee is used there may be sharp corners at the peak of the tee. To prevent them being clipped by the incremental calculations around the object surface an additional line can be inserted exactly at the thus giving a more accurate result.

By default the command is turned off to prevent unnecessary calculation for all other cases.

12.8 Pattn-Nets

By default the net lines are not shown on a circular cross sectioned flat pattern. These can be enabled by setting the "Pattn-Nets" toggle command to "On".

13 SHEET ARRANGEMENTS

The sheet is a rectangular area representing the paper size of a plotter or a sheet of metal or some other material to which the intended output is to be sent.

The term "sheet arrangement" refers to the placing and arrangement of patterns and wire frame images on the sheet which will constitute the output data to be sent to the device.

The section dealing with patterns is primarily concerned with creating a sheet arrangement using the facilities available in the Pattern menu which is accessed via the "Pattern" command in the Compute menu. This section is more concerned with preparing the sheet itself onto which an arrangement is to be placed and is therefore the fore-runner of creating an arrangement.

There are several facilities available for this in the Sheet menu which can be accessed through the "Sheet" command in the Compute menu. They are discussed in the following subsections:

13.1 Sheet Visibility

During the normal design stage, the sheet and any accompanying arrangement is invisible. It can be brought to visibility in either of two ways.

I. Selecting the include or wire frame command indicates that the creation of a sheet arrangement is to begin or continue. To do this, the sheet must be visible, therefore it is drawn to screen along with any arrangement that already exists.

ii. Selecting the hide sheet command. If the sheet is already hidden it will be redrawn.

13.2 Sheet Redraw Procedure

When a sheet redraw is induced in either of these two ways the following process begins:

I. The sheet is redrawn to the screen using the current sheet size settings and is scaled to comfortably fit into the design area. Note that neither the current design or the background are erased but the sheet is drawn over the current scene. If the scene is not wanted in the background then they should be turned off from the scene menu.

ii. If a sheet arrangement exists the redraw may entail recalculation of any cut and weld or wire frame data. Any such recalculation will automatically be done using the current increment settings of the objects (see sec. 9.7) and will be evident by the reporting on the bottom line of the screen. If any of the patterns or wire frames in the arrangement have already been calculated then the existing data will be used

without recalculation.

iii. If the right hand escape button is pressed during the redraw process the sheet redraw will be aborted and the view will be redrawn without the sheet or its arrangement.

iv. If the redraw was induced via the include or wire frame commands then the procedure of placing a new object on the sheet will continue.

13.3 Designing With The Sheet Visible

Even though the sheet has been drawn to the screen it is still possible to return to design mode and continue forming a design with the sheet still visible. It will not be removed until the next screen redraw is induced.

13.4 Hide Sheet Command

If the sheet is visible and you wish to move back into design mode with the sheet hidden, then the "Hide Sheet" command can be selected. This is usually preferable to creating the design with the sheet visible because it tends to obscure the view.

When the "Hide Sheet" command is used, all the data relating to the arrangements is retained. As previously mentioned, the hide sheet command can also be used to resume sheet visibility.

13.5 Manual Redraw

Selecting the "Redraw" command in the compute menu will not hide the sheet arrangement if it is currently visible. The screen retains its existing form.

13.6 Saving A Sheet Arrangement

Resaving the current design file will now also save the current sheet arrangement data with it. This data is not the actual plotting data but simply size and positional data of plots on the sheet; therefore the actual amount of data is quite small. Saving plots themselves takes considerably more disk space. This is covered fully in the section on despatching (see sec. 15).

13.7 New Sheet

As with the "New" command in the "File" menu, the "New Sheet" command in the “Sheet”

menu will clear the sheet arrangement ready for you to create a new arrangement. If the sheet is visible it will be automatically hidden.

13.8 Sheet Size

Before becoming involved in creating a sheet arrangement, it is usually wise to set the required sheet size. This may depend on the device to which the computer is to be attached in order to output the data.

As with the "Room Size" command used to set the design environment, the Sheet Size can also be manually set. The default size is length 3000mm by width 2000mm. These sizes are reset in the following way:

I. Select the "Sheet Size" command from the Sheet menu. An input box will appear requesting the "Length of Sheet".

ii. Type in the new value and press the left-hand or enter button. If the input is legal, the value will be accepted and a second box will appear, requesting the new "width of sheet". If the input is not legal, an error will be reported and the operation aborted.

iii. If there was no error type in the width of the sheet and press the button. The size of the sheet is now reset.

iv. If the sheet is already visible, then the current sheet and arrangement will be re-drawn with the new sizes.

The sheet will always be scaled to comfortably fit the design area when drawn on the screen. If the new scale differs from that used before the sheet size was reset, then the patterns and wire frames in the arrangement will be re-scaled accordingly. It should be remembered that although wire frames can be scaled and drawn as required, the patterns in an arrangement are always full size.

13.9 Parameters

The "Parameters" command prompts for three consecutive values. These are the X and Y origin positions of the sheet and the size of the border.

13.9.1 Sheet Origin

If the sheet size is not read from the plotter or device then it is essential that the origin is set to correspond to that of the plotter. By default the lower left corner of the plot is the origin (0,0) but some plotters by default use the sheet centre. If this

is the case then a manual setting would need to give a displacement of the origin to the centre of the sheet. The values required should be available either from the device manual or by physical measurement.

13.9.2 Border

This is the width of a border around all four sides of the sheet within the sheet size. Only the area within the border is the effective plotting area.

The reason for the border parameter is that some plotters cannot be set to a specific sheet size. Instead each time a sheet is loaded the plotter will move to the extents of the sheet and measure its size. This means that even though successive sheets loaded into a plotter are the same size, if the operator places the sheet in the plotter in a slightly different position then the sheet size reading will be different. Of course when a partitioned plot is to be produced which may spread across several sheets it is imperative that the sheets are the same size. The border parameter therefore allows a degree of variation in the sheet position to ensure that the same "frame" can be fitted within the sheet.

Note that to ensure that the sheet size is not to be re-read each time a plot is sent to the device the "Hard Clips" parameter in the plotter menu should be changed from "Read" to "Sheet". For multiple sheets the "Plotter" command in the Sheet menu should used to read the sheet once and thereby set its size using the current border setting.

13.10 Plotter Sheet Sizing

Usually the determining factor regarding the size of sheet that should be used in creating an arrangement is the physical limits of an output device. If known these can be manually set using the "Sheet Size" command as covered in the previous sections or alternatively a plotter's hard clip limits can be read directly from the device, provided that it is a serial port device and is capable of passing information back to the computer. It is done in the following way:

I. Select the "Plotter" command in the sheet menu.

ii. If an input/output error or data error is reported, then the operation was unsuccessful. This may be because the device is not correctly attached so check the hardware. If the problem persists, then it is probably the data transmission settings that are incorrect. Ensure that the plotter is set to receive HPGL data and that it's transmission setting are matched to the computer settings, indicated in the "Protocols" menu.

iii. If no error is reported, the operation was successful If the sheet is visible at this point, it will be re-drawn with the new sheet size.

iv. The sheet sizes, as read from the device, can be examined using the "Sheet Size" command. The values will appear in an input box. These values should be returned without alteration to retain their current setting.

13.11 Sheet Multiples

Cases will inevitably arise where the maximum sheet size available in a device is not large enough to accommodate the size of patterns to be drawn. This is overcome by spreading the plot over several sheets. The sheet "Width" and "Length" commands are used for this purpose. The procedure is as follows:

I. Select the "Length" or "Width" command in the sheet menu. The length of the sheet refers to the horizontal direction as seen on the screen and the width applies to the vertical The number menu will automatically be activated, allowing an integer number to be selected for the multiple value in that direction.

ii. If the sheet was already visible, it will be re-drawn with the new multiple values. For example, if the selected width multiple was 2 then the sheet/s will be re-drawn still comfortably fitting the design area but with a division line in the middle showing their respective edges. Although the overall sheet appears the same size, it is in fact twice as large. This will be evident if patterns or wire frames exist in the arrangement because after the re-draw, they will appear half their original size. All that has in fact changed is the scale factor.

iii. If at this point, there is more than one sheet they will be numbered in the top left hand corner. This is to allow a selection to be made when it comes to despatching the plots of the individual sheets.

13.12 Saving Sheet Sizes

Sheet sizes and multiple values are treated as part of the configuration of the software, therefore they will be saved in the configuration file at the next opportunity (see sec. 19).

14 ARRANGING SHEET LAYOUTS

The design menu commands, known as the viewer, may have already been encountered (see sec. 9.13). The purpose of the viewer is to allow you to closely view or examine a design and any associated patterns to help ensure that it is correct. The flat patterns could also be examined using the commands available in this menu although that is not their primary purpose.

14.1 The Arrange Menu

The "Arrange" main menu holds a number of facilities for the purpose of arranging patterns and designs onto a sheet in preparation for output via the “Output” menu.

Patterns and design images can be retrieved and manipulated around the sheet, manually nesting the them together as required. Multiple instances of the same pattern can be placed on the sheet. The size of sheet used is set using the commands in the Sheet menu (see sec. 21.16) and the sheet arrangement is always scaled to be comfortably drawn in the design area.

14.1.1 Adding Patterns To The Sheet

Patterns can be retrieved and dragged into position on a sheet. They are drawn to exactly the same scale as the sheet. When despatched to an output device, the patterns will be drawn full size. The method of actually placing them on the sheet is described under the relevant menu command.

14.1.2 Adding Design Images To The Sheet

Any single object or the whole design image can be added to the sheet in a similar manner to the patterns. They can be drawn as many times and in as many positions as required and can be drawn as they appear from any of the available 2D or 3D views. The scaling of the designs is decided by their visible size on screen in relation to the sheet when the design was added. Therefore by using the zoom facility, the design can be sized as required before it is included on the sheet. The result of this is that the same design or objects of the design can be drawn in different views or different sizes together on the same sheet.

14.1.3 Saving The Sheet Arrangement

Each "GRD" file has associated with it the particular sheet arrangement that existed when the file was saved. This arrangement may be empty (i.e. containing no

patterns or design images) but it is nevertheless associated. Therefore saving a sheet arrangement is simply done using the “Save” command in the Files menu, thus saving the design and sheet together.

When a file is loaded using the “Load” command, the arrangement is loaded with it. If on the other hand, the “Append” command is used then only the design will be loaded and not the associated sheet arrangement.

If a design requires different associated arrangements or sheet sizes, then when each sheet arrangement is prepared the files should be resaved under different names (e.g. save TEST.GRD as TEST1.GRD, TEST2.GRD etc.).

14.2 Arrange Menu Commands

14.2.1 Pattern

The "Pattern" command in the “Arrange” menu is for adding patterns to a sheet arrangement. Note that at this point the sheet will not be visible and if the command has not been used before with the current design then the arrangement will be empty. The way "Pattern" behaves depends to some degree on the setting of the toggle command "Multiple" in the same menu. The following describes the use of the "Pattern" command with the various "Multiple" toggle commands:

I. Select the "Pattern" command. If the current design is empty, then selecting the command will give no response. If a design is currently loaded and the command is selected, then the object selector will be activated with the first object in the design flashing. The object selector will only allow a single object to be selected on this occasion.

ii. Select the object whose pattern is to be added to the current arrangement. The result will be as follows:

1. If the pattern has not previously been calculated, then it will automatically be calculated now using its current increment setting. Note that this should preferably be a coarse or draught increment in order to make movement and manipulation of the pattern quicker. This increment is not necessarily the increment to be used when the arrangement is despatched to an output device.

2. When the pattern has been calculated the current sheet will appear on screen in the design area as a thick white rectangle drawn over the design environment. The fact that the sheet arrangement can still be seen permits the object selector to continue to identify any object pattern to be added using the flashing facility. Irrespective of the size of the sheet it will always be scaled to comfortably fit the

design area. All patterns will be drawn to the same scale.

3. The selected pattern outline will be drawn in the top left hand corner of the sheet. If the pattern has more than one piece then only the first piece will be drawn. This allows the pieces to be nested separately in order to give a more efficient use of material. If a pattern already exists in this position the new pattern will still be easily visible because its outline is always drawn in an opposing colour to the background.

iii. Moving the mouse will now cause the pattern piece to be dragged around the sheet. Movement is limited to the sheet edges.

iv. When the pattern is positioned as required press the left-hand button. The new piece will now be fixed in the current arrangement. This is indicated by the piece being filled or coloured in.

v. At this point the setting of the "Multiple" toggle has an effect. The various settings give the following response:

Mult - This is the default setting. Its allows you to position multiples of the same pattern on the sheet without reselecting "Pattern". When the left-hand button has been pressed to accept a pattern position and the pattern is filled, then if the mouse is moved within the next half second or so another duplicate pattern will be automatically dragged away. This process is repeated until either the right-hand button is pressed or the left hand button is pressed to accept the pattern and the mouse is not moved immediately after.

One - This setting simply disables the facility to position multiple patterns as described in the previous paragraph. When the left-hand button is pressed the pattern position is accepted and operation returned to the main menu. "Pattern" must be re-selected for multiple patterns.

Grid - This setting will prevent the object selector from being activated to choose a pattern for positioning. Instead, each object pattern in the design will be calculated and dragged in turn. This allows you to produce a full set of patterns for one design without duplicating. The right-hand button can be used to reject a pattern without placing it in the arrangement. The operation will move immediately to the next pattern. When the last pattern in the design is dealt with, the operation will return to the main menu.

It should be understood that in using the techniques described in this section, that is up to you the user to ensure that no patterns on the sheet overlap. If they do then the arrangement will be accepted without being detected or reported.

14.2.2 Rotating Patterns

When a pattern is being dragged around the sheet it can be rotated to any angle. It automatically appears at an angle of zero degrees but if a numerical key is pressed or F5 is pressed an input box appear to receive a typed angle value (in degrees). When the value is returned the pattern will immediately rotate to the given angle and dragging can continue with the pattern in its new orientation. A new value can be typed at any time. This allows patterns to be nested tightly onto sheet layouts, minimizing material wastage.

14.2.3 Erase

The "Erase" command enables any pattern to be removed. Having selected erase, the vertical movement of the mouse will cause the separate filled patterns to be graphically identified one by one in a similar way to the flashing of objects with the object selector but by filling the pattern with a flowing cross-hatched effect. This allows any individual pattern to be identified and removed from the arrangement by pressing the left-hand button. The erase process is recursively repeated until the right-hand escape button is pressed. For the erased patterns to be completely removed, it will be necessary to force a redraw.

14.2.4 Multiple

The "Multiple" toggle command was discussed in sec. 14.2.1. To briefly recap, the it has three possible settings "One","Mult" and "Grid". These options allow you to either place a single pattern in the arrangement, multiples of the same pattern in the arrangement or all the patterns of a design in the current arrangement.

15 OUTPUT

The word “Output" describes the passing of data to some output device or file. The output data may be a sheet arrangement (see sec. 13) to a plotter, printer or file, or the 3D intersection data to a file usually for transfer to a CAD system. Various forms of output are possible.

If the device is correctly attached and all parameters are adequately set then the act of outputting a prepared sheet is simply a case of accessing the “Output” menu and selecting the correct output command. SHEET LIGHTNING will then do what is necessary to pass the information to the device using the set parameters.

Often the default settings will be found to operate the device satisfactorily. If the output fails however then an error will be reported and both the computer settings in SHEET LIGHTNING and the device settings will have to be checked. In this case it is essential to appreciate a few facts relating to the possible methods of data communication in order to rectify the problem.

15.1 Data Communication

The computer can communicate with a device via either a parallel port or a serial port. Parallel port communication is the simpler of the two for outputting data but has the distinct disadvantage that information cannot be read back to the computer from the device. Serial port communication is in some ways more complex in that it requires more setting up but will allow data to be both output to the device and read from the device.

15.1.1 Parallel Communication

Parallel communication transmits data on a number of pins simultaneously . This makes transmission simple in that all data bits can be transmitted in an instant therefore overcoming any confusion concerning which bits belong to which byte. The cable configurations are usually a simple one to one connection, the only complication being the need for a 25 pin to nine pin adapter on occasions depending of course on the hardware in use. The choices to be made are therefore narrowed to which port is to be used. This is discussed further in the appropriate section.

15.1.2 Serial Communication

The basic difference with serial communication is that data is only transmitted on one pin (usually pin 2). Similarly if data is to be received then it is only received on a single pin (usually pin 3). This means that only a single bit of a byte can be

transmitted at any instance which gives rise to the problem of defining which bit begins the transmission of a byte and which bit ends it. The obvious answer is for a count to be made of bits transmitted but problems can arise if for some reason the transmission gets out of synchronisation. If this occurs the interpretation of the data received will not resemble that sent. To combat this problem certain data checking feature can be set to halt and report on data errors. In SHEET LIGHTNING these features are set using the toggle features in the "Protocols" menu which is accessed through a selector in the "Plotter" menu.

15.2 Devices

SHEET LIGHTNING offers the possibility of transmitting output to either a Plotter or a Printer. The following comments may be helpful regarding the capabilities of SHEET LIGHTNING with these devices.

15.2.1 Plotter

Plotters generally have the facility to communicate via a serial port, a parallel port or both. In any case they must be compatible with HPGL which is a graphics communication language using ASCII text symbols. Many plotters have the facility to communicate in other languages in which case it is essential that the plotter is set to HPGL mode before output commences. If the serial port is used parameters can be read from the plotter such as its hard clips dimensions (maximum size), its resolution (increments per mm) and position of origin. If the parallel port is used then these parameters should be set manually from the commands in the "PLOTTER OUTPUT" menu. It they are not set then the defaults will be assumed but problems may arise such as the incorrect positioning of the plot on the drawing sheet. The default is an A0 sheet with the origin at the lower left hand corner.

15.2.2 Printer

Printer output is only possible via parallel ports and requires no parameters for output. The origin is always assumed at the top left hand corner of the selected sheet (the sheet is selected before output commences). The X dimension is horizontally along the sheet as seen on screen and the Y dimension is vertically down from the top left hand corner of the sheet as seen on screen.

15.3 Setting Up For Serial Devices

There are possibly two main areas of controversy in setting up communications to a serial device. They are cabling and handshaking.

15.3.1 Cabling

Cable configuration is very important for successful communication. There are many different cable configurations in use for serial port communication although SHEET LIGHTNING is very forgiving in that it offer three handshake modes and usually requires just a few key pin connections to be correct. The most important thing to check is that lines 2 and 3 cross (i.e. the transmit data pins TD and the receive data pins RD respectively). Beware that you are not using a "Null Modem" cable designed as the name suggest for modem communications where the cable will be a simple one to one (i.e. pin 3 to pin 3, pin 2 to pin 2 etc.). The individual requirements for each handshake mode are mentioned in their relevant section.

15.3.2 Handshake

The "Handshake" toggle in the “PROTOCOLS” menu (of the “PLOTTER OUTPUT” menu) defines the mode of serial port communication and determines the way in which the computer should communicate with the plotter in order to maintain a steady flow of data without overflowing the plotter's input buffer. There are three options:- Soft, HWire and XoNF. The required setting depends on the plotter and whether it will support the handshake mode. In many cases the plotter will adequately support more than one handshake mode in which case the choice should not matter. The various modes are described as follows:

I. Soft

Software handshaking is the simplest method of handshaking. Its operation depends on the ability of the plotter to respond to an ESC.B command by outputting the available buffer space. To check the feature consult the plotter manual. In this case only pins 2 and 3 are of real importance as mentioned above. The disadvantage with software handshaking is that the flow of data may not be steady but faltering. For this reason software handshaking should be the last resort.

ii. HWire

Hardwire handshaking uses other pins to regulate data flow therefore the plotter must be capable of responding to the hard wire signals and the cable must be of a suitable configuration on the additional pins. To be precise the plotter should have the ability to change the DTR pin (Data Terminal Ready pin 20) from High to Low or vice versa in accordance with the remaining buffer capacity. The cable configuration required to achieve the operation is usually covered in the plotter manual. The most important feature is that the DTR pin should cross with the CTS pin (Clear To Send Pin 5) so that when the DTR is set by the device CTS is detected by the computer in order to suspend data flow.

iii. XoNF

The XON/XOFF handshake is another software controlled method of regulating data flow. In this case it depends upon the ability of the plotter to be set to a handshake mode whereby it will output a specified character when the remaining buffer capacity goes below a certain level and it will output some other specified character when the buffer empties to a certain level. Again in this case the important pins are 2 and 3 as discussed earlier.

This method of handshaking is usually the most desirable because it gives a steady flow of data with a minimum of cable configuration problems. XoNF is therefore the default setting.

15.4 Protocol Settings

The serial port protocol settings available in SHEET LIGHTNING are accessed through the "PROTOCOLS" menu which is a submenu of the "PLOTTER OUTPUT" menu. The settings defined are as follows:

15.4.1 Baud Rate

The baud rate is the rate at which data is to be passed from the computer to the device. If the baud rate setting of the computer is not matched to that of the plotter then an error will occur. The available settings are:

110,150,300,600,1200,2400,4800 or 9600

15.4.2 Parity

Parity is, in effect, a form of data error checking used to check data transmission. There are three possible settings of EVEN, ODD or NONE.

15.4.3 Databits

The databits are the number of consecutive bits that can be treated as data during transmission. This value can be set to either 7 or 8.

15.4.4 Stopbits

The stopbit is the bit that follows the last bit of a byte in serial data transmission and is used as a signal to the receiver that all of the byte has been transmitted. It's value can be set to either 1 or 2.

15.4.5 The Defaults

The default settings are as follows:

Baud Rate: 9600
Parity : NONE
Databits : 7
Stopbits : 1

These values are changed using the toggles available in the data menu.

15.5 Plotter Ports

Plotter may receive serial or parallel data therefore one of the serial ports COM1 or COM2 or the parallel ports LPT1, LPT2 or LPT3 can be used. The port is set using the toggle available in the protocols menu.

Note that by default COM1 is normally configured by the operating system to be the serial port on the computer therefore COM1 is the default setting in SHEET LIGHTNING.

15.6 Printer Ports

Printers can only be sent parallel port data therefore the possible device settings are LPT1, LPT2 and LPT3. The device is set using the command in the "PRINTER OUTPUT" menu.

It may be worth noting here that if the plotter output facilities are used via a parallel port and a printer is attached to that port then the HPGL ASCII text will be passed to the printer. Note that in this case the "Set-Up code will not be passed.

15.7 Printer Set-Up Code

When the printer output facilities are used the printer is first sent a control code which can be set by with the "Setup Code" command in the "PRINTER OUTPUT|CONTROL" menu. The characters are typed into a input box as normal but on this occasion the escape key will not escape the operation. The code must be returned with the left hand button to remove the input box. The reason for this is simply that the escape code is often used in control codes, therefore pressing the right hand button will insert the escape code into the string.

A further addition is the facility to type characters not normally available via the keyboard keys. These can be inserted into the set-up code by holding down the ALT-Key and typing their ASCII value (consult your computer manual) which will be anything from 000 to 255. Note that three decimal digits must be typed to specify a character. For example hold down the ALT-Key and type 074. This is the ASCII code for the letter "J" which will be entered in

the set-up code string. Of course this letter could be entered by typing "J" directly but there are many other characters without a specific key. Note also that when a some characters are entered only a space will appear. This is because not all characters are available in the graphics fonts and it must therefore be represented by a space. This demands care when typing because the results are not always visible, If there is any doubt re-type the code completely.

The current setup code is saved in a configuration file so it is not essential to reset the code each time SHEET LIGHTNING is used if a configuration file has been saved.

15.8 Languages/Forms Of Output

There are several forms of output available from SHEET LIGHTNING.

15.8.1 HPGL

This form of output is a graphics language generally used for plotter communication. It consist of ASCII text consisting mainly of two-character symbols and numerical values.

15.8.2 DXF

DXF format has become standard for passing data between CAD systems. It consists of ASCII text but in a very different format to HPGL. To give a brief description it consists of pair of lines, the first giving a numerical code to define what is given in the second which may be numerical or text and may represent many different things.

Note: ASCII text is readable and therefore either of the above data formats can be read, although interpretation can be difficult and requires a knowledge of the symbols used.

Note: also that one or both of the above formats are supported by word processors. Extensive graphics images could therefore be transferred to a suitable word processor, perhaps for reporting purposes. Drawings in this manual such as that included on the title page was created in this way.

15.8.3 ASCII TEXT

This refers to another form of ASCII text format which is readable. The "print" options in the despatch menu outputs this form of data. In this way the it is possible to obtain the numerical data of patterns which can be plotted by hand. the facility may be useful in the absence of the necessary hardware or in cases where the

pattern is extremely large and hand plotting is preferred to a partitioned plot.

15.9 Output to a Plotter

To output data directly to a device the HPGL language is used, therefore it must be checked that the device is able to receive data in HPGL. This has become the virtual standard in plotter languages although many plotters offer a choice. If this is the case, then it must be ensured that the plotter is set to HPGL mode. The procedure is as follows:

I. Choose the "Plot Sheet" command in the “PLOTTER OUTPUT” menu. In response, an attempt will be made to initialise the plotter for output and if the serial port is being used the computer will read the relevant hard clip limits and plotter settings. Of course the plotter must be capable of outputting this information. If the device is not present, then an input/output error warning will be issued. If this occurs, the process will continue but no attempt will be made to pass data to a device therefore the patterns are simply recalculated and drawn to screen.

ii. If the Pre-draw command in the printer menu is enabled the sheet arrangement will be completely re-drawn with the background erased.

If the "Hard Clips" command in the "Plotter" menu is set to "Read" the plotter will temporarily read in the maximum physical limits of the plotter. If this is done the sheet pre-draw will show the sheet dimensions according the plotter hard clip limits.

If the hard clip limits prove to be different to the original sheet settings then the pattern arrangement may not fit within the boundaries of the sheet in the same way. The options in this situation are to abort the operation and take one of the following steps:

1. If possible, increase the hard clip limits of the plotter (if the hardware permits).

2. Increase the sheet multiple values found in the Sheet menu. This facility allows multiple sheets to be arranged together in order to give a greater plotting area (see sec. 13.11). Note that if the sheet dimensions read from the plotter are different to the original settings, then the sheet divisions of the plot will have changed.

3. Alternatively re-design the sheet arrangement using the new plotting area according to the hard clip limits of the plotter. It is worth mentioning here that it is a good idea to use the "Plotter" command in the Sheet menu before designing a sheet arrangement. This command simply reads the hard clip limits and re-sets the sheet size dimensions accordingly. Of course, it is only possible to use the "plotter" command with success if a plotter is attached,

otherwise an error will be reported and the sheet size will retain the previous values.

The plotter reading facility is useful for obtaining the maximum size of plotting area available. It does have the disadvantage however that the sheet size may change for each despatch operation. The function must therefore be disabled for partitioned plots. In this case it is more usual to permanently read in the size of the plotting area from the "Plotter" command in the "Sheet" menu and then save the configuration (ALT-F2). The "Hard Clips" command can then be toggled to "Sheet" thus disabling the automatic plotter reading.

15.10 Output To A Printer

The process of passing a current sheet arrangement to a printer is accessed via the "Print Sheet Image" command in the "PRINTER OUTPUT" menu. The procedure is almost identical to that given above except that no attempt to read any parameters is made and therefore no warnings are issued. Instead X and Y text values are passed to the printer via the port set in the "Printer" menu after the printer has been initialised with the "Set-Up" code as described earlier. The X values correspond to the horizontal direction and Y the vertical direction as appears on screen with the origin at the top of the left hand corner of the selected sheet.

Note that the whole sheet arrangement is output using this origin irrespective of whether the plot comes completely within the selected sheet area.

The facility is intended for use where it is considered more convenient to plot a pattern manually perhaps because it is very large or because of the lack of available hardware.

15.11 Despatching A DXF Sheet File

Data interchange or DXF files are used to store graphical data in a standard format for transporting between CAD systems. The procedure for storing such a file is identical to that given above except that the "DXF File" command in the "Output" menu is used and a DXF extension replaces the PLT extension in the file name. Note that the file size is significantly larger than that of a similar PLT file.

15.12 Despatching A HPGL Sheet File

HPGL files can be passed a plotter or device by third party software, or the graphical data can be loaded into a word processor. To output the current sheet arrangement select the "HPGL Sheet" command in the "Output" menu. the complete arrangement is despatched with recalculation of data where necessary.

15.13 Despatching A DXF Design File

The 3D wire frame or surface (face) data can be despatched to a DXF file using the "DXF 3D-Design" command in the "FILES OUTPUT" menu. No recalculation will occur so the only data passed to the file is that of objects in the design that have been cut and welded. A single object can therefore be output if it is the only cut object in the design. The file is given an extension HGL having the same file name as the original GRD file from which it was produced. Other extensions can be specifically specified as with all other file name input.

15.14 Recalculation

When either a plot file is being created or data is being despatched directly to a plotter, re-calculation of the data may occur. This allows the you to work with a "draft" resolution using a coarse calculation setting and therefore a relatively inaccurate plot, but guarantee the output to be an accurate plot.

The advantage is that while a sheet arrangement is being prepared, the patterns and images can be calculated and manipulated quickly using the coarse setting but the despatched data will still have the desired level of accuracy without changing any of the settings.

The following points indicate when and where re-calculation occurs:

(a) **Patterns** are always re-calculated before despatch, using the despatch increment setting found in the "increment" command of the despatch menu. This setting is in degrees and is examined and reset using a normal input box. Note that re-calculation can be prevented by setting the value in the "increment" command to 0 degrees. In this case the existing pattern data will be used irrespective of the increment that was used. If the pattern is not yet calculated then it will be calculated using its own increment setting and not that of the despatch menu increment command.

(b) **Wire Frames** are only re-calculated if they are currently in their uncut or outline state. This may be because they have not been cut and welded in the design environment or because their cut and weld data has been automatically disposed of to make room for other data. Such disposal is more likely to occur if patterns have previously been re-calculated with a fine increment in order to make room for the data.

If re-calculation of a wire frame does occur it will be at the individual increment setting of the object and not at the despatch increment. The despatch increment only applies to patterns. To reset individual increments, use the "increment" command in the Compute menu (see sec. 9.7).

Note that automatic disposal also occurs if there is a shortage of memory during normal calculation in a cut and weld or pattern calculation procedure. When it occurs, the word

"Disposing" appears at the bottom of the screen accompanied by a figure giving the current memory availability.

15.15 DXF File Format

The commands for producing DXF files has been covered earlier in this section. It remains to give a little information on the parameter and the way in which the data is written.

(a) Extents

The extents of the DXF drawing correspond to the current size of the multiple sheets as they appear on screen. It is therefore best to ensure that the sheet multiples are large enough to contain the whole of the plot file drawing, otherwise the drawing will go over its extents.

(b) Layers

The following layers are defined:

Layer 0: Contains all data that makes up the outline of a pattern.

Layer 1: Contains all data that makes up a holes in a pattern.

Layer 2: Contains all radial and fold lines on the flat pattern.

Layer 3: Contains all wire frame lines which make up drawings on the sheet other than pattern drawings. Note that these drawings are the 2D data as used by a plotter to produce a hard copy of the image, not the actual 3D data.

With the data divided in this way it is easy to see that should the data be used for cutting purposes then only layer 0 and 1 represents cutting lines. Obviously the holes should be cut first so layer 1 would be processed first, then layer 0. Layer 2 contains other information that could be drawn onto the sheet before cutting commences to assist with rolling and folding and layer 3 would be ignored for cutting purposes.

15.16 Direction Of Cut

The only remaining information that is needed to utilise the data produced by a cutting machine is the direction of cut in order for the cutter tool to be suitably offset from the given line.

15.16.1 Outside Data

If the pattern data is produced for the outside of an object (i.e. when the "In/Outside" command in the "Options" menu is set to "Out" then all data will conform to the following convention:

I. Perimeter - The perimeter of a pattern in cut in an anticlockwise direction.

ii. Holes - Under the same convention holes will always be cut out in a clockwise direction.

15.16.2 Inside Data

Pattern data produced for the inside of the pattern will follow the opposite conversion to the outside (i.e. clockwise for perimeters and anticlockwise for holes).

15.17 Decimals Places

The "Decimals" command in the "Printer" menu is used to set the number of decimal places used in printed output.

16 COLOURING THE DESIGN

The “COLOUR” menu provides commands for changing the colour of objects in the design, both in their outline or whole state and in their cut state. Changing colours can greatly enhance the visual clarity of the design. On the other hand it may simply be a matter of preference.

16.1 The Colour Menu

The colour menu is a special menu which appears at the appropriate time down the right hand side of the design area. It is used to alter the foreground and/or background colours of design objects.

The menu consists of a column of boxes each with a separate colour and a clear column or gap down either side where the cursors are moved. The gap to the left and the associated cursor is for altering foreground colours. The gap to the right is for background colours. Hence the headers F and B.

The cursors are in the form of flashing arrows which point to the colours in the column. They are controlled by vertical movement of the mouse once the colour menu is drawn.

Which cursor is moved is determined by the most recent sideways movement of the mouse. If the mouse veers to the left, the foreground cursor becomes active, if to the right then the background is active.

When the colour menu is used, both cursors are not always active. It depends on the reason for which the menu was drawn. If only one side of the menu is active, then the cursor is unaffected by which way the mouse "veers".

16.2 Solid Colour

Each object has associated with it both a foreground and a background colour. These colours are set by default but can be changed for any individual object using the "Solid" command in the Recolour menu. The procedure is as follows:

I. Select the "Solid" command. The object selector becomes active to identify which object's colours are to be changed.

ii. Select the item/object. The colour menu is drawn with both foreground and background active. At the same time the view is cleared and the selected object drawn solid with its current colours in its current position on screen. Note that the colour menu cannot be overdrawn so it may be necessary to ensure that the object is in a view with a good central visible position before attempting to change the

colours. If the object is not visible, this will not inhibit the process but the results will not be immediately seen.

iii. The flashing cursor on the colour menu indicates which of the two is active. Move the cursor with the mouse to the desired colour and bring to rest for a brief moment. If no further movement is detected in that moment, the solid object is redrawn with its new colour. This is repeated each time the cursor is moved.

iv. When the object is drawn to screen with the desired colours, press the left-hand button or <Enter> key to accept. Operation returns to the main menu.

It may be useful to know that it is possible to select the whole design for colouring using the solid command. This will not allow the foreground colours to be altered, therefore only the background cursor is active. If a background colour is selected as described in the procedure, the whole design is drawn solid with the uniform background colour.

The restrictions of drawing more than one solid object simultaneously are discussed in the section covering the "Viewer" (overdrawing etc.), therefore this facility may not be appropriate to all designs.

16.3 Background

This command is used specifically to set a uniform background colour. The procedure is similar to the previous. Only the background cursor is active, the main difference being that in this case no solid drawing takes place. The difference only becomes obvious when using the viewer is used.

16.4 Outline

All objects in their outline state normally have the same colour because different colours in this state can hinder visual clarity. The "Outline" command in the Recolour menu can be used to reset this uniform colour or indeed to select the individual colouring if preferred. The procedure is similar to that of the solid state but this time with only the foreground cursor active indicating the outline colour. The objects in the design are overdrawn (not redrawn) with the newly selected colour after a brief pause. If the current background colour is selected this is taken to mean the objects are to be drawn in their individual colours. The effect is immediately obvious after overdrawing.

16.5 Wire

Wire frames are normally drawn in their individual colours and this command is therefore by default set to "Colr". If the toggle is changed to its other option "Outln" then the cut object

wire frames assume the same colour as the outline objects.

16.6 Transparent

It is sometimes useful to be able to see and examine the inside cutting lines of a design. The "Transparent" command is a toggle with possible settings of ON and OFF. With transparent ON the difference is seen when drawing objects solid. The rear half of the object is drawn solid as before but the front half remains drawn in wire. This feature can also help to overcome the overdrawing problems when drawing multiple solid objects thereby making a good view of a solid built design possible.

17 OTHER FEATURES

A number of other feature of SHEET LIGHTNING require some discussion. Most of these are tucked away in the "Options" menu.

17.1 Sensitivity

Because the delicacy of touch and the familiarity with using a mouse varies from person to person the sensitivity command is included in the options menu. This is simply available to increase or reduce the mouse sensitivity to requirement Selecting the command activates the number menu which will allow a value to be selected. The lower the value the greater the sensitivity of the mouse. The default value is 4 and the number menu will always start at its current setting. If the sensitivity is to be reduced then simply choose a higher value.

17.2 Sound

Sound can assist in certain places to keep track of the operation of the software. On the other hand it can be annoying and cause unnecessary delay, therefore the ON/OFF sound options are included in the Sound menu accessed through the options menu. A short beep or high pitched blip is heard at the relevant times when sound in switched on.

There are four options:

Selection: indicates the incremental changes when making a selection such as when using the object selector. Default is OFF.

Error: accompanies an error report on the error line. Default is ON.

Abort: indicated an operation has been manually aborted such as a cut and weld calculation or a match operation. Default is ON.

Match: indicates the completion of each individual match operation in a chain matching situation. Default is OFF.

18 EXAMPLES OF DESIGNS

In this section the step by step procedure to create a number of different designs is covered as an introduction to using the design environment. The first few examples are very basic but they will become more complex and will introduce the use of the design facilities already discussed.

18.1 An Equal 90 Degree Pipe Tee

The equal 90 degree (or perpendicular) tee is a common example of a sheet metal product. The parameters to be decided are the diameter of the pipes and the distances from the point at which the C/Ls tee together to the ends of the pipe.

The design can be managed easily from either the 2D or 3D environments. In this case, we will keep to the 3D environment because the visualisation is usually better. We will assume that we are working in a room of the default size and that the cursor is free to move in a plane which is flat to the floor as at start-up. The procedure for design is as follows:

(a) To begin with we must set the first object C/L in position in the design. This C/L can be set anywhere provided the cursor does not have to go beyond the boundary walls to complete the design. For the sake of neatness, we will place the first C/L in line will the X-axis.

I. Move the cursor into the room and choose a convenient point on the tiled surface. This should be near to the corner of the room but far enough away to keep the whole design within the boundaries. Press the left-hand button.

ii. Stretch the C/L out by moving the cursor to indicate that the first point was accepted and a C/L is now active. Move sharply to the left to go into the design menu and select the axes alignment command.

iii. Move back into the design area and position the cursor so that the alignment of the C/L is close to the X-direction and press the left-hand button. The cursor will now be restricted to slide in the X-direction only this keeping the new C/L in line with the axis.

iv. Set the length of the new C/L by watching the length dimension at the bottom of the screen (e.g. 500mm). If incremental movement is too coarse either use the numerical input facility (see sec. 3.7) or zoom into the area using the zoom facility (see sec. 6.8.8) to give finer incremental movement. Press the left hand button to fix the length and position of the C/L in the design.

(b) When the final position of the C/L was set, it will have changed to become visible with its default dimensions. This is a convenient time to **set the desired diameter**.

I. Select the circular cross-section operator from the design menu. The sniffer will now be active.

ii. Go back to the design area, move close to the object to be dimensioned and press the left-hand button. The cursor will now be sliding on its C/L.

iii. Position the cursor at a midway position and press again. The cursor will disappear and the object can be grown to the required size by moving the mouse vertically.

iv. Set the diameter to the required dimension (e.g.200mm) and press to accept.

© Position the first end of the second object at the point on the C/L where the tee is to be taken off.

I. Select the slide facility in the design menu. The object selector will now become active.

ii. Use the object selector to select the object, in this case the only one available. The cursor will now be restricted to sliding along the C/L when moved back into the design area.

iii. The length dimension at the top of the design area should now be used to position the cursor at a set distance from one end of the C/L. To make sure that the dimension is being measured from the first end, slide the cursor up to that end. Sliding it back will now cause the length to be measured from that end. Position the cursor at the point where the tee is to be taken off and press the left-hand button. A new C/L will become active.

(d) Position the second end of the C/L to form the 90 degree bend and the required length.

In this case, because the angle is 90 degrees and the first object was positioned square with the axes, we have a choice of two ways of completing the design. The first way would be to align the new C/L with the other axis (Y-axis) as we did before with the first C/L. The second way, which in this case we will use, is by the angle setting facility in the design menu.

I. Select angle setter in the design menu. The number menu will become active.

ii. Select the value of 90 from the number menu.

iii. Move back to the design area and over to the side of the first object C/L that the 90 degree tee is to be set. Press the left hand button. The cursor will now be moved and restricted to the 90 degree line.

iv. Position the cursor to set the required object length as before, using the active dimensions at the bottom of the screen. Press the left-hand button to accept. The new object will be matched to the other C/L immediately to form an equal tee. The design is now complete. It only remains to perform the necessary Cut & Weld operations or to view the design and patterns as required.

v. Save the design under a new name using the "Save As" command in the "Files" menu.

18.2 Altering To An Equal 60 Degree Pipe Tee

The next exercise is alter a design. We will take the previous example and alter it to a 60 degree equal tee.

(a) Re-activate the second object in order to re-position the free end.

I. Select the re-activate command in the design menu. The sniffer will be highlighted.

ii. Select the second object with the sniffer.

iii. Slide to the free end and press the button. The object will now be re-activated in the same way as when the new C/L was first activated, although its length will be frozen.

iv. If the length is to be changed, turn the freeze length highlight off in the design menu. If movement is cumbersome with the whole object, then select the outline command in the design menu to reclaim the C/L.

(b) Place the C/L in its new position at 60 degrees.

I. Select the angle setter command in the design menu. The number menu will become active as before.

ii. Select the value 60 from the number menu.

iii. Return to the design area and place the cursor so that the reactivated C/L is inclined towards the end of the first object to which the new angle is to be set. Press the left-hand button. The cursor will again be moved and restricted to slide along a line inclined at 60 degrees to the nearest end.

iv. Position the cursor to set the length as before and press the left-hand button to accept. The design is now a 60 degree equal tee.

v. Save the file under a new name.

18.3 An Unequal Tee

The following gives an example of how to change the current design to an unequal and then change it back to equal.

(a) Turn Chain Matching off

To do this go to the options menu and set the chain matching toggle to off. Its default setting is on.

(b) Re-dimension the Tee-off Pipe

I. Select the circular cross-section command in the design menu.

ii. Select the tee-off object using the sniffer.

iii. Position the cursor mid-way to allow the whole object to be grown and press the left-hand button.

iv. Grow the object to the new size. Note that the new diameter of the object should be no greater than the object that it is teeing onto. If it is then parts of the newly dimensioned object will be open-ended.

© Return To An Equal Tee

I. Switch Chain Matching in the "Options" menu back on.

ii. Select the circular cross-section again and use the sniffer to select either object.

iii. Grow the required diameter and press to accept. The diameter of the other object will be matched to it thus reclaiming the equal tee.

18.4 Adding A 45 Degree Tee At The Same Point

The third object should be added in the following way to ensure that it tees onto the same point as the second object.

(a) Position the first point.

I. Select the slide command.

ii. Using the object selector select the second object which tees on to the first.

iii. Go back to the design area and slide the cursor to end of the object that forms a tee with the first. Press the button to make this point the first end of the third object. This will automatically relate the object to the second in the form of a bend and to the first in the form of a tee.

(b) Position the second point.

The 45 degree inclination will again be established using the angle setter. The object will in this case be angled in the opposite direction to the 60 degree tee but on the same side.

I. As before select the angle setter.

ii. Select the 45 degree value from the number finder. On this occasion, because the new object is related to more than one other the object selector will be automatically activated to request which object the angle is to be related. Only related objects can be selected.

iii. Select the first object using the object selector.

iv. Move back into the design area and incline the new object C/L away from the second object but on the same side of the first. Press the button and the cursor will be restricted to a 45 degree inclination.

v. Set the length using the cursor and press to accept. The three piece pipe equal tee design is now complete.

19 THE CONFIGURATION FILE

The configuration of the software refers to the setting of certain parameters within the design environment and pop-up menu system. It would be a lengthy process to reset all the toggle commands and other parameters each time the program is run, therefore the facility to save a configuration file exists.

19.1 Saving a Configuration File

There are two ways in which the configuration of the software can be saved. Both of these require the "Config" command in the files menu to be set to "ON". The effect is that the current configuration of the software is saved to a file named "SHEET.CFG" is the current directory. If the file already exists then it will be over-written, if not it will be created.

(a) Using "Quit" in the Files menu to exit the program SHEET LIGHTNING will automatically cause the configuration to be saved. The hotkey ALT-X on the other hand provides a shortcut to quit but will not save the configuration. Therefore always use ALT-X to retain the previous configuration.

(b) The hotkey ALT-F2 will always save the current configuration

When the configuration is being saved the word "Configuration" will appear on the bottom line of the screen.

19.2 Loading a Configuration

If a configuration file exists in the current directory when SHEET LIGHTNING is executed, it will automatically be loaded. If no file exists, the configuration will assume its default settings.

19.3 The Configuration Parameters

In addition to the pop-up menu toggle parameters, the following additional parameters are saved in a configuration file:

(a) **Sheet Size -** The current sheet size.

(b) **Sheet multiples -** The current sheet length and width multiples.

(c) **View -** The current view and if in 3D the last 2D view.

(d) **Outline Colour -** The current outline colour setting as set by the command in the Re-colour menu.

(e) **The current file name -** The current name file is saved along with its path or directory. At start up this file will be automatically loaded into the design environment, therefore design can continue exactly where it left off.

(f) **Load/Append/Save As input -** The last input given in response to these File menu commands is saved.

20 HOTKEY REFERENCE

A hotkey is a key which can be pressed to perform a particular function of the software as a short cut to using the mouse. Some hotkeys are only available at certain times or in a certain context. This section offers a reference to the hotkeys and the context in which they can be used.

20.1 General Hotkeys:

F6 - Direct return from the main menu to the design area.

F2 - Automatically saves the current file.

F3 - Moves directly to the "Load" command input box in the files menu.

F4 - Moves directly to the "Append" command input box in the files menu.

F5 - Initiates logical numeric input with the current value in the input box.

ALT-F2 : This will cause the current configuration to be saved in the configuration file. If the file does not exist then it will be created in the current directory.

ALT-A : Moves directly to the "PLT Access" menu.

ALT-C : Moves directly to the "Compute" menu.

ALT-D : Moves directly to the "Despatch" menu.

ALT-E : Moves directly to the "Efficiency" menu.

ALT-F : Moves directly to the "Files" menu.

ALT-G : Moves directly to the "Grid" menu.

ALT-H : Moves directly to the "Sheet" menu.

ALT-I : Moves directly to the "Printer" menu.

ALT-L : Moves directly to the "Plotter" menu.

ALT-M : Moves directly to the "Match" menu.

ALT-N : Moves directly to the "Sound" menu.

ALT-O : Moves directly to the "Options" menu.

ALT-P : Moves directly to the "Pattern" menu.

ALT-Q : Moves directly to the "Data" menu.

ALT-R : Moves directly to the "Recolour" menu.

ALT-S : Moves directly to the "Scene" menu.

ALT-T : Moves directly to the "Protocols" menu.

ALT-U : Moves directly to the "Output" menu.

ALT-X : Quits SHEET LIGHTNING without saving the configuration.

20.2 Design Area

F6 - Toggles the datum end an object when sliding is set.

F7 - Short cut to "cut and weld" with the object selector automatically activated for selecting individual objects.

F8 - Short cut to "cut and weld" the whole design without passing into the main or pop-up menus.

F9 - When an object is active this key will switch individual dimensions to relative dimensions (see sec. 7.2.17).

F10- Moves directly from the design area or the design menu to the main menu.

HOME : If in the design area the cursor position will be moved to the origin or corner of the room (i.e. X,Y,Z=0).

20.3 The Viewer

F7 : Will prevent the previous view being erased when selecting another object to view thus allowing a design to be manually built together in solid or wire form (see sec. 9.13).

F8 : Will draw all objects together in solid form. Note that specific arrangement is required to prevent overdrawing (see sec. 9.13).

20.4 Input Box

Del> : Deletes the character at the cursor and moves text forward up to the end of the line.

<Del : Deletes the character to the left of the cursor moving the remaining text forward.

End : Moves the cursor to the end of the text.

Home : Moves the cursor to the beginning of the text.

21 QUICK REFERENCE

The purpose of this section is to give the brief description of each command or facility in the software for quick reference or reminder.

21.1 Design menu

This is the area permanently visible to the left of the screen. Fig.2 identifies the commands by name, their purpose is briefly described below:

21.1.1 3D-View Mode

Appears in the design menu as “V3D”. If in a 2D design environment, then change to 3D. If already in 3D, then returns the cursor to the design area at the last room position before leaving.

21.1.2 2D-View Mode

Appears in the design menu as “V2D”. If in the 3D view change to the last 2D view. If already in the 2D view it changes to the next 2D view place. Repeated presses cycle round the three 2D view planes.

21.1.3 XYZ Stabilisers

Commands appear in the design menu as “X”, “Y” and “Z”. The stabilizers bind or free movement in the X,Y or Z direction. The axes availabe for free movement are highlighted..

21.1.4 Slide/Tee Selector

Appears as the “SLIDE” command and is used to restrict cursor movement in the design area along the C/L of an object for various purposes but particularly for forming object relationships. The object on which sliding occurs is chosen from the object selector which is activated automatically. The slide state can also be reached using the “SNIFF” facility.

21.1.5 Bend Mode Selector

Causes a new object to automatically become active when a C/L is positioned in the room. The new object is related to the previous as a bend. Therefore a string of objects forming a multiple bend can be formed easily.

21.1.6 Sniffer

Appears in the design menu as “SNIFF”. The sniffer allows any object in the design to be selected by identifying it using the cursor directly from the design area. The cursor snaps to the nearest C/L.

21.1.7 Object Selector

Does not appear as a command but is usually activated automatically in conjunction with other commands to select a particular object in the design. The selector opens up into a ladder of numbers and/or names of the objects. The current selection bar is highlighted and the corresponding object will “flash” in the design. The first selector may show the word "ALL" which allows you to select the whole design..

21.1.8 Cross-Section Selector

Selecting the “SECTION” command replaces the design menu tools page with a selection of cross section. Mouse movement is now taken over by the cross section selector. Choose a cross section by moving the highlight bar to the required command using the up and down movement of the mouse or the up and down arrow keys. The sniffer is now automatically activated to allow the object and the relevant end of the object to be selected for the sizing operation to commence. The cross section tools are as follows:

(a) **Circular Cross-Section Selector:** “CIRC” is used to set the cross section of an object to form a cylinder or cone. Both ends become circular.

(b) **Regular Cross-Section Selector:** “REG” is used to set the cross section to some regular sided shape, for example square or hexagon. Straight or tapered regular sided sections can be formed. Both ends become the same regular shape.

(c) **Rectangular Cross-Section Selector:** “RECT” is used to set a rectangular cross section which can be of any width and breadth. Both ends become rectangular although the rectangle can be different in width and height at each end of the object thereby forming a rectangular adapter.

(d) **Adapter:** “ADAPT” can be used to form an object with attributes of width, height and diameter. Both ends are set with this command therefore a total of six values are required. An end with all non-zero dimensions will appear as a rectangle or square with rounded corners. It both height and width are zero it will be a circle. It just height or width is zero it will be an oval. If just the diameter is zero then it will be a true square or rectangle. The ends may be set to be of completely different shape. Variation at both ends simultaneously is not possible in this case.

(e) **Height-Width-Diameter:** “ADAPTEND” is similar in operation to the adapter except that only one end is set. Therefore just three values of height,

width and diameter are required. Both ends may be adjusted simultaneously by selecting a mid line position.

(f) **Proportional:** “PROPORT” allows the current shape of the object to be expanded whilst keeping the same proportions of width height and diameter.

(g) **Oval:** “OVAL” forces height of object, or object end to zero. Therefore the shape is oval and only two dimensions must be set; diameter and width.

(h) **Root Radius Rectangle:** “ROOT” forms a rectangle having a set root radius at the corners/folds. The radius is set using the "Root Radius" command in the “GRID" menu. Both ends of the object are forced to have this root radius irrespective of which end is being modified.

(I) **Diameter:** “DIAM” allows the diameter of the object to be adjusted with no alteration to width and height.

(j) **Height:** “HEIGHT” allows the height of the object is adjusted with no alteration to diameter or width.

(k) **Width:** “WIDTH” allows the width of the object is adjusted with no alteration to diameter or height.

(l) **Root:** “ROOTEND” allows one end of an object to be forced to have a root radius with no alteration to the other end. Both width and height of the selected end are set.

21.1.9 Seam Position

If the object to be operated on is of circular cross section, then the “SEAM” command will allow the seam position to be set. If, on the other hand, the object is of any other type or cross section then this will allow the whole object to be rotated therefore keeping the seam in the centre of one of the sides of the object.

21.1.10 Drag

The “DRAG” command allows any individual object to be moved around the room. Its orientation to the axes is not changed. If any objects are related to the dragged object, then they will be moved to a similar relative position when the new position is set (i.e. they will "catch up").

21.1.11 Offset

The “OFFSET” command allows an object or C/L to be offset from another. This will

usually be done in the case of a junction or tee relationship that requires a smaller object to tee onto a larger at an offset or non-central position. The “Displace Mode” options of “Offset” or “Displace” are prompted for in response to this command. The “Offset” option allows the offsetting object to be dragged in a perpendicular line to the two objects. The “Displace” option allows the object to be dragged anywhere, in both cases he original relationship between the offset and related object is preserved.

21.1.12 Length Freeze

The “LENGTH” command simply allows the length of an *active* object to be fixed or frozen at some value. Further cursor movement will be restricted to maintain this length unless the cursor is sliding along the C/L of another object. Typing numercial keys or pressing F5 will allow the object length to be set. If the length is frozen the object will remain set at this length. If the command is selected whilst the active object is frozen the length will be liberated, allowing the object centre line to be stretched out by the cursor.

21.1.13 Deleting objects

“DELETE” allows any range of objects to be deleted from the design. The object selector is activated twice. The first object selected is the first of the range of objects for deletion, the second is the last in the range. All objects between the two selected in the order in which they appear in the design are deleted.

21.1.14 Angle Setter

The “ANGLE” command allows the angle between the active object and any other related object to be set so that cursor movement is restricted to that angle. The angle is set using the number finder (or typed when the number menu is active). If the active object is related to more than one other a selection is automatically initiated to identify the object the angle is to be set relative to.

21.1.15 Perpendicular

“PERP” is identical to the angle setter except that the required angle is always 90 degrees. Therefore no angle selection is required.

21.1.16 Deactivate

“DEACT” allows any new active object (or centre line) to be deactivated. If the object has been reactivated (and therefore previously placed in the design) it must be deactivated by placing it.

21.1.17 Construction

The “CONSTRUCT” is a toggle command that is active when highlighted. If it is set then any new objects placed on screen will remain in a construction line or C/L form.

21.1.18 Number Finder

The number finder is a tool which is automatically used in conjunction with other commands to select a particular value without using the keyboard. When it is active you only have a limited selection of values to choose from. If these values do not include the exact value you require then type in the value while the number menu is active, or press F5 to bring up the input box.

21.1.19 Viewer

The viewer is activated by selecting any one of its three commands: “WIRE”, “SOLID” or “FLAT”. When active the cursor disappears. Sideways movement of the mouse then allows any one of the three commands to be selected. Vertical movement allows any object to be selected using the object selector. The three commands are as follows:

I. **Wire Frame -** Any object can be viewed individually or if preferred the whole design without any visible background.

ii. **Solid -** Any individual object can be viewed drawn solid.

iii. **Pattern -** The flat pattern of any object can be viewed alongside the solid and wire frame drawings of the object. The “Options|View Mode” command provides option to exactly what this command displays.

21.1.20 Straightener

The “STRAIGHT” command is used to set a tapered object back to straight. The cross section dimensions at the position of the cursor on the object C/L is taken to be the new cross section at both ends.

21.1.21 Axes Alignment

The “AXES” command allows an active object to be aligned with any of the three axes cursor movement restricted to occur along that axis in a similar way to the way the slide facility restricts movement. The choice of axes is made by an automatic axes snap in response to the next position selection made in the room after this command is selected.

21.1.22 Outline

If an active object is drawn as a C/L, then selecting the “OUTLINE” command will cause it to be redrawn in its outline state. If its is drawn in outline then the command will cause the object to be returned to its C/L form. In other words the command toggles the active object between its C/L and uncut or outline state.

21.1.23 Reactivate

The “LIVE” command allows any object in the design to become active again with the cursor positioned on either end. If the object is related to another, it will pivot about the point of intersection. If it is related to more than one other with more than one point of intersection, then re-activation will not be permitted. At re-activation the object length is initially frozen. To unfreeze it use the “LENGTH” command.

21.1.24 Oblique

The “OBLIQUE” command allows a C/L or object to be obliquely offset from its original position. If an object is already active the command cause oblique movement to begin at the next selected cursor position. If no object is active the sniffer is set so that an object can be chosen and reactivated in oblique mode. Oblique movement keeps the object ends in the same plane whilst moving the orientation of the C/L.

21.1.25 Dimension

When an object is active the “DIMENSION” command allows you to change the dimensions which appear at the bottom of the screen from the actual dimensions of the active object to its relative dimensions with another related object. If related to more than one, the object is selected using the object selector. If an object is not active but the cursor is sliding on a C/L then the dimension of this object are displayed. In this case the end dimensions displayed are the same end that the slide distance is currently measured from. The F6 key will toggle the ends.

21.1.26 Unhook

The “UNHOOK” command can be used to sever all or some of the relationships of an object which is being *actively dragged*. If dragged from a mid point all relationships are broken. If dragged from an end all relationships at that end only are broken. The unhook command can also be used if the slide is set to an object C/L. The position of the cursor on the object C/L decides which relationships are broken in a similar way.

21.1.27 Duplicate

The “COPY” command is used to copy and drag away a duplicate object from one

already in the design. Selection is made by the sniffer.

21.1.28 Unfold

Unfolded patterns can also be produced from the "UNFOLD" command in the design menu or the “Compute” menu. The flat pattern of the selected object is calculated and displayed full screen size.

21.1.29 Cut & Weld

The “CUT” command can also be accessed through the "Cut & Weld" command in the "Compute" menu. The selected object/s intersections are calculated and displayed in their cut form.

21.1.30 Lobster

Lobster back bends (or segmented bends) can be produced via the "LOBSTER" command in the design menu or the "GRID" menu. A lobster back bend is created by selecting two objects forming a simple bend relationship.You are prompted to respond to several commands regarding the form of the bend and then prompted for the radius of the bend given the maximum possible as a default. The number of segments is also selected before the bend is generated, thus replacing the original objects.

21.1.31 Scale

Using the “SCALE” command the design can be scaled by a given factor. If the cursor is currently sliding the position of the cursor is used as the scale centre, otherwise a mid 3D position of the design is used. Scaling is also accessible though the "Scale" command in the "GRID" menu.

21.1.32 Zoom All

Using the “ZOOM” command the current view is reset and redrawn to the default zoom setting as determined by the size of room in use.

21.1.33 Wire Frame

The “WFRAME” command allows a wire frame of an object or the whole design to be placed on the current sheet arrangement. If the arrangement is not currently visible it is first redrawn. The command is also available via the "Arrange|Wire Frame" command.

21.1.34 Pattern

The “PATTERN” command allows an unfolded pattern to be placed on the current sheet arrangement. If the arrangement is not currently visible it is first redrawn. The command is also available via the "Arrange|Pattern" command.

21.1.35 Match Chain

The “MATCH” initiates chain matching process starting at the selected object. The command is also available through the "Match|Initiate". It instructs all objects related to the it to match themseleves to it (adjust their size to give maximum fit). Then the objects related to these objects are also matched. All match operation are by default set to ask for confirmation before adjusting the size of any object. The option settings that define the way matching works are found in the “Match” menu.

21.1.36 Match Individual

The “MATCH1" command causes the selected object to be matched to its relatives. The process can also be accessed through the "Match|Individual" command.

21.1.37 Rotate

The “ROTATE” command causes the design to be rotated about the selected X,Y or Z axis. If the slide is currently set to an object then the both this object and all those related in an object chain are rotated. Also in this case the cursor position is used as the centre of rotation. If the slide is not set the whole design is rotated about a mid 3D position of the design.

21.1.38 Orthogonal

The “ORTHOG” command toggle orthogonal mode. If set the command is highlighted and all new active objects will not remain attached to the cursor but will align with the nearest orthogonal axis to the cursor position. The only exception is where the cursor is set to slide on an object C/L. Then the active object will follow the cursor.

21.1.39 Help

The “HELP” command activates the help system which remains active until the command is selected again. Alternatively the F1 key can be used to toggle the help facility. Note that to use the help facility the file "SHEET.HLP" must be available in the current directory. The help appears occupying the bottom lines of the design area. The border indicates the number of help pages on the topic and provides paging facilites for mouse selection.

Help has two modes of operation that can be toggled from the "Options|Help Mode" command. The option settings are either "Query" or "Inter". Query allows help to be

given on selection without any action performed. "Inter" is a simple interrupt that allows you to proceed with the operation while the system interrupts the selection and updates the help box with information on the selected item, in effect giving you help on each item as you work.

21.2 File Menu

The menu is pulled down from the "File" command in the main menu.

(a) **Load -** Loads a file from disk which includes the environment in which it was created. The directory can be searched and files browsed using the facilites is the file load dialog box..

(b) **Save -** Saves a file under the current name, automatically ovewriting the current file of that name.

(c) **New -** The design is cleared to begin a new design. The environment remains unaltered.

(d) **Append -** Loads the design from a file without deleting the current design or changing the environment. The loaded design is added or appended to the existing. The current environment can be used to work on an existing design file by using "New" followed by "Append".

(e) **Save as -** Allows the current file to be saved under another name typed in at the prompt. The Save As command is also accessible by selecting the name text on the second text line.

(f) **Erase -** Erases a specified file from disk.

(g) **PARAMETRIC** - Acceses the parametric menu.

(h) **Dir -** changes the current directory.

(i) **Quit -** Exits SHEET LIGHTNING back to the operating system. The current menu configuration is saved if enabled in the options menu.

21.3 Output Menu

The output menu is pulled down from the "Output" command in the main menu. If contains all commands for producing output either directly to a device such as a plotter or printer, or to a file.

(a) **PLOTTER OUTPUT -** provides access to the plotter output menu. Quick access to this menu can be achieved using the ALT-L key.

(b) **PRINTER OUTPUT -** provides access to the printer output menu. Quick access to this menu can be achieved using the ALT-I key.

(c) **FILES OUTPUT -** provides access to the files output menu. The commands in this menu send output directly to files in various formats.

(d) **LINES -** provides access to the lines menu which holds toggle commands to turn the visibility of various line types on and off in the wire frame and pattern images. Options also exist segment plots of adapter pieces and to control the form of output to DXF files.

(e) **Frame Sheet -** Toggle command having values “On” and Off” indicating whether lines representing the sheet outline should be despatched with a sheet arrangement plot, whether to device or file..

(f) **Recalc Increment -** This command holds the “Output Increment” value. This value is an angular increment defining the accuracy to which objects should be recalculated before an output operation. If the value is zero no recalculation is done. If it is non-zero the pattern data is always disposed of and the pattern recalculated before outputting the data. This allows you to work in a draft increment but produce output of high accuracy.

21.4 Plotter Output Menu

The plotter output menu is a submenu of the "Output" menu. It holds all facilities specifically for setting up and performing output to a plotter.

(a) **Plot Sheet -** Plots the current sheet arrangement directly to a device according to the device settings in the "protocols" menu. The plot can be divided up over multiple sheets. The program will ask you to identify which sheet you wish to plot.

(b) **Device/Port -** You must set the port to the port connected to your plotter. The choices are COM1,COM2,LPT1,LPT2 and LPT3. You also have the an option called “File”. This will stop before outputting data and ask you to type in the device port name. You can use it to access another port or divert output to a file.

(c) **SIZES -** provides access to the plotter size settings and optional parameters. It may be necessary for you to set the values up to correspond to your plotter. On the other hand if you are using serial port communication (usually a COM port) the program may be able to read th plotter device and fill in the relvant information itself using the “Read Plotter” command in the sizes menu.

(d) **PROTOCOLS -** provides access to the protocols menu. The protocols define the speed and mode of data transfer used in serial data transfer. The settings in the program must match the settings on the plotter or device.

(e) **CONTROL -** provides access to the plotter control menu. The values in it define options in the way the plotter operates.

21.5 Sizes Menu (plotter output)

The plotter output sizes menu is a submenu of the "Plotter Output" menu. It holds the settings for the physical size and resolution of the plotter..

(a) **Read Plotter -** This command is for serial port devices only. It attempts to read th physical size and resolution of the plotter directly by sending in various HPGL commands. It the plotter or device supports these commands it will respond by sending back the required information. If successful the “Hard Clips”, “XY-Origin” and “HPGL Increment” are set automatically. Failing this the value must be set by hand or the plot may be displaced and in the wrong scale.

(b) **Hard Clips -** This is an input comman holding the physical width and size of the sheet. In some cases the hard clips are measured by the plotter each time a new sheet of paper is inserted. If you wish to get the maximum size each tme you must perform a “Plotter Read” command each time you replace the sheet. If you are doing a multiple sheet plot be sure to peform a “Read Plotter” command only once before producing any output. Also, for a plotter that has a varying sheet size give it a border to make it tolerate for slight variations in the size of sheet.

(c) **XY-Origin** - The origin of the plot can be at one corner or even n the centre of the sheet depending on the plotter or device. These value are set by the “Read Plotter” command for serial port devices, if successful. If not the values must be set by hand according to the device.

(d) **Border** - A border can be placed around a plot effectively reducing the hard clips size by the given amount. This is particularly useful for plotters that measure the sheet size each time a sheet of paper is inserted. The slight variation caused by the way you put the paper in the device can cause the plot to go over the edge of the hard clips if the full plot size is used. A border prevents this giving a degree of tolerance. Each sheet in the sheet arrangement is the size specified in “Hard Clips” less the size specfied in “Border”.

(e) **Partitions** - The partitions define the number of sheets appearing in the sheet arrangement. It is specified in the form “2x2" for a matrix of sheets 2 wide and 2 high. Each sheet is of the size specified in “Hard Clips” less the “Border”. The partioning facility allows you to produce a plot over as many sheets as it takes to cover it. You can therefore produce any size of pattern on any size device. Note that for this size of sheet and these partitions to be used in a sheet arrangement the “Sheet|Specification” toggle command must be set to “Plotter”. If it is set to “Printer” the printer sizes are used as set in the submenus of the printer output menu.

(f) **HPGL Increment -** This is the resolution of the plotter. A plotter only actually works in whole numbers. Each increment being a very small movement. The increment specified here defines the number of increments per inch the plotter uses.

The value is set by the “Read Plotter” command if successful. If not it must be set by hand according to the plotter specification or the plot may be scaled wrongly.

21.6 Protocols Menu (plotter output)

The protocols menu holds the standard settings for serial port data transfer. The settings must match the settings of the device for the data transfer to work properly. If a parallel port is used the settings are irrelevant.

(a) **Handshake -** This toggle option defines the form of “hand shaking” that takes place between the computer and the device. It is a means of controlling the flow of data to ensure the buffers of the device are not overloaded. There are three types of : Hwire (i.e. Hardwire) which uses one of the serial port pins to enable transfer (must be wired correctly), XON/XOFF which relies on the plotter returning information about the state of its buffer and “Soft” which simply monitors the progress of data passing from the computer to the plotter to satify the system that there is an ongoing data flow.

(b) **Baud Rate** - Defines the rate of serial data transfer between the computer and the device in bits per second. The options are 110, 150, 300, 600, 1200, 2400, 4800 and 9600.

(c) **Databits** - The databits are the number of consecutive bits that can be treated as data during transmission. This value can be set to either 7 or 8.

(d) **Stopbits** - The stopbit is the bit that follows the last bit of a byte in serial data transmission and is used as a signal to the receiver that all of the byte has been transmitted. It's value can be set to either 1 or 2.

(e) **Parity** - The type of checking used for data transfer errors. The options are “None”, “Odd” or “Even”.

21.7 Control Menu (plotter output)

Hold standard HPGL variable settings for the operation of the plotter.

(a) **Velocity -** The velocity of the pen. If set to 0 as it is by default no velocity code is send to the device. Consult your plotter manual for details of the interpretation of velocity values.

(b) **Force** - The downward force of the pen on the paper can be set for best results according to the type of pen and paper in use. A value of 0 prevents any force code being passed to the plotter.

(c) **Acceleration** - The acceleration of the plotter pen can be set to None, 1.G, 2.G, 3.G or 4.G. A higher acceleration can considerably increase the speed of plotting but puts more strain on the device. Consult your manual for recommended settings.

(d) **Rotate 90** - The plotter paper size can be rotated 90 degrees. You may wish to do this to create a more convenient shaat arrangement. If the setting is changed it is immediately reflected in the sheet matrix.

(e) **Time-Out** - Defines a period the program will wait to continue data flow when the flow has been stopped by handshaking or errors. If the specified time (in seconds) runs out a time out error will be issued.

21.8 Printer Output

The printer output menu holds all commands and options required to pass text and/or image data to a printer.

(a) **Print Sheet Image -** If the "Sheet|Specification" toggle command is set to printer the sheet arrangement will use the sheet sizes set in printer output menu and its sub-menus. This command passes the data to the printer in the same way as you would a plotter. The output to the printer are the pattern images. For this facility to operate you must obtain a copy of the shareware program PRINTGL. A license fee must be paid on this software if you wish to retain it and use it.

(b) **Print XY-Coords -** The sheet arrangement is despatched to the printer in a textual format. The values it prints are the XY coordinates of the patters from the sheet origin. You can use this data for hand plotting unfolded patterns.

(c) **Select Printer -** The facility to select a printer is mainly to enable the sheet image output using the "Print Sheet Image" command. The selection of printers may be limited because a DOS program must continually keep updating its libraries to stay in touch with new product releases..

(d) **Device/Port -** Define the port to be used for printer output. The options given are LPT1, LPT2, LPT3 and "File". If set to file another port can be specified or the output can be diverted to a file. The prompt for the port or file appears as the output commands are used. Serial port communication is not supported for printer output.

(e) **SIZES -** Provides access to the printer size menu which hold size and device origin settings.

(f) **CONTROLS -** Provides access to the printer output controls menu that controls the form of output to the printer.

21.9 SIZES (printer output)

The printer output sizes menu holds data defining the physical attributes of the printer. These must be set manually to correspond with the printer is use to obtain correct output. The values are saved in the configuation file.

(a) **Hard Clips -** The physical sheet size of the printer.

(b) **XY-Origin -** The X and Y origin of the printer sheet. Consult your printer manual to obtain the required setting.

(c) **Partitions -** The partitions define the number of sheets appearing in the sheet arrangement. It is specified in the form “2x2" for a matrix of sheets 2 wide and 2 high. Each sheet is of the size specified in “Hard Clips”. The partioning facility allows you to produce a plot over as many sheets as it takes to cover it. You can therefore produce any size of pattern on any size printer. Note that for this size of sheet and these partitions to be used in a sheet arrangement the “Sheet|Specification” toggle command must be set to “Printer”. If it is set to “Plotter” the plotter sizes are used.

21.10 CONTROL (printer output)

The printer output control menu holds setting or values that control the form of output the printer produces.

(a) **Setup Code -** This code is passed to the printer at the commencment of any textual output. It should contain key codes that setup the printer with the required font and page size. Consult your printer manual for what codes are required to produce the desired result.

(b) **Decimals -** Simply defines the number of decimal places to be used on numerical output values.

21.11 FILES OUTPUT Menu

Provides the command for producing output data files in various formats.

(a) **XY-Coords Sheet -** The sheet arrangement is output to a specified file in numerical ASCII text format. The result can be loaded read by a text editor or word processor. The XY coordinates are the X,Y positions of all points along the lines defining the unfolded pattern shapes.

(b) **DXF Sheet -** Ouput the sheet arrangement data to a data exchange file of the same name as the design but with a "DXF" extension, or to a typed file name. The data can usually be retrieved in this form by other CAD systems.

(c) **HPGL Sheet -** Outputs the sheet arrangement plot data to a HPGL file of extension "HGL". Many CAD system, word processors, desktop publishers and other programs support this graphics data format.

(d) **DXF 3D-Design -** Outputs the cut and welded design image to a DXF data exchange file. The data can then be imported to a CAD system that supports DXF format, usually for producing images, perhaps for a brochure. The shaded images of Sheet Lightning designs can be used to create impressive brochure material. Note that only the objects that are cut and welded are despatched to the file. Uncut objects are not included. You should also be aware that to produce shaded 3D images you must make sure that the “DATA|Face Data” option is set to “On” and the design has been cut and welded since this toggle was set. The design images will then be produced as 3D faces rather than simple wire frames this allows shading and solid viewing. By default this toggle is not set because these image take longer to calculate.

(e) **DXF Pattern -** Outputs the unfolded pattern data of all objects in the design in DXF format without reference to any sheet layout. The patterns are placed in a column one above the other. This is the easiest form of pattern output. The patterns can be picked up and arranged to suit by a third party CAD or CAM system.

(f) **HPGL Pattern -** Outputs the unfolded pattern data of all objects in the design in HPGL format without reference to any sheet layout. The patterns are placed in a column one above the other in exactly the same way as the “DXF Pattern” command.

21.12 LINES Menu

The lines menu enables or disables the display and output of the the various types or categories of lines that are used in Sheet Lightning. The line type toggles generally have four possible settings. These are “None”, “Wire”, “Pattn” and “All”. If it is set to “Pattn” the line type appears on unfolded patterns but not on wire frame or 3D images. If it is set to “Wire” the line type appears on wire frame images but not on unfolded patterns. “All” shows the line type on both and of course “None” disables them on both. When image out output only the visible lines are despatched. The program therefore works on the WYSIWYG basis.

(a) **Perimeters -** Lines defining the perimeter of a pattern or 3D shape.

(b) **Holes -** Lines defining an enclosed cut-out in a pattern shape. This type of line is categorised because in cutting processes it is usually necessary to cut the holes out first.

(c) **Folds -** Object surface lines defining the join between one part of an objects cross section and another.

(d) **Radials -** Object surface lines equally spaced around the circumference of an object. The number of radial lines is determined by the setting of the “DATA|Radials” input command.

(e) **Nets -** Object surface lines appearing between radials to ensure the accuracy of the plot is no less than that defining by the objects calculation increment.

(f) **Edges -** Only appearing in 3D view that have been calculated using the “Rigid” setting for the “DATA|Flexibility” toggle.

(g) **Faces -** If the “DATA|Face Data” toggle command was set “On” the cut and welded 3D designs will be produced with faces instead of as a wire frame. This toggle switches the visibility of the faces on or off.

(h) **Seams -** The seam is the point where the object surface is split longitudinally to enable it to be unfolded.

(i) **Face Edges -** It is usually desireable to see the face ends but not the side edges. This toggle can be used show the edges.

(j) **Divisions -** A rectangular or adapter piece can be split into either two or four segments if required. The setting are “Whole” for no split, “Half” to split it in two 180 degree from the seam and “Quart” to split is into four down the middle of each flat side. If you change the toggle be sure to recalculate the unfolded patterns using the “Compute|Unfold” command before despatching output.

(k) **DXF Blocks -** This toggle allow the DXF output to be formatted in blocks. Not all CAD systems support blocks so you may need to despatch the output with this option turned off. Longitudinal lines, perimeters and holes are all placed in separate blocks.

(l) **DXF Lines -** DXF lines can be despatched as either lines or polylines. Polylines are better if supported by your Cad system because they require less data.

21.13 Compute Menu

This menu is pulled down from the "Compute" command in the main menu.

(a) **Cut & Weld -** Performs intersection calculations for any selected object in the design and draws the cut object to screen. Object selection is via the object selector.

(b) **Unfold -** performs intersection calculations for selected objects to produce flat patterns. The patterns are displayed full size on screen as the calculations are completed.

(c) **Increment -** The angular calculation increment can be set to give the required level of accuracy in a Cut & Weld or flat pattern calculation operation. It can be set for individual objects, or for all objects in the design.

(d) **GRID -** Accesses the Grid popup menu. This menu generally holds high level commands to perform specialised operations on designs.

(e) **DATA -** Accesses the Data popup menu. This menu generally holds toggle commands that affect the way the program generates data.

(f) **EFFICIENCY -** Accesses the Efficiency popup menu. This menu holds toggle command options to increase the programs capacity for dealing with complexity.

(g) **Redraw -** Forces the current environment and design to be redrawn.

(h) **Dispose -** Disposes of all cutting data in the design and returns all objects to their whole or uncut state.

(i) **ROBOTICS** - Accesses the robotics popup menu.

21.14 Robotics Menu

The robotics menu is accessed through the “Compute” menu. It holds demonstration features only. These feature generally demonstrate the capability of the program to generate 3D robotic cutting and/or welding path data for both 3D designs and unfolded patterns.

(a) **Torch Weld -** This command performs a visible 3D weld operation on the design in the design environment. The welding torch moves around with the required orientation to give the best weld possible.

(b) **Plate Cut -** This command performs a visible plate cutting operation on the design in the design environment. The cutting head is angled to the vertical to cut the plate with a weld preparation chamfer.

(c) **Increment -** The increment is a distance value defining one step for the welding or cutting torch. The head always passes through all points but the incremental steps between can be set to the required distance.

(d) **Torch Length -** The length of the welding torch can be defined by the user. A length of zero allows the system to choose a default length based on the room size..

(e) **Rotation -** A welding operation on a 3D design can be performed whilst rotating the whole design in order to keep the welding torch in as near a vertical position as possible. Just how vertical the torch is kept at any time often depends on the axis defined for the rotation. The axis used is one of the centre lines of the design object or a construction line. If rotation is required then this toggle command should be set to “On”.

21.15 Arrange Menu

This menu is pulled down from the "Arrange" command in the main menu.

(a) **Pattern -** Used to add a pattern to the current sheet arrangement. If invisible at selection, the sheet will be drawn along with all existing patterns and images on it.

(b) **Wire Frame -** The wire frame command is used to add either an individual wire frame drawing from the design or the whole design image using the current view and scaling as seen relative to the sheet. The sheet is redrawn if it is not visible when the command is selected.

(c) **Erase -** Erases a design or image from the sheet arrangement. The item to be erased is identified using the up/down movement of the mouse of up and down arrow keys.

(d) **Multiple -** This is a toggle option defining options in the way objects are placed in the sheet arrangement. The “Mult” option places multiples of the same image on the sheet by dragging a new image away each time one is placed. The “Grid” option allows a whole design to be placed, one pattern at a time and the “One” option allows only one of the selected objects patterns to be place in any one command. In the last case no pattern copies are dragged away.

21.16 Sheet Menu

This menu pulled down from the "Sheet" command in the main menu.

(a) **Hide Sheet -** If the current sheet is visible it disappears but the sheet arrangement and pattern data is retained in memory. If the current sheet is invisible, then it is re-drawn with all patterns previously placed on it.

(b) **New Sheet -** Clears the current sheet arrangement. The sheet disappears until the next pattern or wireframe placement command is selected, or "Hide Sheet" is selected.

(c) **Specification -** The size of the sheets in the sheet arrangement depend on the setting of this toggle command. If it is set to “Plotter” then the sizes and number of partitions defined in the “Plotter Output|Sizes” menu are used. Otherwise the option is set to “Printer” and the sheet sizes used are set in the “Printer Output|Sizes” menu.

21.17 GRID Menu

The Grid menu is accessed through the "Grid" command in the Compute menu.

(a) **Organise -** Can be used to alter the order of objects in the design.

(b) **Default** - Selects an object whose diameter becomes the new default for all subsequent new objects.

(c) **Root Radius -** Rectangular objects can be created with a root radius at the folds. The current size of this radius is set here in response to an input box prompt.

(d) **Lobster -** Utility to design a lobster back bend (i.e. a swept bend or a segmented bend) from an existing two piece bend in the design.

(e) **Axial Copy -** Objects can be copied at set increments around a chosen axis. The axis must be chosen as either the centre line of an object or as a construction line. If the object being copied is related to the axis object the relationship can also be copied. A box pops up asking for confirmation whether the relationship should be copied.

(f) **Scale -** Scales the whole design by an input factor.

(g) **Spin Grid -** Spins the whole design in the plane determined by the next command at an incremental angle chosen through the number menu.

(h) **Rotate Axes -** Selects one of the three 2D viewing planes in which to spin the design, or section of the design.

21.18 DATA Menu

The Data menu is accessed through the "Data" command in the Compute menu.

(a) **Propagation -** Enables propagation of intersecting lines from the target object to the subject object. Lines are inserted in the circumference of the intersected object so that when its own intersection is calculated the two intersecting objects are calculated along surface lines that meet.

(b) **Triple Points -** Where three objects meet a triple point occurs. If this option is set to “Find” the intersection process hunts for these triple points and includes a calculation line at that point in each of the object surfaces, thereby giving an accurate pattern that get right into the corners of the intersection.

(c) **Flexibility -** Object cut data can be calculated either to allow solid viewing or to allow changes of view and object rotation without recalculation.

(d) **In/Outside** - The unfolded patterns can be calculated and displayed as if looking on the inside of the object, or looking on the outside.

(e) **Face Data -** Cut and weld data is calculated as surfaces or faces rather than simple lines.

(f) **Irregulars -** Enables intersections between complex pieces to be calculated. With this command disabled any objects other than circular sections will always be cut square to the ends.

(i) **Parallel -** In special cases where a flat side of an object intersects with a flat side of another object there are a number of possiblities to gicve a valid intersection. This toggle option simple defines whether the objects are treated equally or one object is allowed to dominate the other. The “Equal” option enables equal treatment. The “Order” option causes the first object appearing in the design to dominate the relationship. If you want the other object to dominate then you must change the order of objects in the design using the “GRID|Organise” command.

(j) **Radials -** Defines the number of radial lines in any circular object throughout its full 360 degrees of circumference. The radial lines are equally spaced and normally visible.. In an adapter object or an object with circular corners each corner sweeps 90 degrees of circumference and should therefore have a quarter of the specified number of radial lines.

(k) **FE Meshing** - When FE meshing is enabled all lines around the circumference of an object are visible. The first object is divided up into equal increments according to the increment setting. These lines are then propogated to the next object in order

to divide it up with common points of intersection.

21.19 Match Menu

Accessed through the "Match" command in the main menu.

(a) **Initiate -** Initiates a chain matching process from a selected object. The “MATCH” command in the design menu performs exactly the same function. Each object is matched to the selected object, then each affected object is matched.

(b) **Individual -** Matches any single object to its relations. The “MATCH1" facility in the design menu performs exactly the same function. Only the slected objects size is altered.

(c) **Match Mode -** A toggle command that can be set to “Ask” or “Auto”. “Ask” forces confimation to be given before any form of matching occurs. “Auto” causes matching to proceed without confirmation.

(d) **Ring -** Ring matching is the name given to the process of matching an object to two objects at once. For example where an object is related to another at both ends. This is a toggle command that can be set to “Ask” or “Auto”. “Ask” forces confimation before a ring match occurs. “Auto” permits ring matching to proceed without confirmation.

(e) **Chain -** Toggle command with values “On” and “Off”. In set to “On” matching occurs along the full chain of objects and affected objects. If “Off” matching stops after the initiating objects relations have been matched.

(f) **Add -** Toggle command with values “On” and “Off”. Add matching occurs when a new object is added. The new object is matched to it relation/s.

(g) **Face -** Toggle command with values “On” and “Off”. Face matching forces the dimensions of a new object to be copied from the face of a related one, or if the new object is hooked to the related one at a mid-point along its centre line then the cross section of the new object is the same as the cross section of the related object at that point along its centre line..

(h) **C.C.Sphere -** This is a toggle command that can be set to “On” or “Off”. It provides a special case for inline situations between circular section objects without oblique offset. The match is performed to maintain a common central sphere. It means that the intersection always results in a straight cut plane.

(I) **Displace -** Toggle command that can be set to “Obliq”, “Turn”, “Offst”, “Ask” or “None”. It determines whether and how a matching operation can displace an object C/L in order to achieve the match. The match can be achieved by skewing the object, turning the object or by simple displacement.

(j) **Dimension -** Toggle command having options "Ask", "None", "Grow" or "Taper". It determines the way in which an objects dimensions can be altered in order to perform a matching operation. This may be by tapering or by proportionally growing the whole object.

(k) **Copy Face -** Special case toggle option having settings "On" and "Off". In a chain matching operation this command determines whether, in an in-line object situation, the dimensions of one end of the object should be copied from the face of another. It only applies to bend relationships.

(l) **Copy Angle -** Control a facility to transmit the angle of taper from one object to another in a matching operation.

(m) **Flicker -** Toggle command that can be "On" or "Off". If the flicker is set to "On" then during an automatic operation each object that is matched first flickers, indicating that it is about to be matched.

21.20 Scene Menu

This menu is pulled down from the "Scene" command in the main menu. The scene menu is generally used to set the background view of the 3D room and to control the positioning of the design within the room.

(a) **Show Room -** This is a toggle command allowing a choice of "Posts", "Walls", "Edges" or "None". The default is “Posts”.

i. **Posts -** The axes are represented by three square posts.

ii. **Walls -** The complete room is drawn. The axes being the corners of the room.

iii. **Edges -** Similar to the posts but without a vertical post thus giving an edging effect to the floor area.

iv. **None -** Nothing is drawn to represent the axes except the floor if not disabled by the "Floor Area" command.

v. **Axes -** A simple line representation of the axes directions with labelling.

(b) **Show Axes** - This is a toggle command with options of “On” or “Off”. It simply controls whether or not the axes icon is visible in the design environment.

(c) **Room Size** - The three room size dimensions are set by typing the values for into length (X), breadth (Y) and height (Z) of the room into input boxes.

(d) **Scale Room -** The current room can be scaled by a given factor thus keeping the relative dimensional proportions the same.

(e) **Centre Design** - Resets the room size to a reasonably close fit to the current design and moves the design to the centre of the room giving a good view of it in all four view planes without zooming or panning.

(f) **Centre View** - Pans and zooms the view to the centre of the design in the view window. No alteration is made to the room size or the position of the design in the room. The “ZOOM” command in the design menu will restore the default view.

(g) **Zoom View** - Zooms the view by the current zoom factor. This command is identical to the “ZOOM” command in the design menu.

(h) **COLOUR** - Accesses the colour menu. Quick access to this menu can be achieved using key ALT-R.

(i) **Floor Area -** The floor area command is a toggle which can be set to "Tiled", "Clear" or "None". The default is TILED.

I. **Tiled -** The floor is drawn with a number of broadly spaced lines parallel to the axes to help visualisation.

ii. **Clear -** The floor is drawn but without any lines.

iii. **None -** No floor is drawn.

(j) **Zoom Factor -** The zoom factor defines the multipication factor used to expand or contract the view scope. A factor of 3 will enlarge or reduce by one third.

(k) **Zoom -** This toggle can be set to “In” or “Out”. It is used to set the zoom facilities for zooming in or out as required. The default is “In”.

21.21 COLOUR Menu

The re-colour menu is accessed through the re-colour command in the scene menu.

(a) **Solid -** Allows changes to both the foreground and background colour of an object using the colour menu.

(b) **Transparent -** Enables an option which causes the near side of a solid drawn object to be transparent but with the far side filled as normal thus showing the inside intersection lines clearly.

(c) **Background -** The inside colour of all objects in the design can be given the same colour.

(d) **Outline -** The current outline colour of objects can be changed.

(e) **Wire -** The wire frame drawing colour can be set to the individual colours of the objects or they can all be drawn using the current outline colour.

21.22 Options Menu

This menu is pulled down from the "Options" heading in the main menu.

(a) **CONFIGURATION -** Accesses the configuration menu. This menu provides facilities for saving the state of the program for automatic reloading at startup.

(b) **System Mode** - Defines which mode the program should start up in. “Design” starts up in design mode, “Param” starts up in parametric mode, “Ask” lets you choose the mode at start up and “Config” starts up in the mode the system was in when the configuration file was saved.

(c) **View Mode**

The view mode determines in how many different ways the objects are represented together when using the flat mode in the viewer. The options are "Flat", "Built" and "Solid". Each separate view is scaled to fit one quarter of the design area of the screen.

Flat - Draws the flat pattern (full screen).

Solid - Draws the flat and solid shape of an object.

Built - Draw the flat, solid and wire frame of individual objects along with the built view of all objects.

(d) **Freedom**

This is a toggle command that controls the way the X,Y and Z stabilisers operate. There are three options: “Axis”, “Plane” and “Multi”.

i. Axis - With the toggle set to “Axis”, when a stabiliser is selected only this axis is highlighted, the others are turned off. The movement of the cursor in the design is therefore restricted to movement along the single axis. Selecting the X,Y and Z dimension values below the main menu allows you to change the setting of the fixed axes.

ii. Plane - When a stabiliser is selected it is turned off and the other two are highlighted. Selecting the Z stabiliser therefore allows movement in the XY plane. Similarly for the other axes.

iii. Multi - Each stabiliser is controlled individually and can be turned on or off by simply selecting it. This gives full control of the cursor restriction in the room. If all axes are on then te movement defaults to the XY plane.

(e) **Sensitivity -** Alters the sensitivity of the mouse to movement. The lower the value the higher the sensitivity.

(f) **Acceleration -** Alters the rate at which the cursor accelerates when the mouse is moved. The setting of this option helps jumping the gap to the design menu when cursor movement in the room is restricted.

(g) **Selection -** The selection of object in the design can be achieved in one of two ways. Either using the object selector or by direct picking. The options are therefore either "Bar" or "Pick".

(h) **SOUND -** Accesses the sound menu.

(i) **MENU TEXT -** Accesses the menu text menu.

(j) **Dialog Boxes** -

(k) **Progress -** Enables/Disables the progress reporting on the bottom line of the screen when calculating intersections.

(l) **Instruction -** Enables/Disables the writing of instructions to the bottom line of the screen during design processes.

(m) **File Viewing** - Controls the display of graphics in the file search dilaog boxes. “On” is the default which enables graphics design viewing. Switching the facility off can be beneficial if file loading on your hardware is slow.

(n) **Help Mode -** Toggle setting of either "Query" or "Inter". Provides for querying the help system without any action performed or allows action to continue and acts as an interupt to update the help box.

21.23 Configuration Menu

This menu is accessed through the "Options|CONFIGURATION" command.

(a) **Save Configuration -** This command saves all the toggle and value settings in the the menus to a configuration file called SHEET.CFG. The configuration file is loaded automatically at start-up so the system should resume the state it had when this command was used. ALT-F2 forces the configuration to be saved at any time.

(b) **Erase Configuration** - Erases the configuration file SHEET.CFG. On start-up the program starts wilth the default settings if this file is not found.

(c) **Quit Save** - This is a toggle command that can be set to either “On” or “Off”. If “On” then the configuration file is automatically saved when the “File|Quit” command is used.

21.24 SOUND Menu

Used to enable/disable sound.

(a) Selection - Indicates the incremental changes when making a selection such as when using the object selector. Default is OFF.

(b) Error - Accompanies an error report on the error line. Default is ON.
(c) Abort - Indicated an operation has been manually aborted such as a cut and weld calculation or a match operation. Default is ON.

(d) Sheet - Indicates the movement of the cursor when selecting a sheet for despatching.

(e) Match - Indicates the completion of each individual match operation in a chain matching situation. Default is OFF.

21.25 EFFICIENCY Menu

Allow the selection of a number of levels of complexity in intersection calculations.

(a) Termination - Calculate intersection with consideration of the termination of objects by other object intersections. A level of termination checking is selected according to the complexity of the case. Lower settings can have a significant effect on calculation time.

(b) Non-Relative - Restricts intersection calculations to related objects for speed purposes.

(c) Conserve - Removes unnecessary intermediate points in vectors which are aligned with each other.

21.26 MENU TEXT Menu

The menu text commands are accessed via the "Options|Menu Text" menu.

(a) **Save Menu Text -** The menu, instruction and error text can be output to file "SH_OUT.MNU". This text can be altered for other languages or customised to requirement.

(b) **Load Menu Text -** The text stored in file "SH_IN.MNU" is loaded to replace the current menu, instruction and error text. If the wrong format is detected an error will be reported and the operation aborted. An attempt to load this file is always made at start-up if the file is found in the current directory.

22 ERROR/WARNING MESSAGES

A number of error and warning messages will be encountered in using SHEET LIGHTNING. They appear on the error line which is the second text line of the screen below the main menu commands.

22.1 Errors

When an error is reported, the word "Error" will appear with a message giving some indication of the problem. The error line appears in red. Its effect is always fatal in that the error prevent the current operation from going ahead. Errors must always be acknowledged by pressing the right-hand <escape> button before execution continues.

22.2 Warnings

A warning is similar except that the word "Warning" appears, the error line appears green instead of red and it can be acknowledged by the press of any key including the escape key.

22.3 Messages

The following messages may be encountered:

1. **Out of memory** - Insufficient memory to perform the current operation. SHEET LIGHTNING will normally attempt to "acquire" memory by disposing of cut data before issuing this message.

3. **Rigid centre line -** An object cannot be reactivated due to the relationships it has holding the C/L in more than one position along its length.

4. **File not found -** No result in a search for files under the a name or template.

5. **Illegal file type -** An attempt to load a file of an incorrect type for the purpose.

6. **Object Active -** An illegal attempt to perform some operation which cannot be done whilst an object is active in the design.

7. **Too Many Objects -** The number of objects in the design exceed the

maximum limit.

8. **Maximum Pieces -** The objects is fragmented into too many pieces in the cut and weld or pattern calculation.

9. **Despatch Increment Too Small -** Not enough memory to hold the despatch data produced by calculations at this level.

10. **No Sheet Prepared -** Attempt to output an empty sheet arrangement.

11. **Device Input/Output Error -** Unable to detect, read or write to an output device.

12. **Append File Not Found -** File search failed in response to append operation.

13. **Input/Output Error -** Unable to read or write data to or from a disc file.

14. **Piece Error -** Cut data does not split into the correct number of pieces. This is invariably due to the pieces being counted from a cut with one increment setting and it being used to select a piece from the re-calculated pattern at a different increment usually during a despatch operation. The solution is to select the despatch increment rather than the draught increment to design a sheet arrangement in this case.

15. **Zero Length Error -** An attempt to place both end of an object at the same position thus given an object with zero length.

16. **Data Error -** An error in the form or format of data read from a file.

17. **Rigid Pivot Point -** An attempt to activate an object from a tied end.

18. **File Format Error -** An error in the format of data read from a file.

19. **No Chain Matching -** An attempt to perform a chain matching operation with chain matching switched of in the match menu.

20. **Illegal Slide Length -** An attempt to numerically set a slide position beyond the limits of an object's ends.

21. **Match Taper/Turn Error -** The object cannot be turned to perform a match. Often occurs with very short object objects.

22. **Security Lock Error -** Dongle not detected in LPT1.

23. **No Offset Relatives -** No related objects to offset the selected object from.

24. **Directory Not Found -** Cannot identify the typed directory.

25. **Match Negative Dimension Error -** The dimensions of an object would become negative if the match procedure were to go ahead, therefore the match is not possible. Try manually altering dimensions closer to the matched condition and then re-initiating a match operation.

26. **Sheet Parameter Error -** The dimensions of a sheet are for some reason unacceptable. Possibly the border is greater than the sheet itself.

27. **Illegal Lobster Tee -** Other objects are teed onto the ones selected to be replaced by the lobster back bend.

28. **Illegal Lobster Bend -** The objects selected to form the lobster back bend have other bend relations at the end which is to be replaced.

29. **Bend Not Found -** No bend relation detected.

30. **Illegal Offset -** The current operation can not be performed with an offset between the selected objects.

31. **Inline Centre Lines -** The current operation cannot be performed with the objects inline with each other.

32. **Illegal Angle -** The current operation cannot be performed with objects that are at the given angle.

33. **Illegal Sheet Parameters -** Illegal settings for the sheet size parameters.

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35. **Illegal Device Communication -** Error in the attempt to communicate with a serial port device.

36. **Successful Device Communication -** A successful interface to a plotter or serial device through the current serial port. All parameters are read back to the system.

37. **No Grid Objects -** No objects in the design relevant to the current operation.

38. **FE Analysis Function Disabled -** An aborted attempt to use FE

object mesh generation facilities. These facilities are disabled in this version of the system. Contact the system developer for details.

39. **Demonstration Version Output Disabled -** An illegal attempt to use the disabled output facilities with the demonstration version of the software. An authorised fully functional version of the system is required to use these facilities.

40. **Object Not Active -** An object must be active to use this facility.

41. **Grid Empty -** No objects in the design to operate on.

42. **Drag Or Slide Not Active -** Either the drag or the slide facility must active in order to use this facility.

43. **Object Or Slide Not Active -** Either an object or the slide facility must be active in order to perform this operation.

44. **Illegal Bend Radius -** The bend radius supplied for the lobster back bend creation is too large to be accommodated by these bend objects.

45. **Invalid Point Format -** The point that was typed in was in the wrong format. The format must be the same as that in which the point was given (i.e. "nnnn,nnnn,nnnn").

46. **Illegal Menu File Format -** The menu data file cannot be loaded because it is the wrong format.

47. **Menu Text Written To File SH_OUT.MNU -** Reports that the menu data has been successfully written to the given file.

48. **Menu Text Read From File SH_IN.MNU -** Reports that menu text data was successfully read from the given file.