



AVEVA

MARINE

ISODRAFT User Guide

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ISODRAFT User Guide

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1 Introduction to this Guide

1.1 Scope of this Guide

This guide introduces ISODRAFT, AVEVA Solution Ltd's isometric plotting facility. It explains the main concepts underlying ISODRAFT and describes how to tailor the options to your own standards.

Note: This guide does not give step-by-step instructions on how to operate ISODRAFT from the command line, as this information is included in the *ISODRAFT Reference Manual*.

1.2 How to Use this Guide

There are two types of ISODRAFT user:

- **Users**, who need to create isometrics using pre-defined option files.
New users will probably want to look first at *Using ISODRAFT*, which describes how to produce isometrics from the command line and how to use the Graphical User Interface (GUI). You may also find it helpful to read, *Piping Components in ISODRAFT*, which describes how piping components are usually plotted and annotated.
- **System Administrators**, who need to set up their own option files. You can create new files or modify existing ones. You must be a member of the **ISOADMIN** team to see the forms you want.
- This guide describes how you can set-up the option files. But, you will probably make most use of the Command Reference chapter in the *ISODRAFT Reference Manual*.

1.3 Training for ISODRAFT

New users of ISODRAFT, who will be responsible for setting company standards for isometric production, should attend an 'Isometric Drawing Production training course', which covers the relationship between the Catalogue database and ISODRAFT as well as ISODRAFT itself. You should have already attended a 'OUTFITTING Basic Training Course' and the 'OUTFITTING Catalogues and Specifications Course'.

1.4 How this Manual is Organised

This manual is divided up into a number of chapters as follows:

Using ISODRAFT - provides an introduction to ISODRAFT and explains how to produce isometrics in the default format.

Cataloguing Requirements - gives a brief description of how the Catalogue database (except Bolt Tables) is structured; in particular, the parts which are essential for using

ISODRAFT. You may need to refer to the *Catalogues and Specifications Reference Manual* for further details.

Bolting - Describes Bolt tables and how bolting requirements are calculated.

Design Requirements - describes how piping components should be designed so that ISODRAFT can process the pipes correctly; in particular, how split points in pipelines are stored in the OUTFITTING DESIGN database. You may also need to refer to the *OUTFITTING DESIGN Reference Manual*.

Piping Components in ISODRAFT - describes how piping components are drawn on isometrics.

Customising the Drawing Sheet - gives an overview of the commands used to customise the drawing sheet.

Material List and Other Reports - gives an overview of the commands used to customise the material list, and explains how to produce other reports.

Isometric Drawing - gives an overview of the commands used to customise the isometric drawing area.

Drawing Change Highlighting - gives an overview of the process of marking changes made to drawings.

Symbol Keys - describes how to modify the symbols (SKEYs) used to show piping components on isometrics, and how to create new ones.

ISODRAFT Transfer File - describes the ISODRAFT-ISODRAW transfer file. This information may be required by advanced users.

1.5 Conventions Used in this Manual

Commands shown in a combination of uppercase and lowercase letters can be abbreviated: the capital letters of the command indicate the **minimum abbreviation**

Note: This convention does *not* mean that you have to type the second part of the command in lowercase letters. You can enter commands in any combination of uppercase and lowercase letters.

For example, you can enter the command: **CHeck**

in any of the following forms: CH
CHE
CHEC
CHECK

You cannot abbreviate commands shown wholly in uppercase letters.

Lowercase italics describe **command arguments**. You should replace the description shown by the text or value you require. For example:

Name	Definition	Example
<i>name</i>	A OUTFITTING element name	/PIPE-A63
<i>refno</i>	A OUTFITTING reference number	=23/1403

Name	Definition	Example
<i>integer</i>	A positive integer	3
<i>value</i>	A positive or negative number	3.142
<i>word</i>	A sequence of up to four letters	FLOW
<i>text</i>	An alphanumeric string	'Enclose between closing quotes'
<i>filename</i>	The path name of a file	//net/sg99/testing. Note that when entering a full path name, it must start with two slashes: the first because all names in OUTFITTING must start with a slash, and the second is the root of the path name.

2 Using ISODRAFT

This chapter provides an introduction to ISODRAFT and describes how to use ISODRAFT to create an isometric drawing; by using either the GUI or by entering commands directly into the command line. The steps for processing isometric plot files using the GUI and Command Line are illustrated by examples in [Example of Detailing the CE Using the GUI](#) and [Example of Detailing Using ISODRAFT Commands](#), respectively.

2.1 Introduction to ISODRAFT

ISODRAFT can be used to produce isometric plot files of pipes and networks, from either the OUTFITTING DESIGN or Fabrication databases, to your own required standards. Normally, you will use these drawings for pipe work fabrication, but you can also use them during on-site erection.

You can produce standard isometrics for zone, pipe, branch or spool drawing elements or for a mixture of these elements. In addition, system isometrics, showing a complete piping network, and equipment trim isometrics can be produced.

Note: Mixed isometrics, containing elements from the OUTFITTING DESIGN and Fabrication databases, cannot be produced.

ISODRAFT produces your isometric drawings automatically, including any associated material lists you request. These material lists can specify: piping components; bolt requirements; pipe cutting lengths; etc. ISODRAFT uses the information in the project's OUTFITTING DESIGN, Catalogue and Fabrication databases to produce the required lists.

The isometrics produced can be fully dimensioned and annotated to ensure that you find them easy to use and unambiguous.

2.1.1 Types of Isometrics

You can include the following types of isometric in an ISODRAFT drawing:

- Combined erection and fabrication isometric (standard)
- Fabrication-only isometric (for shop floor use)
- Erection-only isometric (for field use)
- Spool drawing isometric

Each isometric type has its own forms of dimensioning annotation and material list.

You can also control:

- Complexity (drawing level density) of the isometric
- View direction
- Layout and units of dimensions

- Annotation of the components shown on the isometric (type, part number, specification, etc.)
- Welding information
- Insulation and tracing information
- Material list position (either on the drawing sheet or separately)
- Material list format
- Symbols used to represent different types of piping components
- Drawing sheet size
- Drawing sheet annotation (title block text, company logo etc.)
- Scaling of the isometric within the sheet (window size)
- Text size

All these drawing options have default settings so that you can begin creating drawings quickly.

Normally ISODRAFT will be set up to use **macros**, to produce the isometrics for each project. See the [Software Customisation Guide](#) for information about using macros in OUTFITTING.

2.2 Starting ISODRAFT

You can start ISODRAFT either by starting OUTFITTING in your usual way and selecting the ISODRAFT module, or by switching to the Isometric Generator (ISODRAFT) module from another module within OUTFITTING.

When ISODRAFT starts, the Application Window and the ISODRAFT Explorer are displayed. If you wish to control ISODRAFT from the command line, you can display the **ISODRAFT Command Input & Output** form by selecting **Display>Command Line** from the main menu bar.

2.3 Using the ISODRAFT GUI

ISODRAFT can be completely controlled from the application's GUI. The information displayed on the isometrics is controlled by Options files, allowing you to produce drawings to your own standards. Some option files are supplied with the product: these can be modified, or new option files created, to suit your company standards and projects.

The basic procedure for producing an isometric using the GUI is:

1. Select the type of isometric you wish to produce (Standard or System/Trim).
2. Select the options file you wish to use.
3. Select the element you wish to process, from the members list, or assemble a detail list containing the elements.
4. Select the type of output required.
5. Plot the elements.

These steps are explained in the following sections.

2.3.1 Selecting the Type of Isometric

To produce a Standard isometric, select **Isometrics>Standard** from the main menu bar. The **Standard Isometric** form is displayed.

2.3.2 Selecting the Options File

Options files can be created in three areas: Project, Company or Local. The files are accessed by selecting the required area from the **Options** drop-down list on the **Standard Isometrics** form. The names of the files in the area are displayed in the options scrollable list:

- The Project standards: the files containing standard settings for the project that you are in. There are some example files supplied with ISODRAFT.
- The Company standards: the files containing standard settings for your company. Two metric sample files (BASIC.MET and ADVANCED.MET) and two imperial sample files (BASIC.IMP and ADVANCED.IMP) are supplied with ISODRAFT.
- Local lists the contents of the current directory.

Click on the options file you wish to use, to select it.

2.3.3 Selecting the Elements

You can elect to process either the Current Element and its members, or the contents of the Detail List, by selecting **Current Element** or **Detail List** from the **Detail** drop-down list on the **Standard Isometrics** form.

If you select **Current Element**, you must select the CE in the **ISODRAFT Explorer** or the **ISODRAFT Members** list. If the CE is a Zone, Group or ISOREG, the owned (or grouped) Pipes or Spool Drawings are processed.

Assembling a 'Detail List' allows you to produce isometrics for several elements, which need not be in the same area of the database (or even in the same database), in one operation. To display the **Isometric Detail List** form, select **Detail List** from the **Detail** drop-down list.

Note: This replaces the normal **ISODRAFT Members** list.

The **Isometric Detail List** form contains two scrollable windows: one lists the Members in the selected database, in the same way as the **ISODRAFT Members** list; the other lists elements added to the Detail List.

To add an element to the Detail List, select it in the Members List and click the **Add** button.

Only Zones, Groups, Pipes, Branches, ISOREGs or Spool Drawings can be added. If the element is a Zone, Group or ISOREG, the owned (or grouped) Pipes or Spool Drawings are added rather than the element itself.

2.3.4 Selecting the Output

You can specify that an **Isometric**, **Transfer file**, **Batch macro** or **Bolt report** is produced. Select the required option from the **Produce** drop-down list on the **Standard Isometrics** form.

2.3.5 Detailing the Selected Elements

To detail the selected element(s), click the **Apply** button.

A OUTFITTING plot file for the isometric will be created in the directory specified in the options file. A material list file will also be created if material list production is switched on in the options file.

When detailing is complete, the display will change to show the isometric displayed in the **Display Isometric** window. Two other forms, **ISODRAFT Message** and **Display List**, are also displayed.

2.3.6 Example of Detailing the CE Using the GUI

The most basic method of using ISODRAFT is to detail the current element using one of the sets of options provided with the software. To detail an element:

1. Navigate to a pipe in the OUTFITTING DESIGN database, or a Spool Drawing in the Fabrication database using the **ISODRAFT Explorer** or the **ISODRAFT Members List**.
2. Select **Isometrics>Standard** from the main menu bar. This displays the **Standard Isometric** form, with the current element shown at the top.

Note: The element named next to the **CE** button will be detailed. If you select a *new* current element from the members list you must then click on the **CE** button.

3. Select **Company** from the options drop-down list and select **Basic.Met** from the list of options in the scrollable list box.
4. Ensure that **Current Element** and **Isometrics** are selected in the **Detail** and **Produce** drop-down list boxes, respectively.
5. Click the **Apply** button to start the detailing.

When detailing is finished, the PLOT file is displayed on your screen in the **Display Isometric** form, together with an **ISODRAFT Messages** form showing the name of the file produced, and a **Display List** form showing all the available plots.

2.3.7 Creating and Modifying Options Files

The options files supplied with ISODRAFT can be edited, or new options files created, using the functions available from the **Options** menu. This displays the **Options** form giving access to a number of forms grouping the options logically.

2.4 Controlling ISODRAFT from the Command Line

If you wish to control ISODRAFT from the command line, display the **ISODRAFT Command Input & Output** form by selecting **Display>Command Line** from the main menu bar.

Full details of all the commands are given in the Command Reference chapter, of the *ISODRAFT Reference Manual*. Commands related to the descriptions in this and following chapters are summarised at the end of each chapter.

The information displayed on each isometric can be controlled by enabling and disabling features from the command line. The required settings are normally established by the system administrator and files are plotted using macros.

The basic procedure for producing an isometric plot from the command line is:

1. Set-up the output filename and mode.
2. Enter the Message File name.

Note: Before any pipelines can be processed for isometric plotting you *must* specify an output device and a message file.

3. Set-up any additional options you wish to use.
4. Assemble a Detail List, if required.
5. Plot the required elements, using the DETAIL command.

2.4.1 Output Filename and Mode

The isometrics created by ISODRAFT are sent to an output file; they can then be displayed on screen or plotted. Screen display is handled by the **View Plotfile** function, available from the **Display** menu in ISODRAFT.

Note: External programs can also be used to view and plot the file.

- **Output Filename**

The output filename for the isometric plots is specified by the command:

```
FILEname filename
```

Each plot file will be given the name specified by **FILE**name, with the suffix 001, 002, etc. to make each one unique (e.g. *filename001*). The filenames used are noted in the message file.

A separate output file is created for each DETAIL command. (DETAIL is the command which starts processing the data.)

For example, the commands

```
DETAIL /PIPE1
```

```
DETAIL /PIPE2
```

will send the isometrics for /PIPE1 and /PIPE2 to the files /ISOFILE001 and /ISOFILE002, respectively.

The command

```
DETAIL /PIPE1 /PIPE2
```

will send the isometrics for both PIPE1 and PIPE2 to the file /ISOFILE001.

- **Output File Mode**

If there is more than one Pipe specified in the DETAIL command, or if the isometric for a single Pipe is so complex that it is split over more than one drawing, the plot file will contain several drawings: this is known as a *multiple* plot file. You can specify that each drawing is sent to an individual plot file by giving the command

```
FILE filename SINGLE
```

To change back to multiple file mode, give the command

```
FILE filename MULTIPLE
```

In multiple file mode you can specify the maximum paper length which the plotter can handle. Use the command

```
FILE filename MULTIPLE val
```

where *val* is the maximum length of the plotter's paper roll in millimetres. ISODRAFT will then put as many drawings as possible into each plot file. The OUTFITTING limitation on plotting size is 3275mm²; this cannot be exceeded in either single or multiple file mode.

By default, the plots will be arranged one after the other with their long sides parallel to the length of the paper. In multiple plot mode, you can specify how plots are arranged on the paper by using the STACKINGARRANGEMENT command.

Note: You cannot stack drawings in single file mode.

- **Querying the Output Device**

The current output file and mode can be queried at any time by using the command

Query **OUTPut**

Typical messages you may see are:

```
PLOT FILE PREFIX = /filename SINGLE
PLOT FILE = /filename MULTIPLE PAPER LENGTH 1200
```

2.4.2 Message File

ISODRAFT outputs a report of pipes detailed and drawings produced, together with any problems found when the pipes were processed. The file to which this information is to be sent is specified by the command

MESSAgefilename *filename*

At the end of an ISODRAFT run this file will contain:

- the references of all pipes which have been processed;
- the references of any pipes which have been rejected and the reason for rejection;
- advisory messages, such as item codes being truncated;
- the name of the plot file in which each drawing of each pipe will be stored.

Typical messages showing errors in the design are as follows:

```
ISODRAFT MESSAGE FILE 22 Feb 2000
-----
(33:194) STARTING TO DETAIL PIPE /PIPE2
(33:168) Cannot find FLANGE to match PPOINT 3
        of VTWA /PIPE2/VTWA-1
(33:168) Cannot find FLANGE to match PPOINT 4
        of VFWA 1 of BRAN /PIPE2-1
ISODRAW Mk11.3 (WINDOWS-NT 4.0) ( 20 Feb 2000 : 23:22)
Run on Tues, 22 Feb 1992 14:30
The following drawings are in plotfile plot006
      Drg.    1    /PIPE2
              Plotted
```

Any errors reported in the message file should be corrected and the pipes for which errors were reported should be run through ISODRAFT again.

The message file name can be queried by using the command

Query **MESSAgefilename**

2.4.3 Assembling a Detail List

Assembling a 'Detail List' allows you to produce isometrics for several elements that are not in the same area of the database (or even in the same database), in one operation. You can use the ADD and REMOVE commands to specify which elements you wish to place in the

Detail List for processing. These commands work in different ways for OUTFITTING DESIGN or Fabrication database elements, as explained below.

Note: OUTFITTING DESIGN and Fabrication database elements cannot be mixed in a system isometric drawing.

- **Adding OUTFITTING DESIGN Database Elements to the Detail list**

Elements are placed in the Detail List using the **ADD** command. If a high level element (Pipe or above) is added to the list, any member elements which ISODRAFT is to ignore should be placed in the Remove List.

The commands used to create these lists are:

```
ADD  gids
REMOve  gids
```

where *gids* are the general identifiers defining the elements.

For example, a Zone containing many Pipes could be put into the Add List with the command

```
ADD  /ZONE-A
```

and the Pipes for which isometric plots are not required could be put into the Remove List with the command

```
REMOVE  /PIPE100  /PIPE120  /PIPE130  ...
```

Note: The Add and Remove Lists can each contain up to 100 entries.

If a pipe is added to the Add List more than once, it will be processed as many times as it appears in the list. However, if that pipe is in the Remove List at least once, it will not be processed at all.

To clear both the Detail and Remove Lists, enter the command

```
DEtail  Empty
```

Alternatively, clear just the Remove List by entering the command

```
REMOve  Empty
```

Note: There is no equivalent command to clear the Add List only, since it is meaningless to have entries in the Remove List while the Add List is empty.

- **Adding Fabrication Database Elements to the Detail List**

The Detail and Remove Lists are used differently for Fabrication database elements. Elements, from Spools and Fields up to the complete database World /*, can be added to the Detail List by using the ADD command. Individual Spools or Fields are shown as 'Spool/Field x of Spool Drawing y'; while Spool Drawings or above are shown as a list of Spool Drawings.

For example:

```
Add  /ISOREG-1
```

will add all the Spool Drawings members of that element to the Detail List.

When you wish to remove some elements from the Detail List, the specified Spool, Field or Spool Drawing(s) are deleted from the Detail List using the REMove command. The Remove List stays permanently empty when working with the Fabrication database.

Note: You cannot remove individual Spools or Fields from a Spool Drawing

- **Querying the Detail and Remove Lists**

The contents of the Add and Remove Lists can be queried at any time by using the command

Query DETAILLIST

You can use the RECREATE DETAILLIST command to save the Detail List to a macro file, which can then be read in again when required. See the [Software Customisation Guide](#) for information on using macros.

2.4.4 Isometric Detailing Commands

The DETAIL or CHECK commands, followed by the pipes to be detailed, start the data processing.

- **DETAIL** causes ISODRAFT to process the pipeline elements and to send the resulting isometric drawings and material lists to a file.
- **CHECK** causes ISODRAFT to process the pipeline elements without producing a plot file. This can be useful as a check that a complex pipeline will be successfully drawn before batch mode plotting.

Note: The explanations which follow, which refer to the DETAIL command, also apply to the corresponding CHECK commands.

The lowest level elements that can be processed are Branches in the OUTFITTING DESIGN database or Spools and Fields in the Fabrication database. An attempt to process a lower level element will produce an error message. Higher level elements, such as Pipe, Site or Zone, and group elements, such as Group or GPWLD, can be specified, in which case ISODRAFT will detail all Pipes owned by them (unless Pipes have been removed from the list using the REMOVE command).

The elements to be detailed can be specified in 3 ways:

- By giving their OUTFITTING identifiers (i.e. the names or reference numbers of the elements)

For example, the commands:

DETAIL /PIPE2

DETAIL /ZONE3 /ZONE4 /ZONE5

DETAIL /GROUP1

will detail the pipe /PIPE2 only, all pipes in Zones /ZONE3, /ZONE4 and /ZONE5, and all pipes in the group element /GROUP1, respectively.

- By adding them to the **Detail List** (see [Assembling a Detail List](#)), then using the command:

DETAIL ALL

which causes all elements in the Add List (and their members) to be detailed, except those specified in the Remove List.

- By selecting them as the current element and then detailing the CE.
The command

DEtail CE

will detail all pipes owned by the current element.

See the DETAIL command for how to plot a complete system isometric and equipment trims.

Note: ISODRAFT will ignore all Equipment and Structural elements included in the selected elements and any elements owned by them.

2.4.5 Example of Detailing Using ISODRAFT Commands

Assume that a Project DB includes the MDB /TESTSITE which, in turn, has access to a OUTFITTING DESIGN database named PIPING/AREA-A and a correctly set up Catalogue DB. This has a structure as shown in [Figure 2:1.: Example database hierarchy](#).

The following sequence of commands could be used to create isometric drawings, in the default format, of the two pipes /PIPE-Y and /PIPE-Z. The example assumes that data consistency checking in the OUTFITTING DESIGN database finds no faults.

/TESTSITE	Specify MDB
OUTFITTING DESIGN	Enter OUTFITTING DESIGN, which has data consistency checking facilities
CHECK /PIPE-Y / PIPE-Z	Check data consistency of pipes

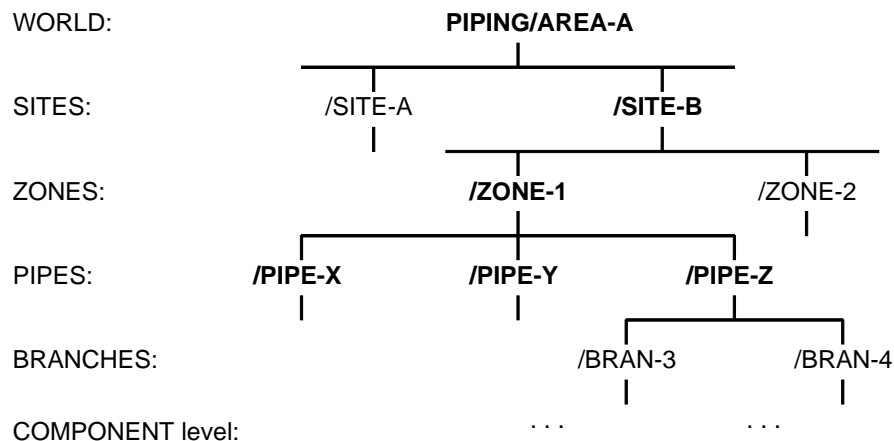


Figure 2:1. Example database hierarchy

A data consistency report is generated: a report of no errors is assumed

ISODRAFT Enter ISODRAFT module

FILE / Specify ISODRAFT output device
ISOFILE SINGLE

The two pipe drawings to be created will be sent to separate plot files, identified as / ISOFILE001 and /ISOFILE002

MESSAGEFILENAME / Specify filename for messages
ISOMESS

QUERY OPTIONS Check the current settings of all ISODRAFT plotting options

ADD /ZONE-1 Set up list of pipes to be drawn by ISODRAFT

REMOVE /PIPE-X

DETAIL ALL Draw isometrics of required pipes

The last three commands could be replaced by:

DETAIL /PIPE-Y /PIPE-Z

2.4.6 Isometric Output Format

The general format of a typical ISODRAFT output drawing is shown in [Figure 2:2.: Typical isometric drawing sheet layout](#). A range of examples, showing some of the different options, are shown in Appendix *Examples of Isometrics* of the [ISODRAFT Reference Manual](#).

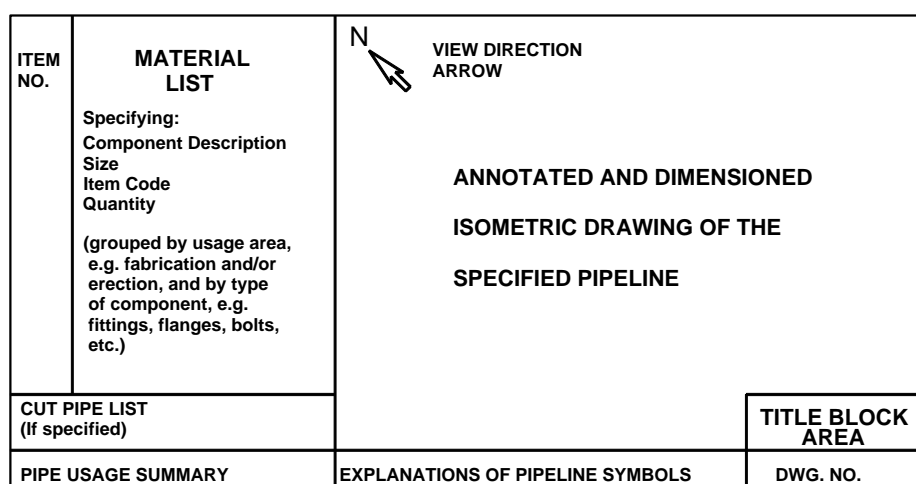


Figure 2:2. Typical isometric drawing sheet layout

2.5 Querying

You can use the QUERY command to query all ISODRAFT variables, and also to query references and other attributes of elements in the Catalogue and OUTFITTING DESIGN databases which you may need to know in ISODRAFT.

This can also be done in the GUI, by selecting **Query>General** from the main menu bar and then selecting the required options from the displayed Query form.

The current setting of any of the user-definable options can be queried by using the command

```
Query setting
```

For example:

```
Q SIZE
Q WASTage
Q MESSAGEfilename
Q ITEMcode
```

The general querying command

```
Q OPTions
```

will display the current settings of *all* options.

2.6 Option Defaults

Each of the available options has a default setting which will be used by ISODRAFT unless otherwise specified.

The command

```
OPTions DEFault
```

will reset *all* options to their default states at any time. Note that you can find out which options have been changed from their defaults by using the QUERY command. You can save the current options to a macro file using the RECREATE OPTIONS command. See the *Software Customisation Guide*, for information on using macros.

An isometric drawing with all options set to their default states is shown in Appendix *Examples of Isometrics* of the [ISODRAFT Reference Manual](#).

2.7 MDB Mode

You can change the current multiple database, and also the current User and Project during an ISODRAFT session without having to leave ISODRAFT and enter MONITOR. The MDB command puts you into **MDB Mode**, where you can use a limited number of MONITOR commands.

You can either update the current MDB to save your changes before entering MDB Mode, or ignore any changes made since your last SAVEWORK command (see [Saving](#)).

MDB UPDATE Save design changes and enter MDB Mode.

MDB NOUPDATE Enter MDB Mode without saving changes.

When you are in MDB mode, you can give the following commands, which are the same as the corresponding MONITOR commands. For more information, see the [MONITOR Reference Manual](#).

EXCHANGE DEFER Alters the databases in the current list of the current MDB
CURRENT

PROTECT Temporarily alters your access rights to specified databases

USER PROJECT Changes the current user and project

VAR Allows you to set variables

QUERY Allows you to query:

Users, including the number of active users, Teams including the set (current) Team, Databases, including copied Databases, MDBs, Macros and Variables

To leave MDB mode and return to normal ISODRAFT mode, give the command EXIT.

2.8 Saving

2.8.1 Saving and Restoring the Current Display Status

You can save and restore the display set-up (including the full forms and menus set) by using the RECREATE and INSTALL commands.

For example:

RECREATE /DS1 Saves the display status in file /DS1.

RECREATE /
DS1 OVER Saves the display status in file /DS1. Any existing file /DS1 is overwritten

RECRE DISPLAY Saves modal settings. Read back in using \$M.
/DS2

INSTALL SETUP Restores the display definition stored in file /DS1. (Refers to file saved by RECREATE, *not* RECREATE DISPLAY.)
/DS1

Note: Forms resized or moved using the cursor will be INSTALLED to their original size.

2.8.2 Saving Work

The command

SAVEWORK

saves the current ISODRAFT additions or modifications without leaving ISODRAFT. It is good practice to use this command regularly during an ISODRAFT session.

2.8.3 Seeing Changes Made by Other Users

The command

GETWORK

updates the ISODRAFT database with the changes made by other users, if the database has been opened in multi-write mode.

2.9 General OUTFITTING Facilities

This section describes some commands which are available in many OUTFITTING modules, including ISODRAFT. Only a brief summary is given here. See the [MONITOR Reference Manual](#) for full details.

2.9.1 Finding the Current User Status

The **STATUS** command gives you information about your current user status and that of the DBs to which you have access.

2.9.2 Finding the Current System Status

The **SYSTAT** command gives you information about the current active status of the project in which you are working. It lists all users who are currently accessing the project, the modules and databases which they are using, and whether they are examining (Read-only status) or modifying (Read/Write status) the database.

2.9.3 Listing Multiple-Database Information

The **LIST** command allows you to list most of the available project information held in the System Database (with the exception of confidential details such as other users' passwords, which can only be listed by the Project Co-ordinator using the ADMIN module of OUTFITTING). There are also forms of the **QUERY** command which display information about the project configuration.

2.10 Summary of Commands

The following commands are described in this chapter:

ADD	Adds named elements to the list of elements to be processed by the CHECK or DETAIL command.
CHECK	Extracts isometric data from the database to create a transfer file.
DETAIL	Generates and plots isometric drawings for the elements in the list.

FILE	Specifies the name of the file to which the isometric drawing output is sent.
INSTALLSETUP	Restores the settings to those saved in the macro or binary file using the RECREATE command.
MESSAGEFILENAME	Specifies the name of the file to which messages about the progress of the detailing are sent.
OPTIONS	Allows you to reset all options to their default settings.
QUERY	Allows you to find the current settings of the options.
RECREATE	Creates a macro or binary file which can be read into ISODRAFT to restore the current settings.
REMOVE	Removes elements from the list to be detailed.

3 Cataloguing Requirements

Most of the reference material needed by ISODRAFT is stored in the Catalogue database. Such data includes:

- Component item codes and descriptions, to be printed on material lists
- A definition of the two-dimensional symbol (SKEY) for each type of component
- A definition of any specific types of end connections to be associated with particular components
- The bolting requirements of all flanged components

ISODRAFT cannot function correctly unless the Catalogue is properly set up. This information supplements the *Catalogues and Specifications Reference Manual*.

This chapter describes how the data, except for bolting information, is stored in the Catalogue. How to set up the Bolting part of the Catalogue is described in *Bolting*.

3.1 Component Descriptions

The texts used to describe components in the material list are stored in the Catalogue DB as **text elements**. The relevant part of the DB's hierarchy is:

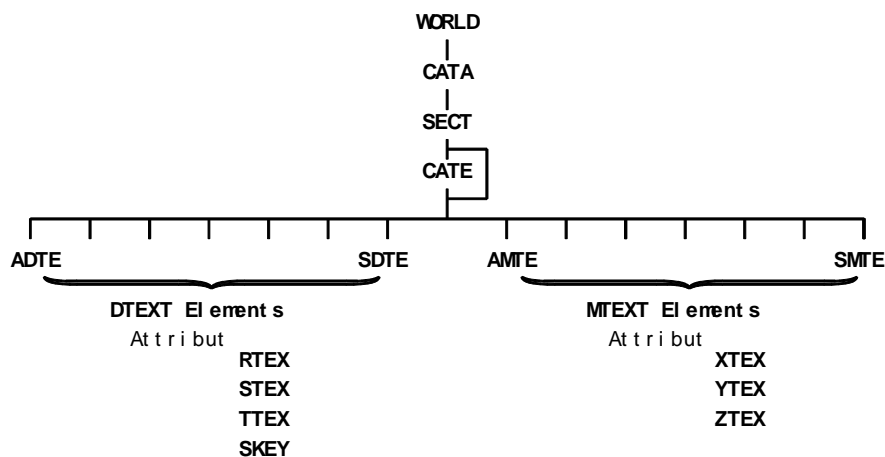


Figure 3.1. Storing text elements in the DB

where: ADTE to SDTE are **detail text** (DTEXT) elements and AMTE to SMTE are **material text** (MTEXT) elements.

Note: Both sets of text elements normally exist at the same level, as shown, although the DTEXT and MTEXT elements will often be in different sections (SECT) or categories (CATE).

Text elements are created as described in the [Catalogues and Specifications Reference Manual](#). For example:

```
NEW DTEXT 17
NEW MTEXT 17
NEW BMTE
```

Each component description is split into two (optionally three) parts:

- The **geometric description**, e.g. 90 degree butt weld elbow
- The **material description**, e.g. ASTM A-234
- Optional **attached text**, e.g. UDA and/or other attribute settings (defined by the ATTEXTS command)

ISODRAFT will assume that the RTEXT attribute of a DTEXT element contains the geometric description and that the XTEXT attribute of an MTEXT element contains the material description.

Thus the material list description

ELBOW 90 BUTT WELD SCH.40 ASTM A105 FORGED	would be obtained from a DTEXT element whose RTEXT attribute is
ELBOW 90 - BUTT WELD SCH.40	and an MTEXT element whose XTEXT attribute is
ASTM A105 FORGED	New lines (i.e. Returns) in RTEXT and XTEXT are ignored.

Note: The values of the DTEXT attributes can be set to an expression, enabling the text to be parameterised. When the attribute is queried in ISODRAFT, the expression will be evaluated and the actual value will be output.

Attached text allows you to append text defined by an expression (typically the setting of one or more UDAs or other attributes) to the descriptions of elements whose GTYPs comply with a predefined selection rule. The syntax to define the text and its associated rule is

ATTEXTs *expression* **ON** *selection_rule*

Example:

```
ATTEXT ('Flange '+AFTER (NAME,'/')+' at position '+STR (POS)+' has '+STR  
(NWELDS)+' additional welds') ON all flange with (NWELDS GT 0)  
ATTEXT ('Colour: ' + :Colour) ON ALL BRANCH MEMBERS
```

The description shown on the isometric will be RTEXT + XTEXT + ATTEXT. The ATTEXT syntax may be used as often as required, each expression being evaluated and appended to the description whenever the selection rule returns a True result.

The command ATTEXTS NONE cancels all current ATTEXT rules.

3.2 Symbol Keys

Each type of component has a two-dimensional symbol which is used to represent it on the isometric drawings. Each symbol is referred to by a text attribute, known as the **symbol key** (SKEY) attribute, of a DTEXT element.

The overall relationship between a pipe component and its descriptive elements is shown by the following structure:

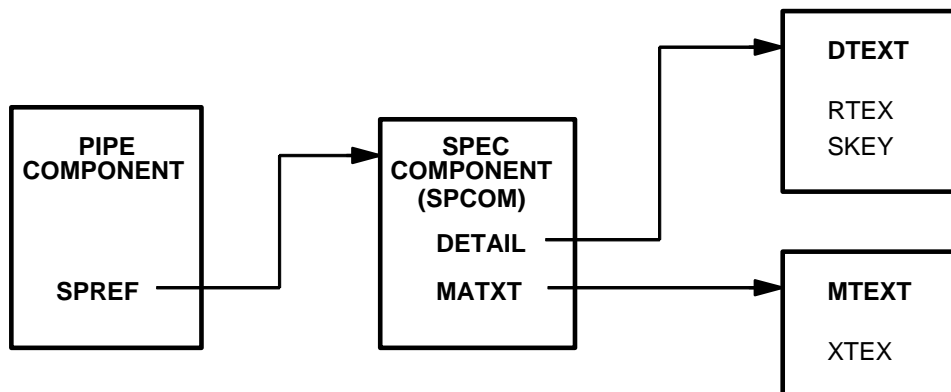


Figure 3.2. Symbol keys

The chapter *Symbol Keys* of the *ISODRAFT Reference Manual* contains a complete list of the standard symbols and their associated symbol keys, grouped together by generic type (ELBO, FLAN, PCOM, VALV, etc.).

You can usually select any symbol listed under the relevant generic type. For example, the following symbol keys are valid for a CAP, refer to section *CAP Symbol Keys* of the *ISODRAFT Reference Manual*:

KABW KACP KASC KASW

If an incorrect SKEY is specified, a default rectangle will be drawn at the corresponding component position.

Note: It is essential to comply with the p-point configurations for the given symbol. The chapter, *Symbol Keys* of the *ISODRAFT Reference Manual* shows the configuration for each symbol.

The SKEY attribute of a DTEXT element is set by using the syntax

SKEY *symbolkeytext*

where *symbolkeytext* is the symbol key attribute code enclosed between closing quotes. For example:

SKEY 'KABW'

The default SKEY symbols should be suitable for most applications. For specialised needs it is possible to redefine existing symbols, or to create new symbols, for some types of component. This is described in the chapter *Symbol Keys*.

3.3 Component End Conditions Defined By P-Points

The symbols used to represent piping components on isometrics, as defined by the SKEYs described in the section [Symbol Keys](#), can be drawn with different end conditions (i.e. with different connection types). The connection types which may be represented are:

BW	Butt Weld	-----
CP	Compression	—□-----
FL	Flanged	—□○-----
SC	Screwed	—□-----
SW	Socket Weld	—□-----
PL	Plain	-----

Figure 3:3. Component end conditions

By default, each SKEY has one standard end condition which applies to all its connection points. A different end condition can be set for individual connection points for the following types of components:

BEND	CROS	ELBO	FILT	FLAN
INST	OLET	PCOM	REDU	SHU
TEE	TRAP	VALV	VFWA	VTWA

P-point elements of type PTAXI, PTCAR and PTMIX have a PSKEY (P-point Symbol Key) attribute which is, by default, set to NULL, so that ISODRAFT uses the standard end connections defined for that SKEY.

The end condition for a connection point may be changed in PARAGON by setting the PSKEY of the corresponding p-point to one of connection end conditions shown above. To revert to the default end condition representations for a component, reset its PSKEY attributes to NULL.

3.4 Alternative Item Codes

By default, ISODRAFT takes the item code from the SPCOM name. If you require different item codes for different clients, you can set them in the Catalogue database as User-defined Attributes (UDAs) of the SPCOM. UDAs are described in the [LEXICON Reference Manual](#). See the ISODRAFT command ITEMCODE for details of how to display the item codes stored as UDAs.

The following example shows part of a specification. The SPCOMs have UDAs :clienta and :clientb. Note that SPECON cannot output UDAs in tabular format, and so this example is just for illustration.

TYPE	NAME	...	BLTREF	:clienta	:clientb
FLAN	*/FL100	...	/BL100	F100L	FLAN100
BOLT	*/BL100	...	/BL100	unset	unset

Note: ITEMCODE LENGTH LONG/SHORT has no effect on alternative item codes. Also note that any text after a delimiter character in a UDA will be stripped off (for example, ABC:123 would become ABC).
Alternative item codes for bolts are set in a different way, as described in [Alternative Item codes for Bolts](#).

3.5 SHOP Flag

ISODRAFT divides items in the Material List into **Fabrication** and **Erection** components. It does this by looking at the SHOP flag of the component: if the SHOP flag is set to TRUE, ISODRAFT lists the component as a Fabrication item; if it is set to FALSE, ISODRAFT lists the component as an Erection item.

Although you can set the SHOP flag specifically in the OUTFITTING DESIGN or SPOOLER modules by using the SHOP TRUE or SHOP FALSE command, it is more usual to set a SHOP selector in the Specification so that the flag is set automatically when the component is selected. The default, if the SHOP flag is unset, is SHOP FALSE.

Consider, for example, the following Specification.

Note: The default for SHOP is shown as =, meaning that the default setting is taken from the first SPCOM which satisfies all other selectors.

HEADING NAME DEFAULTS	TYPE	PBOR0	ANGL	SHOP	CATREF	DETAIL	MATXT
-	-	-	90.0	=			
*/EA65	ELBO	65.0	90.0	TRUE	/AAEA200KK	/AAEA200-D	/A6B
*/EC65	ELBO	65.0	45.0	TRUE	/AAEC200KK	/AAEC200-D	/A6B
*/EA80	ELBO	80.0	90.0	TRUE	/AAEA200LL	/AAEA200-D	/A6B
*/EC80	ELBO	80.0	45.0	TRUE	/AAEC200LL	/AAEC200-D	/A6B

It is usual for all items except those such as Bolts, Gaskets, Valves, etc. to be designated Fabrication items; i.e. to have the SHOP flag set to TRUE.

If you need to make an exception to this convention, for example, to specify a loose flange for field fitting, you can override the SHOP flag during the component selection process thus:

```
NEW FLAN SEL WITH SHOP FALS
```

If you wish to use the same component with SHOP either TRUE or FALSE, you can have two lines in the Specification (i.e. two SPCOMs) with the *same* basic Name but with distinguishing delimiters; for example, */FL100:T and */FL100:F. ISODRAFT ignores the delimiter and shows the same item code for each flange in the material list, i.e. /FL100.

3.5.1 Tube SHOP Flag

ISODRAFT takes the SHOP flag setting for tube from the specification. If the SHOP flag is not specified, the default is FALSE. It is most probable that you will want small bore tube to be treated as an erection item (SHOP FALSE) and larger bore tube to be treated as a fabrication item (SHOP TRUE). It is also important that spool isometrics have tube with SHOP TRUE.

To access tubes with the same bore, schedule, material, etc., yet with SHOP TRUE at one point and FALSE at another, it is necessary to have *two* tube SPCOMs. These will have identical lines in the specification except for their SHOP column entries and their names. You do not want to have two different item codes for the same tube on the isometric, so the names should be identical up to a suffix delimiter (a colon ':' by default). ISODRAFT will ignore the suffixes and the items will be totalled together on the Material list.

Example:

TYPE	NAME	PBOR0	SHOP	CATREF	DETAIL	MATXT	CMPPREF	BLTREF
DEFAULT								
-	-	-	=					
TUBE	*/TU100:TR	100	TRUE	/TUAAA	/D.TUA	/M.TUA	=0	=0
TUBE	*/TU100:FA	100	FALS	/TUAAA	/D.TUA	/M.TUA	=0	=0

In this example, the first tube SPCOM will be selected by default and this has SHOP TRUE. To select the second SPCOM, with SHOP FALSE, you need to use one of the following commands in your design module:

```
SEL LSTU WITH SHOP FALS
SEL HSTU WITH SHOP FALS
SEL TUBE WITH SHOP FALS
```

If the lines in the specification were reversed, then tube would be selected with SHOP FALSE by default and would need to be selected specifically with SHOP TRUE for spool isometrics. It is recommended, therefore, that you arrange the lines for tubes with SHOP FALSE first for small bores and SHOP TRUE first for large bores.

4 Bolting

This chapter describes how to set up Bolting information in the Catalogue. For more information about other aspects of setting up the Catalogue for ISODRAFT, see [Cataloguing Requirements](#).

4.1 How Bolts are Stored in the Catalogue

In order to ensure that the correct quantities and specifications of bolts are called up in the material lists, the bolting requirements of all flanged components are held in the Catalogue DB. This information is stored in **Bolt Sets (BTSE)** and **Bolt Tables (BLTAB)**, which are positioned in the hierarchy as follows:

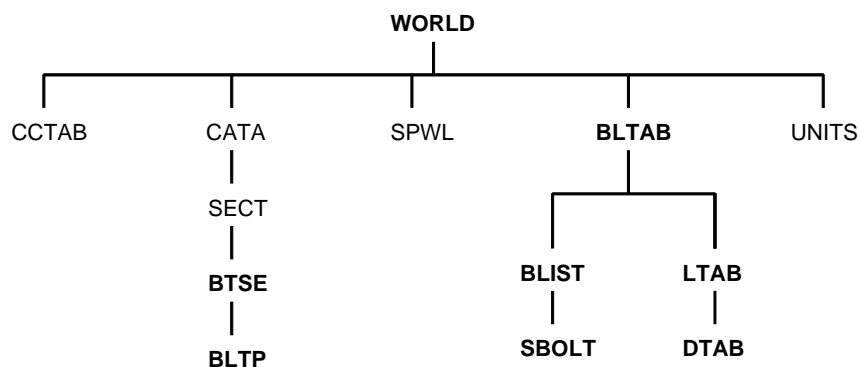


Figure 4.1. Bolt Table Hierarchy

4.2 Bolt Sets (BTSE)

A **BTSE** (Bolt Set) element is the administrative element for catalogue component bolting information. It can also be used to store information for the standard bolt holes.

The standard bolt holes are numbered from 1 to NOFF.

It owns **BLTP** (Bolt Point) elements, which store the bolting information for an individual bolt for a particular type of flange.

BLTP should be used for non standard bolt holes and also specifically for jacking bolts. Existing projects which do not use standard bolts will carry on working without problems.

Catalogue components can have a Bolt Reference (BLRF) array attribute which points to one or more Bolt Set elements. If a Component has the same bolting requirements for each

connection, a single entry in the Bolt Reference array will be sufficient. Otherwise, an entry for each P-point which represents a bolted connection is needed.

Bolt Sets have the following attributes:

NOFF	Number of standard bolt holes in the bolt circle.
BDIA	Bolt diameter.
BTHK	Bolt length through flange (flange thickness and facing thickness).
BTYPE	The Bolt type, which is used to select the Bolt from the specification, and also affects how the bolt length is calculated.

Bolt Points have the following attributes:

NUMB	Bolt hole number in the bolt circle.
BDIA	Bolt diameter.
BTHK	Bolt length through flange (flange thickness and facing thickness).
BTYPE	The Bolt type, which is used to select the Bolt from the specification, and also affects how the bolt length is calculated.

The BDIA, BTHK and BTYPE attributes of Bolt Points can all be set from Catalogue Component or Design parameters.

4.2.1 Number Attribute (NUMB)

All bolts must have a number, which is stored in the NUMB attribute of the Bolt Point. Bolts must be numbered in order around the bolt circle, except for jacking bolts, which will have NUMB set to zero. Thus bolt number one on a flange will correspond to bolt number one on the connecting valve, and so on (see [Figure 4.4.: Example Data Flow Diagram for a Wafer Lug Valve](#)). The bolt order must be symmetrical, so that two like flanges can be bolted together when one is reversed.

Jacking bolts do not require a matching Bolt Point number on the mating flange.

4.2.2 Bolt Length Attribute (BTHK)

The BTHK (Bolt length) attribute is used as follows:

- Components with blind holes have the tapping depth assigned to their BTHK attributes.
- Jacking bolts have their BTHK set to the sum of the flange/facing thickness and the required opening gap.
- Wafer components with no special bolting needs do not require a Bolt Pointset. If supplied with a Bolt Pointset, a BTHK value of zero indicates that the face-to-face dimension of the wafer component is included in the bolt length calculation.
- Wafer components with special bolting needs require a Bolt Pointset, with BTHK values assigned only to the respective Bolt Points affected. For example, a wafer lug valve with four bolted through holes and four tapings for cap screws on each side requires eight Bolt Points. The four bolted through numbers have their BTHK set to 0. The cap screw numbers have their BTHK set to the tapping depth. See [Figure 4.3.: Example Wafer Lug Valve](#).

4.3 Bolt Tables

The **BLTAB** (Bolt table) element is the administrative element for bolting information selected from the bolt specifications. It can own **BLIST** (Bolt List) and **LTAB** (Length Table) elements.

- The **BLIST** (bolt list) element is the administrative element which owns **SBOLT** (single bolt) elements.
- Each **SBOLT** has the following attributes:

NST	A pointer to a standard length array
BITEM	Additional bolt items to be used when calculating bolt length
BITL	The lengths of the additional bolt items

- The **LTAB** (length table) element is the administrative element which owns **DTAB** (diameter table) elements.
- Each **DTAB** element has a **BLen** (bolt length array) attribute containing all available lengths of bolts for a given diameter and material specification. This data is used by ISODRAFT for calculating bolt length requirements. When ISODRAFT calculates that a bolted connection requires a bolt length, it scans the **BLen** array for a length which is equal to or greater than the calculated length. The length found is output in the material list.

Note: Any associated nuts, washers and other bolting items can be included in the material list by adding them to the bolt detail and material descriptions in the catalogue definitions. Nuts and washers can be included in bolt length calculations as described in [Additional Bolting Items in Bolt Length Calculations](#).

4.4 Bolt Specifications

Pipe Specifications have a reference attribute **BSPEC**, which can be set to a specification containing bolts. Bolts will be selected from the Bolt Specification, if the **BSPEC** is set, or from the Piping Specification of the flanged component, if it is not set.

For bolting across Specification breaks, bolts will be selected from the second Specification if they cannot be selected from the first Specification. The first Specification is defined as the gasket Specification if a gasket is present, otherwise it is defined as the downstream Specification.

In the event that bolts are included in vendor supply, for example, an orifice flange assembly, a null **BLTREF** or **CATREF** indicates that no bolts will be output for that joint.

4.4.1 Bolt Selection from Specification

Each SBOLT element in the catalogue is referenced by an SPCOM in the Bolt Specification. Bolts are selected individually for each BLTP from the Specification for the appropriate Bolt Diameter and Bolt Type (BTYP). Examples of valid BTYPs are:

MACH	machine bolts
STU	stud bolts
CAP	cap screws
JACK	jacking screws
TAP	tapped holes

A BTYP of BOLT will also be used as a pointer to the default bolt type, set in the specification. This can easily be set by altering the value for BTYP in the DEFAULT column of the specification. This implies that a separate line for each required BTYP must be included in the Specification for each size of bolt.

The way the selection works is explained in the following rules:

- If the BTYP for both sides of a joint is set to BOLT, the default bolt of the given diameter is used.
- If the BTYP for one side of the joint is set to BOLT and the other side is set to anything other than BOLT, the type of bolt specified by that BTYP will be used.
For example, a bolted through flange may have its BTYP set to BOLT, and the mating flange may require cap screws and have its BTYP set to CAP. In this case, the cap screw overrides the default type, and the selected BTYP will be CAP.
- If both BTYPs are the same and are not set to BOLT, the bolt with that BTYP is selected.
- If the BTYPs are not equal and are also not set to BOLT, the BTYP from the start of the bolt is selected.

4.4.2 Bolts SHOP Flag

ISODRAFT takes the SHOP flag setting for bolts from the specification. The BOLT SPCOM should be defined with a SHOP selector column. If the SHOP flag is not specified, its setting defaults to FALSE. If the same bolt is to appear in both fabrication and erection lists, two SPCOMs must be defined which are the same up to a suffix delimiter, in the same way as for tube. Also, two flanges must be defined using the delimiter character, in order to reference the two bolts. The flanges will be totalled together if they are to appear in the same section of the material list.

Example:

TYPE	NAME	PBOR0	SHOP	CATREF	DETAIL	MATXT	CMPREF	BLTREF
DEFAULT								
-	-	-	=	=				
FLAN	*/FW100:TR	100	TRUE	/FWAAA	/D.FWA	/M.FWA	=0	/SBOL100T

FLAN	*/FW100:FA	100	FALS	/FWAAA	/D.FWA	/M.FWA	=0	/SBOL100F
. . .								
TYPE	NAME	PBOR0	SHOP	CATREF	DETAIL	MATXT	CMPREF	BLTREF
DEFAULT								
-	-	-	=	=				
BOLT	*/BL100:TR	100	TRUE	/BLAAA	/D.BLA	/M.BLA	=0	/SBOL100T
BOLT	*/BL100:F	100	FALS	/BLAAA	/D.BLA	/M.BLA	=0	/SBOL100F

This example will give bolts in the same part of the material list (fabrication or erection) as the flange with SHOP TRUE or SHOP FALSE (default TRUE in this example).

The following example will give bolts in either list but total the flanges together in the fabrication list, by selecting the flange with BOLT SHOP or BOLT SITE. Note that item codes that are identical after removal of the specification name and the characters following the delimiter will be totalled together.

Example:

TYPE	NAME	PBOR0	SHOP	BOLT	CATREF	DETAIL	MATXT	CMPREF	BLTREF
DEFAULT									
-	-	-	=	=					
FLAN	*/FW100:SH	100	TRUE	SHOP	/FWAAA	/D.FWA	/M.FWA	=0	/SBOL100T
FLAN	*/FW100:SI	100	TRUE	SITE	/FWAAA	/D.FWA	/M.FWA	=0	/SBOL100F
. . .									
TYPE	NAME	PBOR0	SHOP		CATREF	DETAIL	MATXT	CMPREF	BLTREF
DEFAULT									
-	-	-	=						
BOLT	*/BL100:TR	100	TRUE		/BLAAA	/D.BLA	/M.BLA	=0	/SBOL100T
BOLT	*/BL100:F	100	FALS		/BLAAA	/D.BLA	/M.BLA	=0	/SBOL100F

4.4.3 Additional Bolting Items

Each type of bolt can have further material associated with it, such as washers, nuts, locking nuts and isolation tubes. The BLTREFs of the bolt SPCOMs point to SBOLT elements. SBOLTs have the attribute BITEMS, which is a word array with up to 25 elements. Each word represents a BTYP selector for a further bolt SPCOM in the Bolt Specification, such as NUT, LNUT, WASH or ISOL. This generates a further selection of material from the Bolt Specification.

Consider the following example extracted for a single bolt diameter in a Bolt Specification.

Example:

TYPE	NAME	BDIA	BTYP	CATREF	DETAIL	MATXT	BLTREF
DEFAULT							
-	-	-	STUD				
BOLT	*/BOLTS:M	0.5/8	STUD	/BOLT1	/STUD	/A193-B7	/SS-FAMS-M
BOLT	*/NUT:M	0.5/8	NUT	/NUT1	/NUT	/A193-B7	=0
BOLT	*/WASH:M	0.5/8	WASH	/WASH1	/WASH	/A193-B7	=0

The SBOLT element holds the list of bolt item STYPs, their lengths, and the extension thread length. For example, the SBOLT given by /SS-FAMS-M may have the following attributes:

SBOL	/SS-FAMS-M				
BITEMS	NUT	WASH	WASH	Bolt item STYPs	
BLIST	0.1/2	0.1/16	0.1/16	Bolt item lengths	
XLEN	0.1/8	Extension thread length			

Each additional bolt item may have an effect on the calculated length of the bolt (see [Additional Bolting Items in Bolt Length Calculations](#)).

4.4.4 Selecting Bolts with Temperature or Pressure

The temperature and pressure attributes from the owning branch can be used as selectors for choosing bolts. You can set ranges of values by using a comma or you can give a specific value.

TYPE	NAME	BDIA	TEMP	BTYP	BSEL	CATREF	DETAIL	MATXT	CMPREF	BLTREF
DEFAULTS										
-	-	-	-	STUD	=					
BOLT	*/BOLTO-S:20	20.00	0,100	STUD	OUT	/BOLT	/STUD-D	=0	=0	/SBOL-STUD-20
BOLT	*/BOLT-C:20	20.00	0,100	CAP	IN	/CAP	/CAP-D	=0	=0	/SBOL-CAP-20
BOLT	*/BOLTO-C:20	20.00	0,100	CAP	OUT	/CAP	/CAP-D	=0	=0	/SBOL-CAP-20
BOLT	*/BOLT-W:20	20.00	0,100	WASH	IN	/WASH	/WASH-D	=0	=0	/SBOL-WASH-20

BOLT	* / BOLTO- W:20	20. 00	0,10 0	WASH	OUT	/WASH	/WASH- D	=0	=0	/ SBOL- WASH- 20
BOLT	* /BOLT- N:20	20. 00	0,10 0	NUT	IN	/NUT	/NUT-D	=0	=0	/ SBOL- NUT- 20
BOLT	* /BOLT- S:20HIG H	20. 00	101, 200	STUD	IN	/BOLT	/STUD- DHT	=0	=0	/ SBOL- STUD- 20
BOLT	* / BOLTO- S:20HIG H	20. 00	101, 200	STUD	OUT	/BOLT	/STUD- DHT	=0	=0	/ SBOL- STUD- 20
BOLT	* /BOLT- C:20HIG H	20. 00	101, 200	CAP	IN	/CAP	/CAP- DHT	=0	=0	/ SBOL- CAP- 20
BOLT	* / BOLTO- C:20HIG H	20. 00	101, 200	CAP	OUT	/CAP	/CAP- DHT	=0	=0	/ SBOL- CAP- 20
BOLT	* /BOLT- W:20HIG H	20. 00	101, 200	WASH	IN	/WASH	/WASH- DHT	=0	=0	/ SBOL- WASH- 20
BOLT	* / BOLTO- W:20HIG H	20. 00	101, 200	WASH	OUT	/WASH	/WASH- DHT	=0	=0	/ SBOL- WASH- 20

4.4.5 How Bolts are Selected from the Specification

The steps in selecting a bolt from the Specification are as follows:

- For each bolt in the set, get the BTYP values for each end of the bolt:
 - If they are both set to BOLT, select the default bolt for that bolt diameter.
 - If they match and are not set to BOLT, select the bolt with that BTYP.
 - If they do not match, and one value is set to BOLT, select the bolt with BTYP set to the other BTYP.
 - If they do not match and neither is set to BOLT, select the bolt with BTYP set to the BTYP of the start of the bolt.
- Set the bolt item code from the bolt SPCOM name, and the bolt description from its detail and material texts.
- Go to the SBOLT given by the BLTREF of the SPCOM:
 - Get the values of BTYPs for items such as nuts and washers from the BITEMS array.
 - Get the length values of these items from the BLIST array
 - Get the extension thread length from the XLEN attribute.

For each STYP selector:

- Select the item by its BTYP and the current bolt's BDIA. Get its item code from its SPCOM name and its description from its detail and material texts.
- Add its length to the accumulated bolt length.
- Add the extension thread length to the accumulated bolt length

7. Go to the NSTD length table. If the accumulated length is within 1mm over an available length from the BLEN array, then round down, else round up. If the NSTD reference is NULL, use the actual BLEN of the SBOLT, since this is intended as a pre-set length for special purposes, for example, a large flange with exact torque requirements.
8. Include the bolt length in the item code and description in accordance with the existing ISODRAFT options.

When the bolt member count has been satisfied, combine the data to obtain totals of like bolt items in the material list. See [Bolting Information](#). Any error conditions such as a bolt item cannot be selected, or there are no long enough lengths in the NSTD table, will cause the system to generate an error message, output a material description of "Bolts undefined" for that joint (with zero item code, size and quantity), and continue detailing.

4.5 Bolt Length Calculation

The calculated length of a bolt to be used to secure a joint has to take account of various factors, as explained in the following sections.

4.5.1 Bolt Length Calculation for Standard Flange-Gasket-Flange

The calculation of bolts will be triggered by the presence of a connection type starting with F (flange face) or L (lap joint), and will be completed by a second occurrence of such a connection type.

The calculation steps are:

1. At the first component, go to the CATREF, and find the BTSE name assigned in the BLRF for the appropriate p-point of the component. Store this reference (BTSE1).
2. Go to the second component. If its connection type starts with G (a gasket), get its P1 - P2 distance, and go to the next component.
3. If its connection type starts with F or L, go to the CATREF, and find the BTSE name assigned in the BLRF for the appropriate p-point of the component. Store this reference (BTSE2).
4. Check for matching member counts of BTSE1 and BTSE2 (excluding jacking bolts, that is with BTYP JACK).

For each bolt in the set:

5. Get the Bolt Points with corresponding NUMBER.
6. Check for matching BDIA values.
7. Add the two values of BTHK (bolted flange thickness) together.
8. Add the gasket thickness (if found).
9. Select the bolt from the Specification and obtain its standard (rounded up) length. See [Bolts SHOP Flag](#).

The following error conditions generate an error message, output a material description of "Bolts undefined" for that joint (with zero item code, size and quantity), and continue detailing.

- No valid BOLTREF or CATREF for the second flange
- Unequal number of members for BTSE1 and BTSE2, excluding jacking bolts
- Mismatch in BDIA attributes between equivalent members of BTSE1 and BTSE2

4.5.2 Bolt Length Calculation for General Wafer Components

The calculation of bolts through wafer components is similar to the standard flange-gasket-flange calculation, with the following differences:

- If within the joint, a component's connection type starts with W (wafer), get its P1 - P2 distance, and go to the next component.
- The joint may contain any number of wafer components, (for example, butterfly valves, orifice plates, spectacle plates, spades) and any number of gaskets. The total bolted through length will be summed and added to the flange thickness.

4.5.3 Bolt Length Calculation for Components with Mixed Bolt Needs

The calculation of bolts through components with mixed bolt needs is the same as the general wafer calculation, but allows for different bolts in the set to start and stop at different places within the joint.

If a wafer component has a BLRF of CATREF, and the BTHK value for a particular bolt is not zero, then the specified thickness is used in the calculation. If the BTYP of the bolt is JACK, TAP or CAP, this will complete that particular bolt. A subsequent BTYP of JACK, TAP or CAP for the same bolt NUMBER further on in the joint triggers a new bolt.

This allows for such a situation as a wafer lug valve, with some bolts going through the whole joint, and some cap screws on either side bolting into the body of the valve (see [Figure 4:4.: Example Data Flow Diagram for a Wafer Lug Valve](#)).

Bolts are selected from the Specification as normal, but each bolt path through the joint will be able to start and stop at different points within the joint. A BTYP of NULL indicates that no bolt is required at this particular point in the joint. This is only permitted before the start of a bolt or after the completion of a bolt. If inconsistent combinations of bolt path data are found for a particular bolt, an error message is output, no bolts are output for that joint, but detailing will continue.

4.5.4 Bolt Length Calculation at Nozzles

Nozzle flanges are handled in exactly the same way as other flanges, via a BLRF of CATREF.

4.5.5 Additional Bolting Items in Bolt Length Calculations

Once the accumulated flange and wafer thickness has been calculated, the lengths of elements such as nuts and washers must be added; see [Figure 4:2.: Additional Item Lengths](#).

The bolt SPCOM SBOLTS have the attribute BLIST which is a real array of lengths corresponding to the BITEMS array. Each individual length may take an actual value or be set to zero. BLIST may have more lengths than BITEMS has selectors, allowing lengths to be taken into account without further selection of material, e.g. if nuts are included in the bolt description.

The extension thread length will be taken from the XLEN attribute of the SBOLT and added to the calculated length.

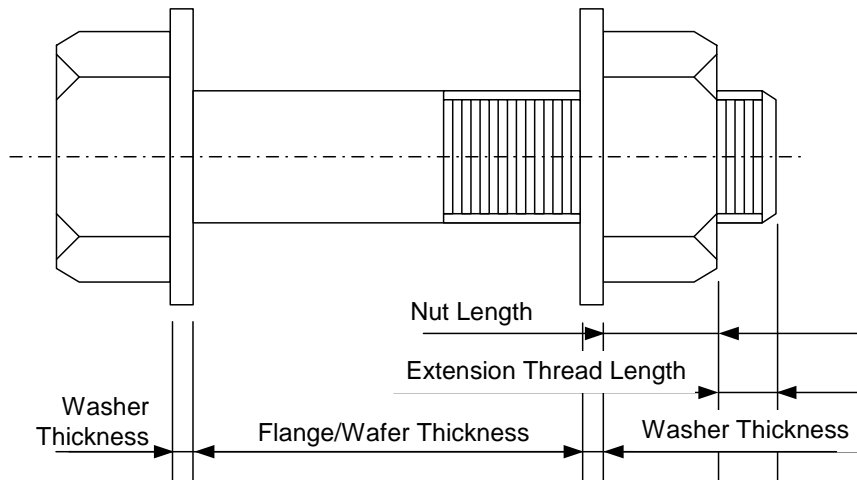


Figure 4.2. Additional Item Lengths

4.6 Part Numbers and Tags

If part numbers have been set in SPOOLER, you can specify that these numbers are used on the material list, or that ISODRAFT will generate part numbers automatically.

- If part numbers generated by ISODRAFT are used, like bolt items will be totalled in the material list.
- If part numbers from the database are used, one part number is allocated to each type of bolt item in an individual bolt set, i.e. if all bolt items in a set are identical, there will be a single part number for the set.

For more information about part numbers, see the MATERIALLIST command in the Chapter, *Command Reference Section*, of the [ISODRAFT Reference Manual](#).

Multiple bolt item part numbers tagged to the same location will be combined as for additional items on the material list; for example, B10/B11/B12.

4.7 Example Wafer Lug Valve

The diagram below illustrates the wafer lug valve scenario. This example shows a wafer valve requiring a set of 12 bolts. There are:

- 4 machine bolts which pass through the whole joint from one flange to the other (Numbers 2, 3, 6, 7).
- 4 cap screws on each side of the valve, which secure the valve to the flanges (Numbers 1, 4, 5, 8).

The machine bolt lengths are based on the combined thickness of the two flanges, both gaskets, and the valve itself. The cap screw lengths are based on the thickness of a single flange and gasket combined with the tapping depth in the valve body.

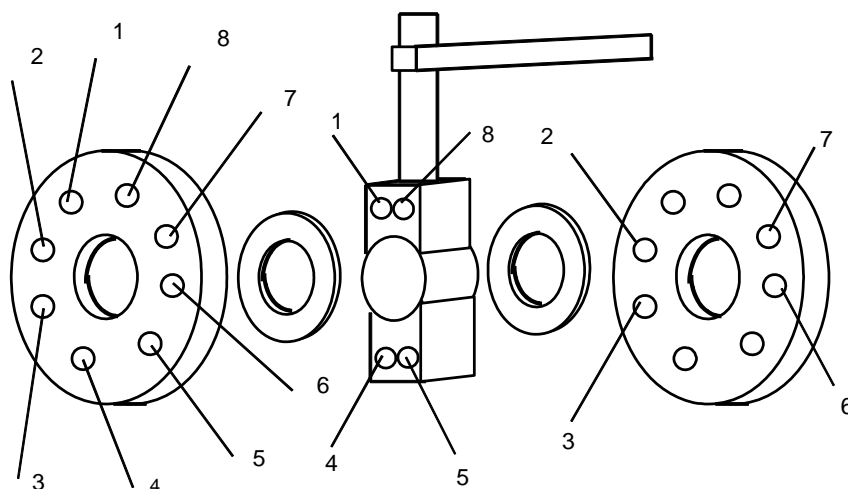
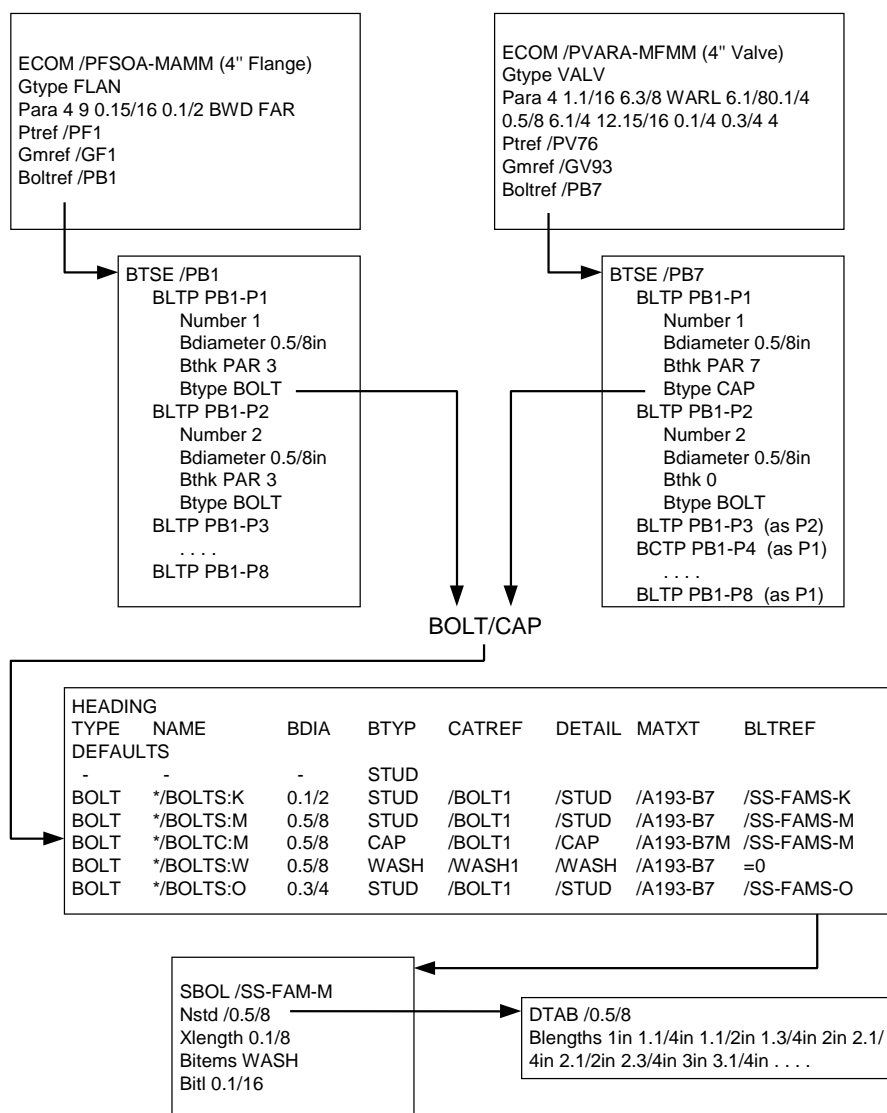


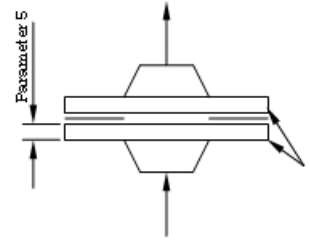
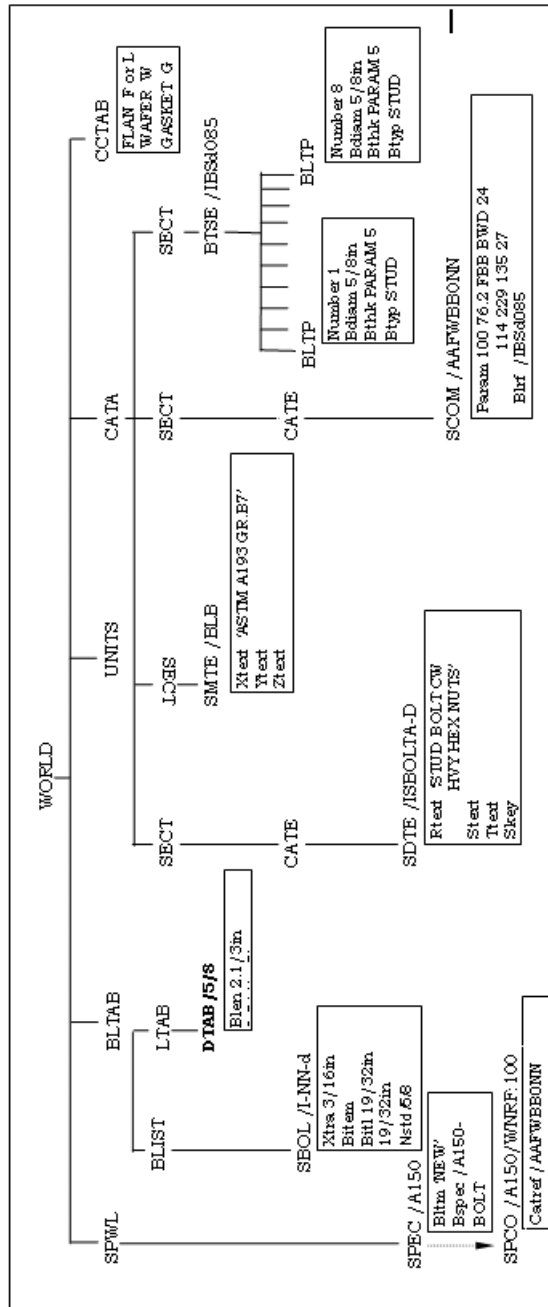
Figure 4.3. Example Wafer Lug Valve

A data flow diagram for the bolting of a typical lug valve is shown in [Figure 4.4.: Example Data Flow Diagram for a Wafer Lug Valve](#)



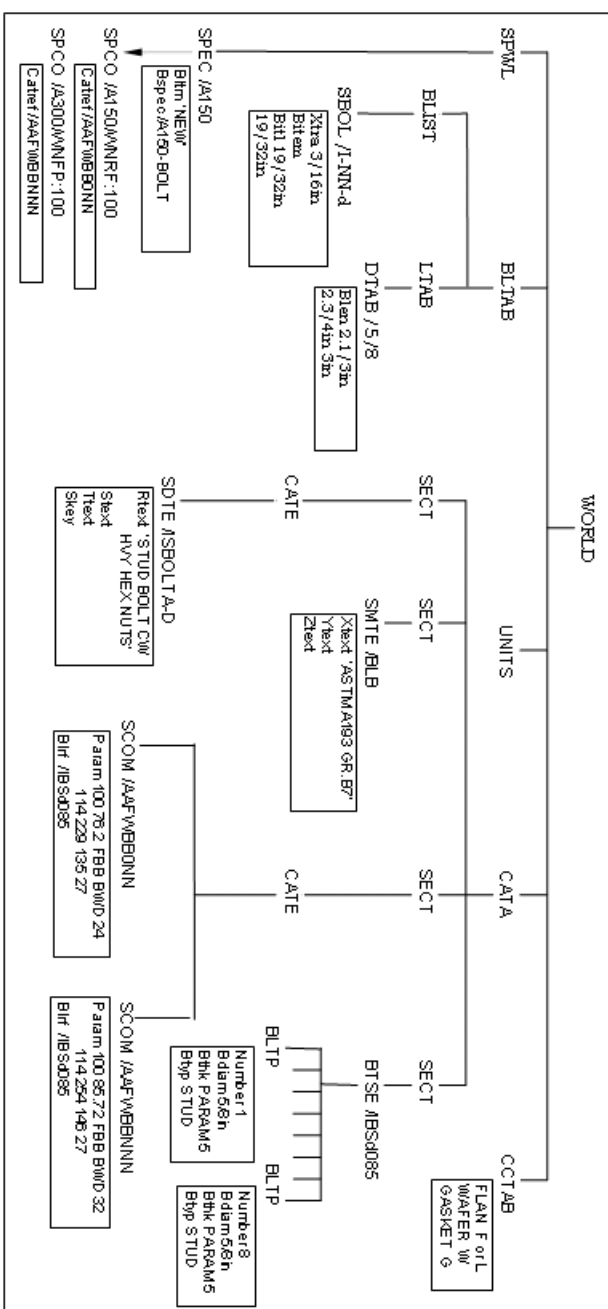
$$\text{Bolt length} = (0.15/16 + 0.5/8 + \text{Gasket thk.} + \text{Xlength} + \text{Washer thk.}) = 2\text{in}$$

Figure 4.4. Example Data Flow Diagram for a Wafer Lug Valve



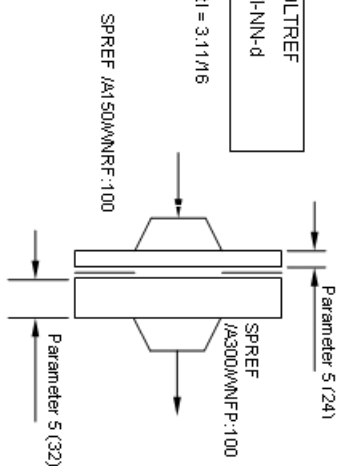
TYPE	NAME	BDIA	BTYP	CATREF	DETAIL	MATXT	CMPREF	BLTREF
BOLT	* / I-NN-d	5/8in	STUD	/ ISBOLTA	/ ISBOLTA-D / ELB	=0		/ I-NN-d

Bolt Length Calculation = Param 5 of Flange + Gasket Thickness + Param 5 of Flange + Xtra length + Bth = 3.7/16
 Bolt Length Calculation = 3.1/2" (next available length from DTAB)

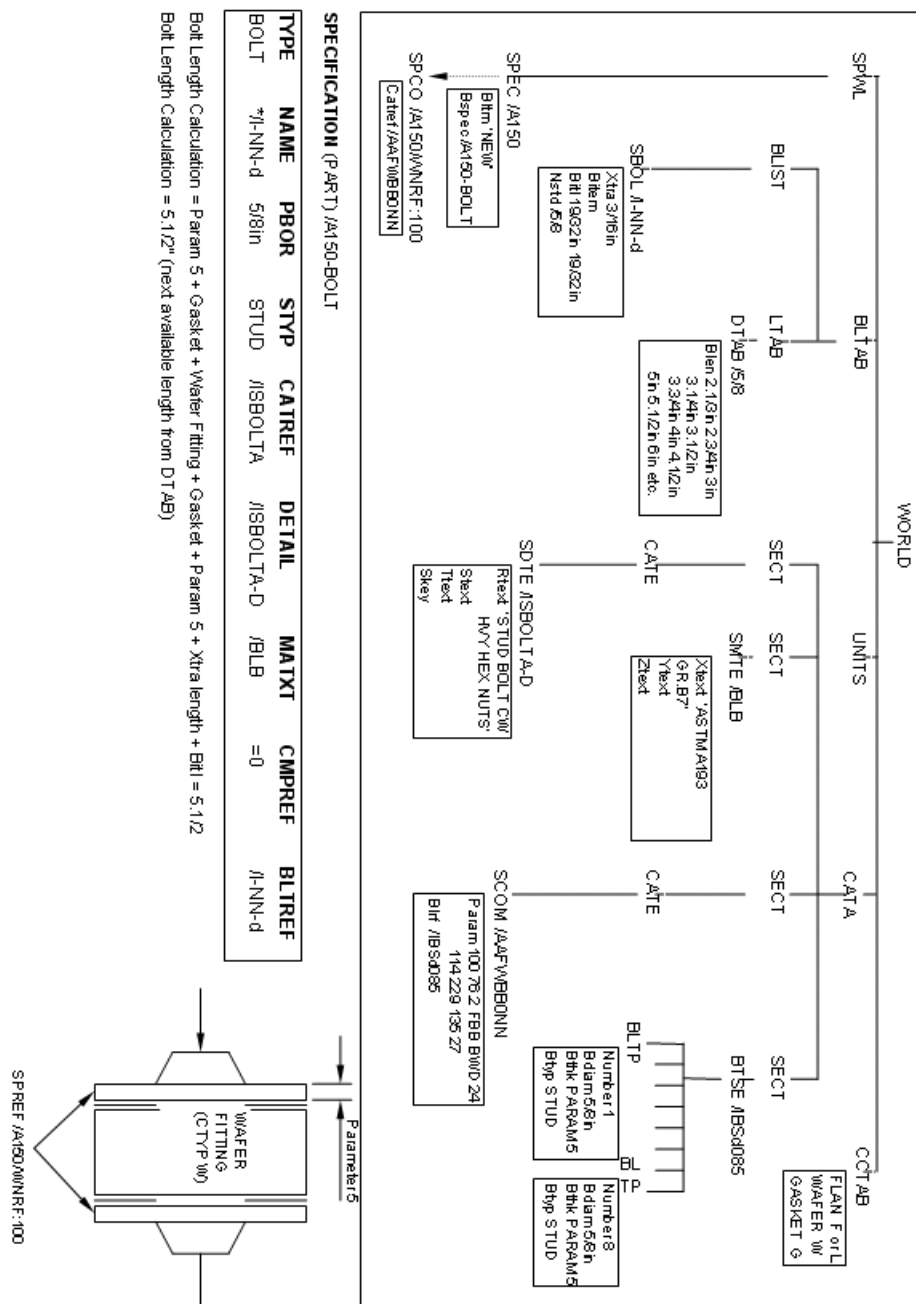


TYPE	NAME	BDIA	BTYP	CATREF	DETAIL	MATXT	CMPPREF	BLTREF
BOLT	*I-NN-d	5/8in	STUD	/ISBOLTA	/ISBOLTA-D	/BLB	=0	/I-NN-d

Bolt Length Calculation = Param 5 of Flange 1 + Gasket Thickness + Param 5 of Flange 2 + Xtra length + Btl = 3.11/16
 Bolt Length Calculation = 3.3/4" (next available length from DTAB)







4.8 Administrative and Geographical Use of Selectors

In addition to Bolt Specific Type and Diameter, a further degree of bolt selection is allowed via the BSEL attribute of a branch member. This takes a value which can be used as the answer to a BSEL selector in the Bolt Specification. For example, if the Pipe Specification dictates that the same flange has black bolts indoors and galvanised bolts outdoors, then BSEL could be set to either IN or OUT at the flange, with entries for both bolts in the Bolt Specification. Any inconsistency in BSEL attributes between the two ends of the joint

causes the system to generate an error message, output a material description of “Bolts undefined” for that joint (with zero item code, size and quantity), and continue detailing.

A further application of this assists in the selection of isolation kits. A flange may require an isolation kit in certain duties and not others. The kit may require longer bolts to cater for isolating washers. A different flange BSEL value will cause a Bolt SPCOM to be selected which has an SBOLT that includes isolating washers in its additional items arrays.

4.9 Alternative Item codes for Bolts

Alternative item codes for bolts are taken from UDAs set on DTAB elements, not on the SPCOM like other elements (see [Alternative Item Codes](#)).

Note: For additional bolting items such as washers and nuts, the UDA must be set on SPCOM because these items do not have associated DTAB elements.

Alternative item codes for bolts are set up as follows:

- The SBOLT elements must have NSTD attributes pointing to the DTAB elements.
- The names of the UDAs on the DTAB elements must be related to the UDAs set for the SPCOM. For example, if the SPCOM UDAs are named: clienta and :clientb, there could be one set of DTAB UDAs named :clienta1, :clienta2, etc. and another set of UDAs named clientb1, clientb2, etc.
- The item codes are stored in the DTAB UDAs as texts separated by spaces. As text UDAs have an upper limit of 120 characters, the list of item codes continues across the set of UDAs.
For example, if each item code is 10 characters in length, then 11 item codes can be stored in each UDA of the set.
- The number of item codes stored in each set of UDAs (for example, all the UDAs for clientb) should correspond to the number of elements in the BLEN attribute of the DTAB.

Notes:

- Normally, you can use the same SBOLT element for bolts with the same geometry, and the material will be defined in the Specification. When bolts of the same geometry but different materials are used, you will need to create a DTAB for each SBOLT element and an SBOLT for each Bolt SPCOM if you want to set different item codes.
- You must ensure that a consistent set of UDAs are set up in the LEXICON database.

The following example shows part of a specification and part of a catalogue. Note that SPECON cannot output UDAs in tabular format, and so this example is just for illustration.

Part of Specification

```

TYPE    NAME      ...    BLTREF  :clienta  :clientb
FLAN    */FL100   ...           F100L    FLAN100
BOLT    */BL100   ...    /BL100   unset    unset

```

Part of Catalogue

```

SBOLT/BL100
  NSTD /M16
DTAB/M16
  BLEN 90 100 110 120 130 140 150 160
  :clientb1 AB99XA AB99ZB AB29ZC CA33QD
  :clientb2 CA99PE JX34YF JY21ZG AC99AH
  :clienta1 B12345 B19432 B21456 B92134 B54321 B45123 B51234 B41592

```

4.10 Producing a Bolting Report

You can use ISODRAFT to produce a Bolt Report, either to your screen or to a file. This takes the form of an ASCII file with a page length defined by you. You can specify whether you want the bolting information to replace an existing file, or be added to an existing Bolt Report File (which is the default).

To produce the report you use the BOLTREPORT command. This command is similar to the DETAIL command in that the pipes to be detailed are scanned for bolting information. You can repeat the report whenever you want, interspersing the BOLTREPORT commands with DETAIL and CHECK commands as necessary. You will experience the same system checks when producing a bolt report as for detailing, but here only diagnostic information relevant to bolting will be produced. Therefore, you do not have to make sure that all piping geometry is correct before producing a bolt report. Note, however, that it is important to have the piping's connectivity correct before producing a bolt report to make sure that all pipes are scanned properly.

You can include positional information in the report. This will give you the centreline position of either associated flange in the coordinate system (as currently defined by the AXES command). The format of the positional information will be determined by the UNITS command.

For further information see the chapter *Command Reference Section* in the [ISODRAFT Reference Manual](#).

4.11 Summary of Commands

The following commands were described in this chapter:

BOLTFILE	Specifies the name of the bolt report file, produced using the BOLTREPORT command.
----------	--

BOLTING	Specifies where and how the bolt lengths are to be shown in the material list.
BOLTREPORT	Allows you to output a report on the bolting containing identical information to that displayed in the material list.

5 Design Requirements

This chapter describes how the pipework should have been designed so that ISODRAFT can produce the correct output.

5.1 Using the SPOOLER Module

The SPOOLER module allows you to:

- Define the contents of the spool drawings, independent of the OUTFITTING pipe and branch hierarchy,
- Add spool breaks to the design model, interactively;
- Create numbering data for welds, pulled bends, parts, spools, and joints.

A Spool Drawing (SPLDRG) in SPOOLER normally forms a single isometric in ISODRAFT (although it may be plotted over a number of sheets). Each SPLDRG contains a number of Spools (fabricated assemblies) and Fields (erection parts) connected as a continuous piping network. The spooling data is defined in the Fabrication database, from where it is cross-referenced to the OUTFITTING DESIGN database.

You define your spool drawing by selecting pipes and components that form a logical interconnected network, from the graphical display. Spools and Fields are automatically set where changes in the SHOP flag status occur. You can also change the SHOP flag on a component; force a Spool Break between two components; and add additional field welds, either on a component p-point or in a length of tube, to further split up the piping network. To provide even greater control, you can also set the SFLimit (spool/field limit) attribute on the SPLDRG to BRAN, PIPE, ZONE, SITE or WORLD (the default). This will force a spool break at any change of element at the specified hierarchic level; for example, at every change to a new Pipe.

Once all the elements have been added to your spool drawing you can automatically number the spools, parts, welds, pulled bends and joints. These numbers can be easily updated if you make changes to your spool drawing or the design is changed.

Once you have defined and numbered your spool drawings they can be plotted using ISODRAFT. Normally one spool drawing is shown on an isometric. Alternatively, you can produce isometrics showing either a single spool or a network of spool drawings.

Further instructions on using SPOOLER can be found in the chapter *Pipework Spooling* of the *Pipework Design User Guide*.

5.1.1 Numbering in Spool Drawing Isometrics

SPOOLER can automatically create numbering for spools, parts, welds, non-welded joints and pulled bends. These numbers are stored in the database and can be output on the isometrics by enabling the required commands.

Spools	SPOOLNumbers FROMDB
Parts	MATERiallist PARTNumbers FROMDB
Welds	WELDNumbers FROMDB
Joints	JOINTNumbers ON
Pulled bends	PBENDNumbers ON

5.2 Attachment Point Elements

ISODRAFT uses attachment point elements in the OUTFITTING DESIGN database, known as ATTAs. These are used to allow forces, moments, etc. to be applied to a Branch during pipe stressing, to show the location of support points, and to attach comments, flow arrows, etc. to the pipes on isometric plots.

The different types of ATTA are defined by the setting of the ATTY attribute. The ATTY settings and other information for different types of ATTA are given below.

- **Support ATTAs** are used to mark the position of a support on the pipe. There are three types of support ATTA:
 - ATTY type **Unset**, is the default type of support ATTA. They are used to mark the position of a support on the pipe. They should be positioned on the centreline of the pipe. If they are not:
 - Data consistency checking will generate an error message, unless ATTACHECK OFF is set, in which case no check is made for correct orientation and position of the ATTA.
If the ATTA is within tube, it will be shown at the p-leave of the previous component or at the head of the branch. This can cause the dimensioning of another Support ATTA further down the pipe to be omitted.
 - To position the ATTA within the previous component, connect it in list order after the component, then explicitly set its position, or move it BY the required distance along the direction of P-arrive. Note that, in this case, data consistency checking will generate an error message unless ATTACHECK OFF is set.
For elbows and bends, ATTAs can be positioned on projected centrelines, i.e. at the intersection of centrelines.

Note: Supports on tube are shown with a symbol, but supports on components are not.

- ATTY type **Hang** ATTAs are used in the Hangers & Supports application to indicate where a hanger type support will be attached to a pipe or component. They operate in exactly the same way as ATTY type Unset ATTAs.
- ATTY type **Weld** ATTAs are also used in the Hangers & Supports application. This type of ATTA is used to position the plotted ATTA symbol a fixed distance inside the previous component (e.g. at the P0 of an ELBO component), rather than moving the ATTA to that point as you have to do with ATTY types Unset and **Hang**.
It is accomplished by the ATTA having a P3 point, which is a set distance in the arrive direction from P1/P2. The distance between P1/P2 and P3 must be set in

the catalogue for each ATTA of this type, which is then used with the same component (e.g. ELBO). The symbol for this ATTA is automatically plotted at its P3 point.

Inserting the ATTA directly after the previous component ensures the ATTA symbol is plotted at the correct point inside that component.

- **Comment ATTAs** are used to add a drawing note with leader line to a position on a pipe, typically to add a floor penetration note. They have ATTY set to CCCC. If this point is not already dimensioned, for example, if it lies in the tube, additional dimensions to the ATTA position will be output.

Comment ATTAs should lie on the centreline of the pipe.

To position the ATTA within the previous component, connect it in list order after the component, then explicitly set its position, or move it BY the required distance along the direction of p-arrive. Note that, in this case, data consistency checking will generate an error message unless ATTACHECK OFF is set.

- **Non-dimensional Comment ATTAs** are used to add a drawing note with leader line to a position on a pipe; for example, to add a paint colour note. They have ATTY set to CCNN. This type of ATTA will not be dimensioned.
- **Flow ATTAs** are used to position inline flow arrows in tube. They have ATTY set to FLOW. The arrow is not dimensioned, but it is positioned on the tube in proportion to its 3D position.

The flow is from head to tail if the FLOW of the branch is the default forwards direction, or tail to head if the FLOW of the branch is backwards.

Note: The FLOWARROWS INLINE command must be used for flow arrows to be plotted for tube.

- **Split ATTAs for a normal isometric** are used to set the point at which an isometric should be split onto the next sheet. They have ATTY set to XXXX.

ISODRAFT sometimes repositions split points if a more logical point is close. For example, consider a Tee with a Flange welded to P3, and a Gasket and a Valve connected to the Flange. If a Split ATTA is positioned between the Tee and the Flange, ISODRAFT will split the pipe at the natural break formed by the Flange.

Normal Split ATTAs are ignored for system isometrics and equipment trim isometrics.

- **Split ATTAs for system isometrics** have ATTY set to SSSS. This type of ATTA is used to set the point at which a system isometric or equipment trim isometric should be split onto the next sheet. The same considerations as for normal isometric split ATTAs apply.
- **Insulation symbol ATTAs** have ATTY set to INPP (insulation personnel protection). This ATTA type is used to set the start and stop points of insulation on the isometric. The insulation specification note will be shown at the start point of the insulation, but not at the end (to avoid duplicating information).

5.3 Reference Dimensions

If you have set up reference dimensions in the OUTFITTING DESIGN database, you can display them in ISODRAFT. To define a reference dimension, you must specify:

- A reference datum (see [Setting up a Reference Datum](#)).
- A point on a Pipe which you want to dimension (see [Setting up a Reference Dimension](#)).

The dimensions will be drawn from the reference datum to the point on the Pipe.

Note: Reference dimensions do not affect normal dimensioning. Also, they can be switched on and off independently of other types of dimensions. See the REFDIMENSIONS command.

5.3.1 Setting up a Reference Datum

For an element to be used as a reference datum, it must have:

- An origin.
- An SKEY attribute, or a parent with an SKEY attribute.

The elements you can set up as reference datums are:

- Datums;
- Equipment;
- Equipment and structural primitives such as boxes;
- Sections;
- Piping components;
- Point elements, if they have a parent with an SKEY attribute.

SKEY Attributes

Elements used to define reference datums must have an SKEY attribute, or a parent with an SKEY attribute (but note that the parent element need not be the immediate owner of the element). If the element does not have an SKEY, and no parent has an SKEY, miscellaneous SKEY types (see below) will be used.

Elements with SKEY attributes are:

Datums
Equipment
Pipes
Structures
Substructures
Drawings
Frameworks
Subframeworks

Different SKEY types have different ATEXTs associated with them, as shown in the following table.

SKEY Type	Item Type	ATEXT	Default text
HST*	Horizontal steelwork element	460	BEAM \$?
VST*	Vertical steelwork element	461	COLUMN \$?
BLD*	Building	462	? \$BUILDING CL
EQU*	Equipment item	463	CL EQUIPMENT \$?
PIP*	Piping component	464	CL PIPELINE \$?
FLR*	Floor level	465	? \$FLOOR LEVEL
WAL*	Wall	466	? \$WALL

SKEY Type	Item Type	ATEXT	Default text
GRD*	Grid line	467	GRID LINE \$?
XXX*	Miscellaneous	468	?

In SKEY Types above, the symbol * represents any character. In Default texts, \$ is replaced by a new line and ? is replaced by the name of the element. If the element does not have a name, ISODRAFT looks up the hierarchy until it finds a parent with a name, and then uses that parent's name.

If the reference element is a Datum element, it will have an STTEXT attribute. You can set this attribute to any identification text you want. If you do this, the text you enter will be used instead of the name. If you leave STTEXT unset, the Datum name will be used.

You should set the SKEY text to the SKEY type you require. Note that you can also redefine the text in the ATEXT element using the ATEXT command.

For example:

Pipe PIPE7-1 has an SKEY type VSTC. Associated with the pipe is ATEXT 464, with default text CL PIPELINE \$?. The following text will appear on the isometric (see [Figure 5:1.: Example plot file](#)):

```
CL PIPELINE
PIPE7-1
```

[Figure 5:1.: Example plot file](#) also shows a reference to a Datum, which has its STTEXT element set to Zero Elevation.

Figure 5:1. Example plot file

Origins and Centrelines

On the isometric, a centreline is drawn through the origin of the reference element. The reference dimension will be drawn from the centreline of the reference element.

The centreline will normally be the Z-direction of the element, as defined by its orientation.

- For Piping components the p-leave will be used.
- For Sections the p-line direction will be used.

- For Points, the direction of the centreline will be to the next Point in the list. For the last Point in the list, the direction from the previous Point will be used. If there are no other Points, the direction will be the Z-direction of the Point's owner.

5.3.2 Setting up a Reference Dimension

The point on the Pipe can be any of the following:

- The origin (P0) of an Elbow, Tee, Bend, Cross or OLET.
- The origin of a Support or Comment ATTA.
- The position of the Head or Tail of a Branch.

Note: 180 degree bends cannot be used for reference dimensions.

Piping Components and ATTAs

A reference dimension is set up by setting the DMTYPE and DMFARRAY attributes of the piping component or ATTA.

Support ATTAs and dimensioned or non-dimensioned Comment ATTAs can be used to create reference dimensions for points on tube. Split ATTAs and Flow ATTAs cannot be used.

DMTYPE is a **word array** attribute. It defines which component of the reference dimension is plotted. It can be set to one or two of the following:

EAST
NORT
ELEV or
POS for a 3D dimension

Figure 5.1.: Example plot file shows one reference dimension to a piping component with POS specified, and one reference dimension to a Datum, with ELEV specified.

DMFARRAY is a **reference array** attribute. It is used to set the names of the reference elements.

For example:

```
DMTY NORT EAST
DMFA /N.WALL /VESS2
```

produces two reference dimensions, one giving the distance North to the centreline of the element named /N.WALL and one giving the distance East to the centreline of the Equipment named /VESS2.

```
DMTY ELEV
DMFA /SITE.BASE
```

produces a reference dimension to the centreline of the element /SITE BASE.

The dimension will be to the centreline of the Reference element.

Branch Heads and Tails

A reference dimension is set up by setting the HDMTYPE and HDMFARRAY attributes of the Head or TDMTYPE and TDMFARRAY attributes of the Tail.

HDMTYPE and TDMTYPE are **word array** attributes. They can be set to one or two of the following:

EAST
NORT
ELEV or
POS for a 3D dimension

HDMFARRAY and TDMFARRAY are **reference array** attributes used to set the names of the reference elements.

5.3.3 Errors

If you try to create a one-dimensional reference dimension which is parallel to the centreline of the reference datum element, the result will be as follows:

- Dimensions to piping components or Sections will be omitted and an error message will be given.
- Dimensions to other elements will be created with a different centreline direction, and a warning message given.
- If you try to create a one-dimensional reference dimension where the dimension line runs parallel to the pipe centreline, no dimensions will be plotted and you will be given an error message.

5.4 Additional Items for Material Take-off

Items which are not part of the pipe (that is, items which are not dimensioned or shown graphically on the isometric) can be shown on the material list. Examples of this type of item are flange jacking screws, valve handwheel chains and extra blanks for testing. The additional items are associated with a normal piping component.

The additional items are associated with a piping component by setting the reference array attribute MTOR of the piping component to point to up to 20 additional Specification references (SPREFs). The item codes and descriptions will be extracted as normal.

The Detail and Material texts of the additional items are set as normal. A length or a quantity can be set for an additional item by setting the detail text element MTOL or MTOQ attributes, respectively. The description or quantity will appear in the Material list under Miscellaneous items.

5.5 Data Consistency Checking

Before you process a pipeline using ISODRAFT, you should carry out a data consistency check on the pipeline in OUTFITTING DESIGN or INTREPID. This is particularly important during batch mode operation.

5.6 Summary of Commands

The following commands were described in this chapter:

JOINTNUMBERS	Puts the joint numbers created in SPOOLER on the isometric.
MATERIAL PARTNUMBERS FROMDB	Allows you to select whether to use the part numbering from the database.
PBENDNUMBERS	Puts the pulled bend numbers created in SPOOLER on the isometric.
SPOOLNUMBERS FROMDB	Lets you specify whether to use the numbers created by SPOOLER or let ISODRAFT number the spools itself.
WELDNUMBERS FROMDB	Lets you decide whether to use the numbers created by SPOOLER or let ISODRAFT number the spools itself.

6 Piping Components in ISODRAFT

This chapter illustrates how piping components are normally represented on isometrics. The symbols for components are specified by the SKEY attribute of each component type when it is defined in the Catalogue database. You can define different symbols if required, as described in [Symbol Keys](#).

The symbols are shown in the chapter *Symbol Keys of the ISODRAFT Reference Manual*. This chapter mainly discusses how the symbols are annotated.

6.1 In-line Components, Including Straight-through Valves

- The number plotted in a box next to each component corresponds to the number of the item in the material list. All Valves and other in-line components have a flow-direction arrow plotted alongside (unless the arrows have been suppressed using the FLOWARROWS command). See [Figure 6:1.: Valve with flow arrow](#).

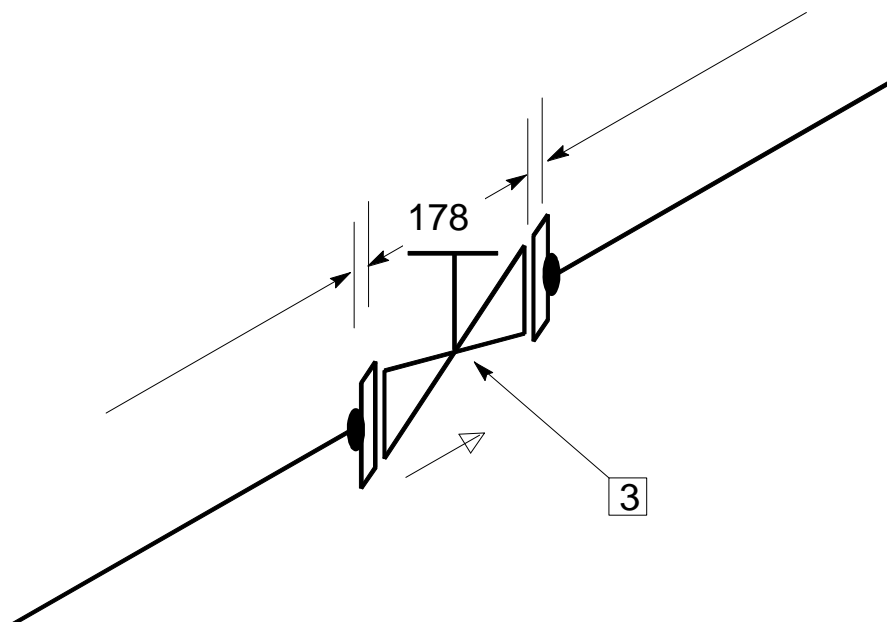


Figure 6:1. Valve with flow arrow

The directions of Valve operating spindles are correctly indicated. Valve operating spindles that are skewed will be plotted in the nearest primary direction and a comment

giving the actual direction added, as shown in [Figure 6.2.: Valve with flow arrow and direction text for spindle](#).

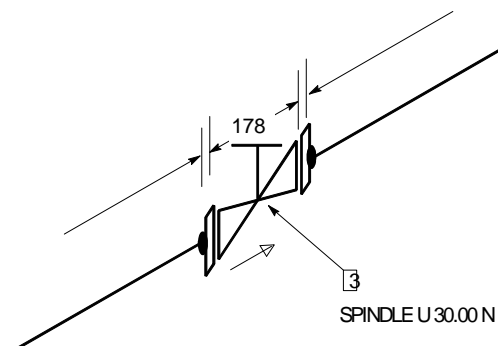


Figure 6.2. Valve with flow arrow and direction text for spindle

If Valve tag numbers have been stored in the OUTFITTING DESIGN database as names, they can be included in the boxed enclosures below the material list numbers if they are switched on using the TAGGING command. See [Figure 6.3.: Valve with tag number](#).

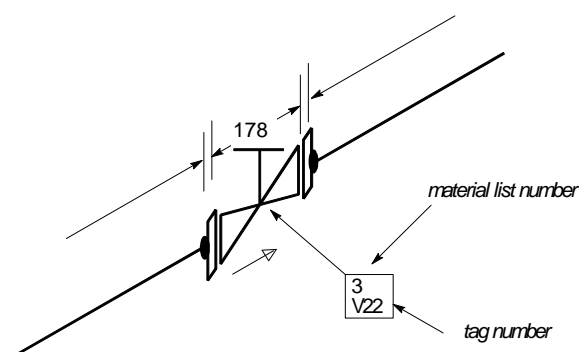


Figure 6.3. Valve with tag number

6.2 Angled, 3-Way, and 4-Way Valves

For angled, 3-way and 4-way Valves the dimension lines along valve body sections will not be plotted. The body dimensions will be shown with arrows pointing to the component. The figures will be followed by the units.

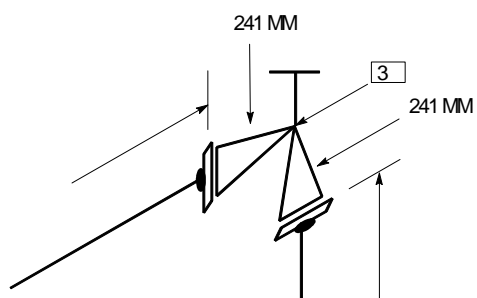


Figure 6:4. Angled valve with dimensioning detail

6.3 Flanges

The material list numbers for the Flange, associated Gasket and Bolts will be plotted as shown in [Figure 6:5.: Material list numbers for flange, gasket and bolts](#).

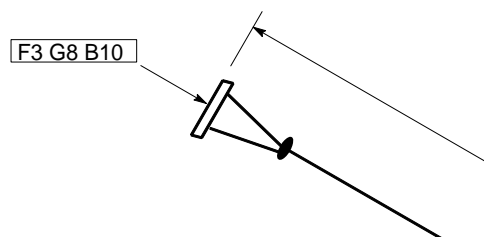


Figure 6:5. Material list numbers for flange, gasket and bolts

6.3.1 Pairs of Flanges

A connected pair of identical Flanges will be represented by two adjacent symbols, but the identifying material list number will be shown only once. Only one gasket and one set of bolts will be called up for each pair of Flanges.

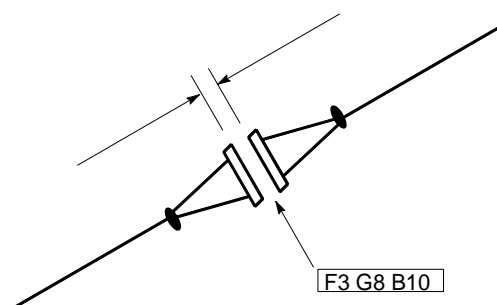


Figure 6:6. Single material list number for a pair of matched flanges

Where the connected Flanges have different codes, separate material list numbers will be shown. In this case, the associated Gasket and Bolt identifying numbers will be attached to the second Flange of the pair only.

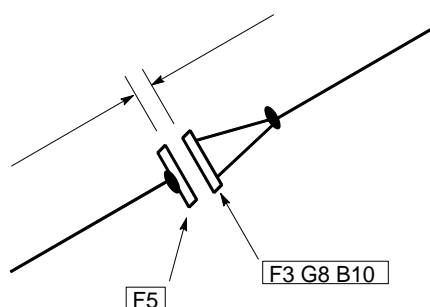


Figure 6:7. Flanges with different codes and separate material list numbers

6.3.2 Reducing Flanges

In addition to the normal material list numbers for the Flange and associated Gasket and Bolts, the primary and secondary sizes will be listed along with the message REDUCING FLANGE.

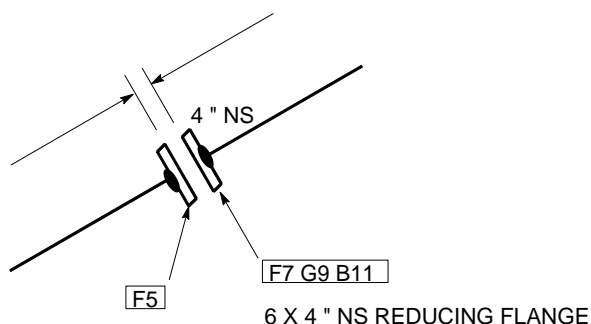


Figure 6:8. Additional information for a reducing flange

For offset reducing Flanges, the dimension and direction of the offset will also be shown.

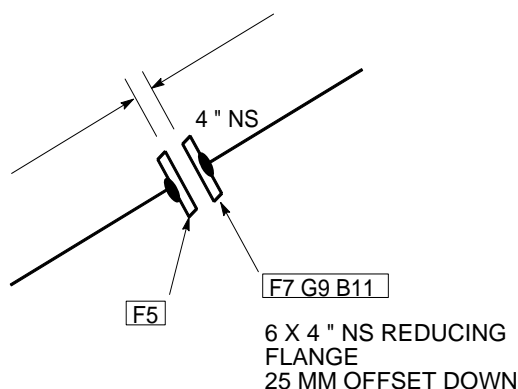


Figure 6:9. Additional information for an offset reducing flange

6.4 Wafer Fittings

Where in-line wafer fittings, such as 'Butterfly Valves', 'Orifice Plates, etc., are incorporated between pairs of flanges, a single set of extra long bolts is required. The material list number for these bolts will be added to the Flange and Gasket numbers for the second flange of the pair. The first Flange will also have the Flange and Gasket serial numbers attached, since two Gaskets are necessary in total for such an assembly.

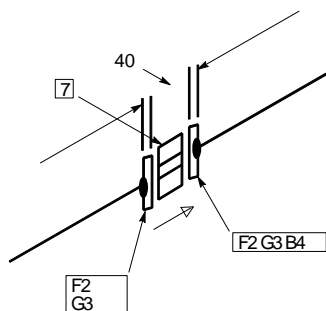


Figure 6:10. Bolt information for flanges enclosing wafer fittings

6.5 Tees and Crosses

TEEs, OLETs and set-on Tees (stub ins) are represented by different symbols. The primary and secondary bore sizes of the tee are indicated by a message linked to the tee intersection point. The branch pipe size is plotted against the third pipe leg.

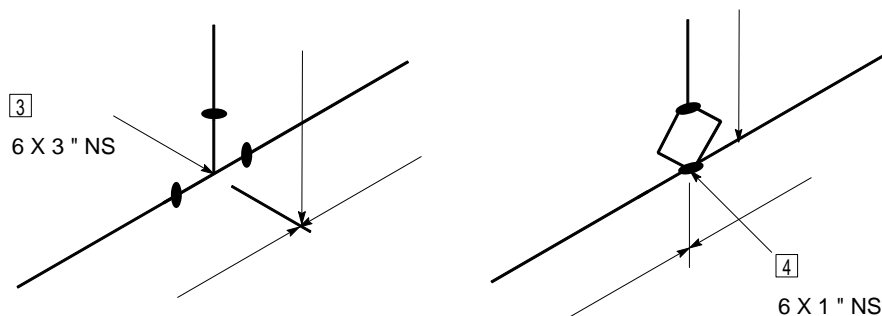


Figure 6:11. Presentation of TEEs and OLETs

A reinforced set-on Tee will have the text REINFORCED added to supplement the plotted shape. See [Figure 6:12.: Extra text for a reinforced set-on TEE](#).

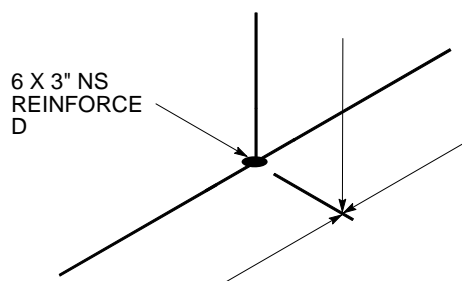


Figure 6:12. Extra text for a reinforced set-on TEE

6.5.1 Crosses

Crosses are shown like Tees except that the message specifying the main and branch sizes is extended to show the second branch size.

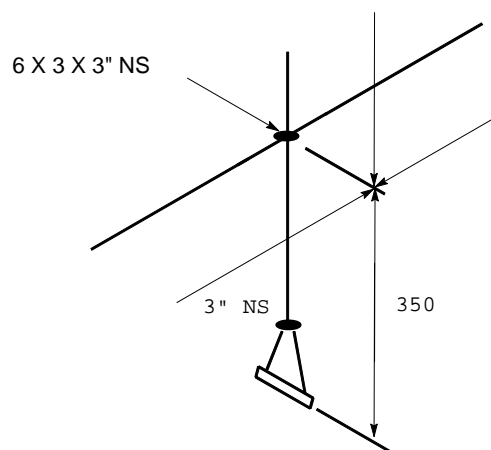


Figure 6:13. Additional information for crosses

6.6 Reducers

For both eccentric and concentric Reducers, the large (primary) and small (secondary) diameters will be shown in a message of the format LxS NS, where **L**arge and **S**mall are the nominal bore sizes.

An eccentric Reducer has its symbol drawn so as to show the orientation of its flat side, with the offset dimension listed under the size message.

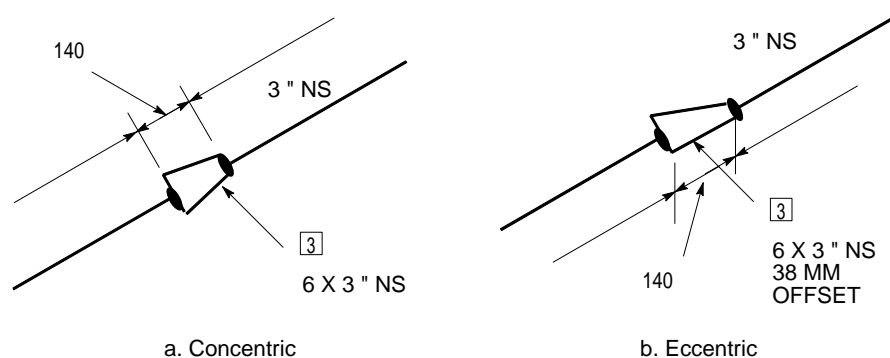


Figure 6:14. The annotation of reducers

Text information about the direction of the flat side of eccentric Reducers is not normally plotted, but you can plot it using the alternative text facility. See the ATEXT command, and ATEXTs 243 and 283. For example, the text FLAT DOWN can be output by this method.

6.6.1 Reducers With a Branch Connection

These can be based upon either a concentric or eccentric body.

Due to the difficulty of drawing an unambiguous representation of this type of component, all information relevant to the Branch will be given in a message line as shown in [Figure 6:15.: Annotation of a reducer with a branch connection](#). The message 1"NS refers to the branch bore, while the 100 (mm) dimension is an overall one from the connection point of the reducer to the flange face.

The dimension along the body of the Reducer to the branch will not be indicated and the branch pipe will be drawn at the mid point of the reducer.

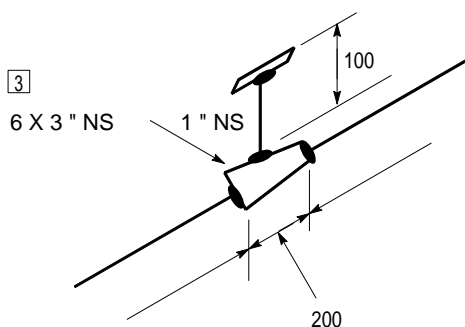


Figure 6:15. Annotation of a reducer with a branch connection

Note that, in the case of eccentric Reducers, the Branch may only be taken off the flat side or off the face directly opposite the flat side.

6.7 Elbows and Bends

Butt weld, socket weld or screwed Elbows will be distinguished from pulled Bends by the addition of appropriate markers. These will be plotted on both pipe legs a short distance from the intersection.

The material list number for a separate component, as opposed to a pulled Bend, will be indicated in a boxed enclosure in the usual way. A pulled Bend is treated as part of the tube and its length around the tube centreline is included in the material list.

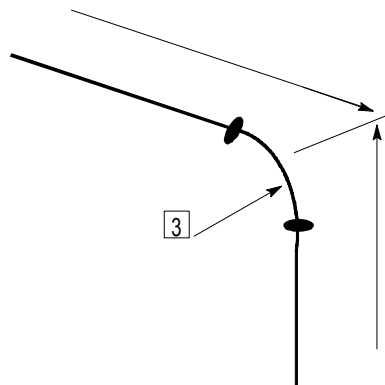


Figure 6:16. Representation of a butt welded elbow

Right-angled Bends and Elbows are not marked as such, since the angle is usually obvious. All other bend angles are identified by a message giving the acute angle in degrees, linked to the intersection point of the legs. See [Figure 6:17.: Identification of an acute angle](#).

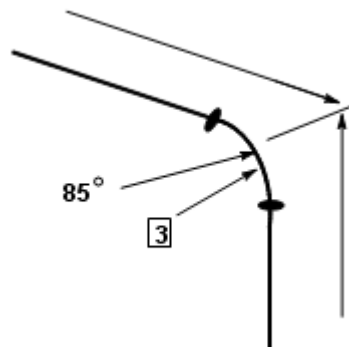


Figure 6:17. Identification of an acute angle

6.8 Pipe Ends and their Connections

Pipe ends will normally have specific text messages plotted next to them under the following circumstances:

- At the start of a Pipe;
- At the end of any type of off-line Branch;
- At the end of a Pipe.

These messages show how the Pipe is continued from that point onwards, or any special Pipe end condition that may be required, as shown in the following examples.

Full XYZ coordinates will be listed for the start and finish of each Pipe. They will also be listed at the end of every Tee off-line branch; in other words, at all connection points.

Where a Pipe is connected to another Pipe, the coordinates will be preceded by a CONT. ON message. In [Figure 6:18.: Information at an end of a pipe connected to another pipe](#), a Branch of PIPE-36A is the HREF or TREF of the Pipe shown.

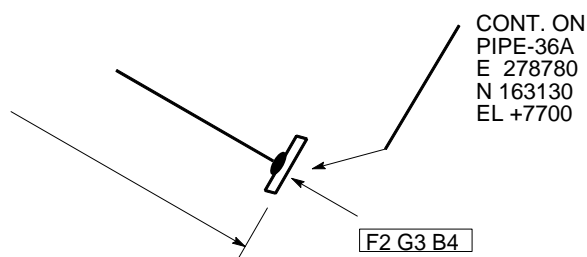


Figure 6:18. Information at an end of a pipe connected to another pipe

All changes in elevation will be marked at the first intersection point after the new elevation occurs.

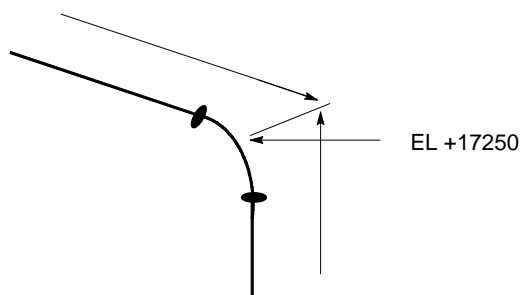


Figure 6:19. Detailing a change of elevation

6.8.1 Connection To Equipment Nozzle

The pipe end will carry the message **CONN. TO** followed by the relevant identification.

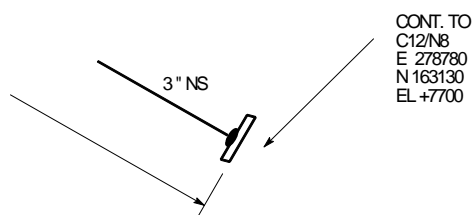


Figure 6:20. Detail of connection of a Pipe to a Nozzle. C12/N8 is the HREF or TREF identifier of the Branch.

6.8.2 Continuation on Another Drawing

The Pipe end will carry the message **CONT. ON** or **CONT. FROM** followed by the identification of the other drawing.

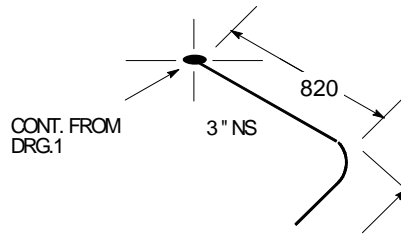


Figure 6:21. Detail of continuation from another drawing

6.8.3 Special Pipe End Conditions

An appropriate message will define the special condition, as illustrated in [Figure 6:22.: Labelling of special Pipe end conditions.](#)

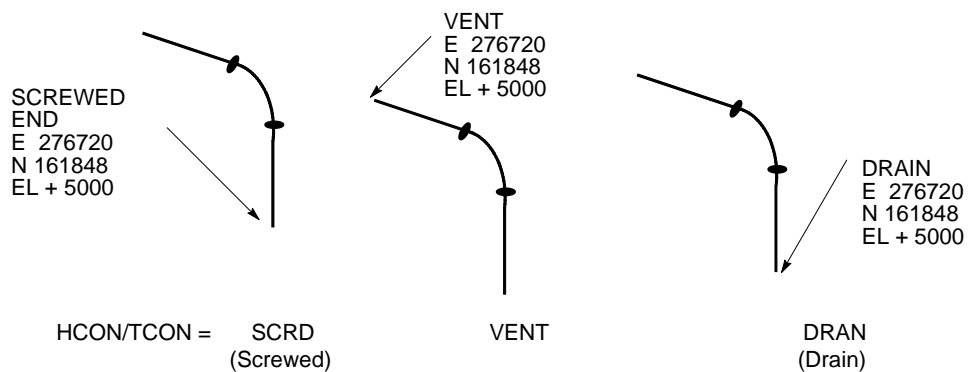


Figure 6:22. Labelling of special Pipe end conditions

6.8.4 Open and Closed Ends

Where a Branch end (HCON or TCON) is deliberately designated OPEN or CLOS it will *not* have any specific text marking it as such on the isometric drawing. Only the coordinates of the Pipe end will be plotted. The alternative text facility (see the ATEXT command) can be used to add extra messages to OPEN or CLOS ends if required.

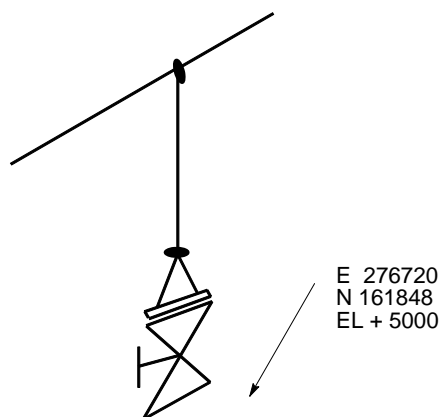


Figure 6:23. Coordinate marking at open and closed ends

6.9 Field-fit Welds and Loose Flanges

Because field-fit welds and loose flanges require fabrication on site, they will be highlighted by a message linked to the pipe end.

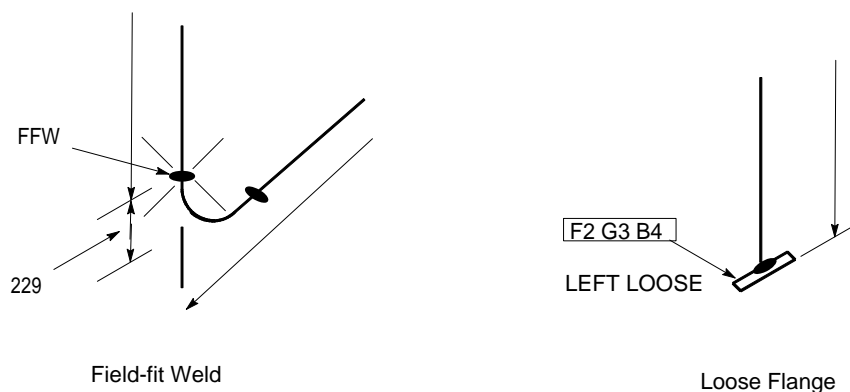


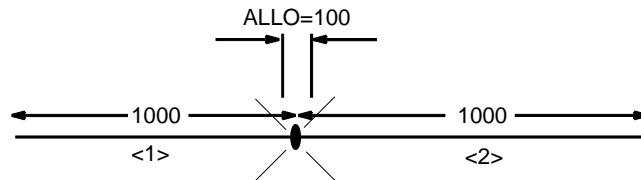
Figure 6:24. Messages for on-site fabricated connections

If a cut pipe list has been specified using the CUTPIPELIST command, ISODRAFT automatically includes a site fitting allowance in each calculated cut pipe length which is associated with:

- A flange with its LOOSE attribute set to TRUE (i.e. a loose flange);
- An element of generic type WELD which has the symbol key WF (i.e. a field-fit weld).

This allows for material loss during on-site preparation of the pipe end. The allowance is taken from the value of the ALLO attribute for the weld or flange, if set. If the value of ALLO is not set (or zero), the allowance defaults to 150mm. However, this default allowance can be altered by using the LOOSEFLANGEALLOWANCE command. The fabricator and/or erector can be informed of the in-built allowance by means of a message in the title block area.

For a field-fit weld in Tube, the allowance will be added to the Tube before the Weld. In order to put the allowance on the following Tube, the Weld must be flipped in OUTFITTING DESIGN so that p-arrive is P2.



If P-arrive = P1, <1> will be 1100 and <2> will be 1000
If P-arrive = P2, <1> will be 1000 and <2> will be 1100

Figure 6:25. Field fit allowance in Tube

Note: The dimension shown on the isometric drawing is the actual designed dimension.

6.10 Identifying Pipeline Specification Changes

The level at which a change in specification is identified may be set to Component or Branch by using the SPECBREAK command.

Where there is a specification change part way through a section of pipeline, the new specification reference shows both the normal and new specification references by means of arrowed messages similar to those used for dimensions.

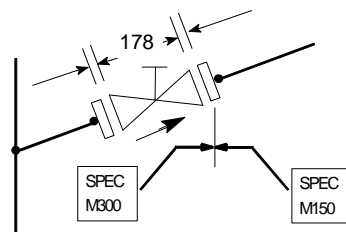


Figure 6:26. Marking a change in pipeline specification

The normal specification reference for the pipeline will appear in the bottom right-hand corner of the isometric drawing, immediately above the pipeline reference.

Changes in tracing, paint, insulation and material specifications can also be shown.

Single out of specification items are shown as in [Figure 6:27.: Single out of specification item.](#)

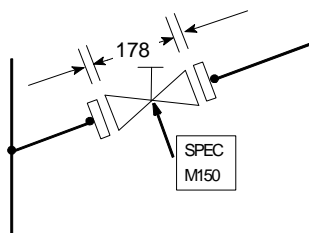


Figure 6.27. Single out of specification item

6.11 Fixed Length Piping

Fixed length piping is treated differently from normal fabricated pipework.

6.11.1 Fixed Length Piping In Isometric Drawings

The differences on the isometric are as follows:

- All fixed length pipe items, including straight tubes, are classified as fittings, and so each one is labelled with a material list number.
- No material list numbers are plotted for flanges, because all flanges are integral parts of the purchased pipe fittings. Material list numbers appear only for gaskets and bolts at flange positions.
- Where a flange has a different pressure rating or class from the normal one for that pipeline, a message giving the actual rating is linked to the flange. Rating changes of this type will occur mainly at vessel nozzle positions or at instrument valves.

6.11.2 Fixed Length Piping In Material Lists

Fixed length piping is treated in material lists as follows:

- No Fabrication Materials list will be plotted because all pipework assembly will be done on site. All materials will be listed under the Erection Materials heading.
- Where a spool includes a flange which has a different pressure rating or class from the normal one for that pipeline, a letter S will be added to the end of the spool's item code.
- If this occurs more than once on a single spool, a digit will be added to indicate the number of occurrences. Thus a spool which includes n special flanges will have Sn appended to its item code.
- If the line type identifier in the specification is FP, the overall length of the straight will be appended to the item code before any special flange is denoted. The units of length will be mm or sixteenths of an inch. For example:

`PFK-ST2500 or PFK-ST2500S1`
- Note that the length of the straight is not appended for symbol key FPPL, which is intended for standard length barrel nipples.
- If the line type identifier in the specification is FX, the length should already have been built into the code by the user and will not, therefore, be appended in this way.
- Spool numbers will not be listed for FP or FX piping, even when spool numbering has been specified, because fixed length pipes are classified as components not pipe spools. Fixed length straights will, however, have material list numbers.

- Multiple use of any fixed-length fitting will result in a single entry in the material list, with the quantity specified, in the normal way. Straights having identical lengths and bores are treated as identical fittings in this way.

6.12 Summary of Commands

The following commands were described in this chapter:

LOOSEFLANGE-ALLOWANCE

Allows you to specify an additional length on cut tube to compensate for field-fitted welds and loose flanges, where the ALLO attribute has not been set.

7 Customising the Drawing Sheet

This chapter describes how you can customise the layout and content of an ISODRAFT drawing sheet. The default layout is shown in [Figure 2.2.: Typical isometric drawing sheet layout](#).

7.1 Isometric Types

The ISOTYPE command specifies whether the information in the drawing is intended for:

- In-shop fabrication;
- On-site erection;
- Both in-shop fabrication and on-site erection;
- In-shop fabrication of pipe spools (i.e. prefabricated sections of a larger piping assembly);
- Any of the isometric types can be produced with the drawing area left blank.

A list of cut pipe lengths can be included if required. See the CUTTINGLIST command.

7.2 Units

You can specify the units for options such as paper size, margin widths, and cutting allowances. The default units are taken from the Catalogue. They can be changed before the value is input by using one of the commands:

MILLimetre

MM(a shorter form of the preceding option)

FInch(i.e. feet and inches)

INch

followed, in the same command line, by one of the options:

BOres

DISTances

to define the type of measurement to which the units are to apply. This sets the **current units**, which remain in force until reset.

For example, a loose flange allowance could be specified in feet and inches by the command sequence:

FINCH DISTANCES

```
LOOSE 1'6
```

Alternatively, values may be input together with their corresponding units regardless of the current unit settings. Thus the previous example could be replaced by the single command

```
LOOSE 1 FT 6 IN
```

without affecting the current unit specifications.

The current units may be queried at any time by using the command

```
Query UNIts
```

Units for dimensions on the isometrics are specified by the UNITS command. They can be queried by the Q OUTPUT UNITS commands.

7.3 Sheet Size

7.3.1 Designed Drawing Sheet Size

The overall size of the plotted sheet produced by ISODRAFT can be set to an ISO (International Standards Organisation) A-size, to an American standard size, or to specified dimensions. The selected paper size will be divided between the isometric area, the material list area, the title block area, the margins, and so on.

The default paper size is A2. It is changed by using the SIZE command.

7.3.2 Plotted Drawing Sheet Size

By default, the actual size of the plotted drawing sheet generated by ISODRAFT is the same as the designed sheet size. You can specify a plotted sheet smaller than the designed sheet by using the PLOTTINGSCALE command.

7.3.3 Margins On Drawing Sheets

You can set the top, bottom, left and right margins on the drawing sheet by using the MARGIN command. The default is a 5mm margin all round.

7.3.4 Blank Areas on the Drawing Sheet and Material list

An area along the bottom of the drawing area and Material list can be left blank by using the RESERVE command

A reserved area can be used for standard notes (a company name etc.), or it can be filled by information on a pre-printed sheet using the UNDERLAYPLOT command, or filled by Dynamic Detail Areas (see [Dynamic Detail Areas on the Drawing Sheet](#)).

7.4 Using a PLOT File as a Backdrop

An ISODRAFT PLOT file can be superimposed on a PLOT file created in another OUTFITTING module by using the UNDERLAYPLOTFILE command. Up to eight underlays

can be specified. The underlay PLOT files *must* have the same sheet size as the ISODRAFT output.

7.5 Arranging Multiple Plots on the Paper

If you are sending multiple plots to a single file (see the FILE command), the plots will normally be arranged one after another along the length of the paper. If the paper is wide enough, you may want to arrange the isometrics so that several are plotted across the width of the sheet. You can control how the plots are arranged by using the STACKINGARRANGEMENT command.

7.6 Dynamic Detail Areas on the Drawing Sheet

You can include Dynamic Detail Area PLOT files on the isometric. These show details of particular items of interest; for example, they could be used to show details of: welded supports; drain and vent assemblies; reinforcement pads; complex manifolds; and so on. They can be produced in OUTFITTING DRAFT or any other suitable 2D drawing program.

All branch members and specification components possess a DPFN attribute, which can be set to hold a Detail PLOT File Name. ISODRAFT can automatically pick up these PLOT files when detailing and place them on the isometric. The detail plot is output on the same drawing sheet as the component with which it is associated.

Detail Area output is switched on using the POSItion DETAILPlots command. Up to 20 Detail Areas can be plotted on a single isometric.

Detail plots may also include the parameterised values of system and user defined attributes (UDAs). The attributes shown can be defined using the PLOTDETAILDATA command. This allows you to select which attributes to show, the x and y coordinates relative to the origin of the plot file, the character height, and the angle at which the text is to be plotted.

The different attribute types are displayed as follows:

- Array type attributes are shown separated by a new line.
- Reference attributes are expanded to give the full OUTFITTING name.
- Logical attributes are output as T(rue) or F(alse).
- Position, orientation and direction attributes are separated by spaces.

7.6.1 Positioning the Detail Plots

A blank area of the drawing sheet can be set aside for the detail plots, using the RESERVE command. This command allows you to reserve an area at either the bottom of the plotting area or of the material list area. The plots can then be positioned in this area using the POSItion DETAILPlots command.

The POSItion DETAILPlots command requires you to specify the following additional information:

- The start point coordinates (x and y) for the first plot.
- The direction to line up further detail plots.
- A stacking integer and direction, to stack the plots if the number of plots exceeds the integer. Producing a matrix of parallel rows and column.

Alternatively, a grid reference number may be specified for each detail plot, in the DPGRID attribute of the component. The plot will be positioned at the origin of the defined area of the drawing area. The drawing area is divided up into 9 sections (3 x 3 square) as shown in the following diagram.

9	6	3
8	5	2
7	4	1

If the same grid square is referred to by more than one detail plot on an isometric, then only the first occurrence for that grid square will be output. This effectively limits the number of detail plots on an isometric plot to a maximum of 9.

Note: You cannot mix manually placed and stacked detail plots on the same isometric.

7.7 Drawing Frame

By default, ISODRAFT plots a frame round the drawing. You can suppress the frame, for example when pre-printed sheets or underlays are used, using the FRAME command.

7.8 Sheet Cutting Marks

The CUTMARKS command draws right-angled cut marks at each corner of the overall sheet area. The default is CUTMARKS OFF. When multiple plots are defined on the same drawing sheet, cutting marks will be superimposed.

7.9 Line Summary Box

This is a horizontal boxed area that is plotted across the bottom of the isometric drawing. It is subdivided into four standard information areas, as illustrated in the examples in Appendix *Examples of Isometrics* of the *ISODRAFT Reference Manual*. The contents of each area are self-explanatory.

You can change or remove the contents of the four boxed areas using the ATEXT (alternative text) command.

7.10 Title Block

7.10.1 Additional Title Block Text

You can customise the title block using the TITLEBLOCK command. You can add extra lines of text, and create boxes round them by using underscore and vertical bar characters.

7.10.2 Suppressing Title Block Text

You can suppress title block information by using the POSITION command to reposition title block text at 0,0.

7.10.3 Drawing Issue Numbering

When an isometric drawing is reissued after a change, the issue number of the drawing can be updated and plotted automatically by ISODRAFT. The issue number is taken from the REV attribute of the pipe being detailed. See the MARK REVISION command.

When this facility is used, the issue number is shown in the title block in the bottom right-hand corner of the isometric (see [Figure 7:1.: Standard style of title block](#)). The default text is ISS: *n*, where *n* is the issue number. This can be changed to REV: *n* or any other text by using the ATEXT (alternative text) command to change ATEXT 254.

DATE: 24 JUL 87	
PROJECT NO: VC12	
BATCH REF: A16 /OG8 /CG14	
PIPING SPEC: PS20	ISS: 1
0-4201	DRG NO 1 OF 2

Figure 7:1. Standard style of title block

7.10.4 Project Numbers on Drawings

The default is for no project number to be shown on the drawings. The project number can be included in the title block area on isometric drawing sheets by using the PROJECTNUMBER command. The project number shown on entry to the project can be plotted automatically, or a number can be added manually.

7.10.5 Date Format

By default, the date is shown in UK format. It can be changed to European or American format, or switched off completely, using the DATE command.

7.11 Alternative Text

The four main areas on the isometric drawing all contain, by default, standard text generated by ISODRAFT. The areas are:

- The isometric drawing area itself;
- The title block in the lower right-hand corner;
- The material list;

- The line summary across the bottom.

Example:

The FABRICATION MATERIALS heading on the material list.

The CONN. TO and CONT. ON messages in the drawing area

The BATCH REF and PIPING SPEC labels in the title block area.

Each text item is identified by a unique code number, known as its ATEXT (Alternative Text) number. The ATEXT command allows you to change the text characters associated with a particular ATEXT number, so that your specified text replaces the standard wording at the corresponding location on the drawing. You can also suppress the text by setting the ATEXT to null.

7.12 User-Positioned Text

All items of text used for the standard annotation of isometrics have default positions at which they are plotted on the drawings. Some of these annotations can be repositioned using the POSITION command. The POSITION command can also be used to position up to 100 pipe attributes such as the pipe's name, owner, temperature rating, and pressure rating, and including User-Defined Attributes (UDAs) and user-specified text. You orient the text at a specific angle if you require.

The standard text annotations which can be repositioned are as follows:

- The Y direction arrow;
- Spool numbers;
- Sheet numbers; i.e. X in DRG X OF Y;
- Sheet totals; i.e. Y in DRG X OF Y;
- The project number;
- The bend radius;
- System isometric names;
- The date.

The POSITION command also allows you to create a number of user defined tables to hold information you need (for example on welds or drawing history). See below for more details.

All text must be positioned in a margin, away from the drawing area, to avoid overwriting the isometric plot of the pipe itself. You will normally allocate a suitable margin by using the MARGIN command. No checks on your specified text positions will be made by ISODRAFT and it is up to you to ensure that you do not corrupt the isometric by poor positioning.

Note: It is possible to output the same piece of textual information at more than one position on an isometric plot by defining its coordinates more than once. The new definitions will *not* overwrite the earlier ones.

You can also suppress the output of attributes, system attributes or all user-positioned text.

7.12.1 User-Defined Tables

ISODRAFT allows you to create tables of detailed information to appear on your isometric. You can select which types of data you wish to appear (within the areas described below) and then automatically lay out the table. Lines are generated automatically between columns of data and under the tables, allowing you to concentrate on the data you wish to include.

You use the POSITION command to define the type of table you require, what data will be included and where it will appear on the isometric. The types of table you can define are:

- **Weld Tables** include all welds on the isometric (by weld number or by full identifier of weld prefix and number), and information chosen by you to appear in the columns you define. For each column on the table you can specify title, which weld attribute/UDA should be used to fill the column, column width and the order the columns appear on the table. You can select a single attribute, used to sort the table either forward or backward, by using the POSITION WELDTABLE command. The default choice is the weld number. If you choose another attribute, ISODRAFT sorts the information first by your chosen attribute, and then by weld number. If set, Additional Weld (AWELD) attributes can appear in the weld table.
- **Bending Tables** show bend data on the isometric. This is made up of a schematic drawing of the bend (dimensioned and annotated), followed by the table showing relevant lengths, radii and angles. You can output such bending tables as an ASCII file which can then be used to transfer data to a bending machine. You control this using the POSITION BENDTABLE command.
- **Revision Tables** show the history of the drawing, and can be defined to include the originators of the drawing, the checking procedures and so on. The UDAs shown in the table could be at a pipe, branch or drawing level. For this you use the POSITION REVISIONTABLE command.
- **System Tables** include the pipe attributes for a for a drawing or system isometric. You can show pipe attributes (including the name) and UDAs in the table. This table is defined using the POSITION SYSTEMTABLE command.

7.13 Summary of Commands

The following commands were described in this chapter:

ATEXT	Allows you to change the standard text on isometric drawings.
CUTMARKS	Draws cut marks at each corner of the overall drawing sheet.
DATE	Allows you to specify the format for showing the date in the drawing's title block.
DISTANCES	Allows you to specify the units to be used for input of lengths and distances.
FRAME	Specifies whether or not drawing frame lines and text are drawn.
ISO	Defines the view direction to be used for generating the isometric plot.
ISOTYPE	Defines whether the isometric will show fabrication information, erection information or both.

MARGIN	Specifies the width of the margin on each side of the drawing.
MARK	Allows you to define whether the pipe revision attribute (REV) should be incremented and the Branch detail attribute should be set to true after successful detailing.
PLOTDETAILDATA	Allows you to define whether and how to display parameterised attribute data in the detail plot files.
PLOTTINGSCALE	Defines the scale of the plot independently of the drawing sheet size.
PROJECTNUMBER	Writes the project number into the title block area.
RESERVE	Allows you to specify a blank area at the bottom of either the drawing or material list areas.
SIZE	Specifies the overall dimensions of the drawing sheet.
STACKINGARRANGEMENTS	Allows you to define how to position multiple plots on a single sheet.
TITLEBLOCK	Allows you to add extra text to the title block.
UNDERLAYPLOT-NAME	Allows you to define a background plot file; letting you plot directly on top of this background plot.

8 Material List and Other Reports

This chapter describes how you can customise the material list, whether it is produced as part of the isometric plot or separately.

You can assign unique part numbers in OUTFITTING DESIGN to provide full material traceability. or you can assign part numbers in ISODRAFT on the drawing (however, these numbers are not maintained between revisions of the drawing).

ISODRAFT can give you the following reports:

- A cut pipe list, on the sheet with the material list or separately
- A pipe support schedule
- Pipe and insulation summaries

You can create a specific material list on the isometric that contains only the information you're interested in, and only in the order you want. This user defined material list allows you to select which standard columns are to appear on the list (the standard columns being part number, description, item code, nominal size, quantity and remarks) and which order the selections should be displayed in. You can choose to omit certain columns, and specify which component and SPCOM attributes/UDAs for selected components will be output in the remarks column. You can also choose extra columns to be shown on the list that are filled with chosen attribute data.

By default, a material list will be plotted on the left-hand side of the isometric drawing.

In a combined fabrication and erection isometric, shop fabrication materials will be separated from site erection materials. Note that the shop/site information for fabricated components is obtained from the specification via the design. The shop/site information for tubing is set directly from the specification, because tubing is only implied in the design. See [Tube SHOP Flag](#) for more details about the Tube SHOP flag.

Components are grouped under the headings Pipe, Fittings and Flanges, Gaskets, Bolts, Valves/In-line Items, Instruments and Supports.

The entries in the material list are numbered sequentially. These part numbers are used for cross-references to the drawing area. Each entry contains the material's description, nominal size, item code and quantity.

The MATERIALLIST command allows you to control the following features of the material list:

- Whether or not the list is shown on the drawing at all (default is for the list to be plotted).
- Whether a material list file is produced for each drawing, or one list file is produced for all drawings.
- Whether the fabrication section of the material list is sorted by Spool or not.
- Whether any items should be excluded from the material list and, if so, what items.

- The position of the list on the drawing sheet area (default is left-hand side).
- The size of the characters used for plotting the list.
- The spacing between the lines in the list.
- The width of the Item Code column in characters (default is 8 characters).
- Whether or not component descriptions are included in the list (default is for descriptions to be included).
- How the overflow of long lists is to be handled (default is for the list to be continued on a separate sheet with a blank drawing area).

8.1 Part Numbers

If part numbers have been set in the OUTFITTING DESIGN database using the Spooling Application, you can specify that database part numbers are used, or that ISODRAFT will generate part numbers automatically. You can:

- Set up user-defined columns for the material list;
- Choose whether part numbers should be displayed separately for each spool;
- Choose whether to output an individual entry for each component, tube length and bolt set, or whether the part numbers for like items will be totalled.

If you are using totalled database part numbers, multiple part numbers will be output, separated by spaces, in a single entry in the material list, wrapping round if the column width is exceeded. The quantity column will show the total of like items. Part number tags will still be individual.

8.2 Changing the Text in Material List Descriptions

A complete item description in a material list consists of the geometric description (derived from the attributes of a DTEXT element) and the material description (derived from the attributes of an MTEXT element).

For example, a material list description might be a combination of the DTEXT

```
ELBOW 90 - BUTT WELD SCH.40
```

and the MTEXT

```
ASTM A105 FORGED
```

giving a complete description of

```
ELBOW 90 - BUTT WELD SCH.40  ASTM A105 FORGED
```

The text in MTEXT and DTEXT elements is defined in PARAGON by setting the attributes of MTEXT and DTEXT to the text required. Both elements have three attributes which can contain text:

- The three attributes of DTEXT are RTEXT, STTEXT and TTEXT.
- The three attributes of MTEXT are XTEXT, YTEXT and ZTEXT.

The default attributes whose text will be displayed on the material list are RTEXT and XTEXT. If the other attributes have been set in the OUTFITTING DESIGN database, you can choose to display the STTEXT or TTEXT using the DTEXT command, and the YTEXT or ZTEXT using the MTEXT command.

Different attributes can be used, for example, to produce descriptions in different languages.

The MTEXT elements in item descriptions can be suppressed completely by the MTEXT IGNORE command. For example:

```
ELBOW 90 - BUTT WELD SCH.40  ASTM A105  FORGED
```

with MTEXT elements suppressed would be shown as

```
ELBOW 90 - BUTT WELD SCH.40
```

Similarly, the DTEXT elements can be suppressed using the DTEXT IGNORE command.

8.3 Excluding items from the Material List Descriptions

ISODRAFT allows you to exclude selected items from the material list. This facility is particularly useful for modelling items that are to be bought in as single units. For example, you could represent such a bought-in component by combining two branch members. In this case, you would want only one entry on the material list. A specific example would be if you modelled an orifice flange assembly using FTUBs as tapped connections. Here you would not want the FTUBs appearing on the material list.

The exclusion of an item is determined by the system attributes MTOC and MTOT for branch members and leave tubes, and the attribute MTOH for branch head tube. The settings you can use are:

ON	Include on the material list and draw normally.
OFF	Exclude from the material list but draw normally.
DOTD	Exclude from the material list and draw dotted and dimensioned.
DOTU	Exclude from the material list and draw dotted but undimensioned.

8.4 Item Codes

8.4.1 Specification Names in Item Codes

By default, the item code is the SPCOM with the specification name and leading / omitted. For example, an SPCOM set to RF300/100CRMM would give the item code 100CRMM.

To use the full SPCOM as the item code, use the ITEMCODE LENGTH command. Note that using the full SPCOM will probably require a larger setting than normal for MATERIALLIST CODE, which sets the width of the column for the item code on the material list.

You can set up multiple item codes for elements in the OUTFITTING DESIGN database by creating user-defined attributes (UDAs) for the SPCOM of the element. For bolts, the UDAs are created as attributes of a DTAB element. You can then use the ITEMCODE command to specify that the item code should be taken from a UDA instead of from the SPCOM.

8.4.2 Using Names As Item Codes

Instruments are a special case. The instrument name (e.g. FCV-70) will be incorporated into the description instead of being plotted in the item code area, because an instrument name may require more characters than the item code format permits. (The description is

normally longer than the space allowed for the item code, and it will wrap onto the next line if necessary.)

The material list item code for an instrument may be defined as either its OUTFITTING name or its specification reference (SPREF) using the INSTRUMENTS command.

- When the NAME option is specified, the name will be appended to the description and the item code column will be blank.
- When the SPREF option is specified, the specification reference will be listed in the item code column.

Component types that can be **tagged**, that is types that can have their names plotted automatically on the isometric drawing, can be treated like instruments. The types that can be tagged are:

INST	VALV
FILTER	PCOM
TRAP	VENT
VTWA	VFWA

If you want a type of component to be treated like an instrument, you must give the component type a **key number**. The key number is related to its symbol key (SKEY) using the KEYS command.

8.4.3 Item Code Suffixes

OUTFITTING does not allow any name or specification reference (SPREF) to exist more than once. Identical components can be distinguished by having a suffix added to their SPREFs. ISODRAFT normally removes suffixes from item codes in the material list by recognising the **delimiter** character which separates the suffix from the rest of the SPREF, and then totals the items together. Items with the same item code but different descriptions can be listed separately using the ITEMCODE SEPARATE command. Alternatively, you can change the delimiter character using the DELIMITER command. ISODRAFT will not recognise the suffix and so will no longer remove suffixes from item codes in the material list.

8.5 Cut Pipe Lengths

A list of cut pipe lengths can be appended to the material list by using the CUTTINGLIST command.

Each separate pipe length is identified by:

- Its cut piece number, shown enclosed in angled brackets on both the isometric and on the cut pipe list.
- The cut length in mm or feet and inches depending on the units set.
- The pipe bore.

There is also a remarks column, which is used, for example, to note when a cut pipe length contains a field fitting allowance or pulled bend.

You can also include the item codes as a column in the list.

8.6 Filing Material List Data for Printing

8.6.1 Filing the Full Material List

As well as, or instead of, plotting the material list on the isometric drawing sheet, you can send it to a text file by using the MATLISTFILENAME command. You can specify that one file is generated per plot, or that one file is generated for all plots. The file can then be printed and edited for other purposes.

Material list numbers are shown on the isometric plot even when the material list is sent only to a file.

8.6.2 Filing a Pipe Support Schedule

The SUPPORTFILENAME command sends a pipe support schedule to a file.

Note: For system isometrics, no information will be written to the support file.

8.6.3 Pipe and Insulation Summary Files

In addition to the full material list, two types of summary can be output to file using the CLLENGTHFILE command:

- A pipe centreline length summary;
- A pipe plus insulation centreline length summary.

Note: That pipe and pipe plus insulation summaries cannot both be produced in a single run. If either type of file is specified, the other is automatically suppressed.

8.6.4 Cut Pipe Report Generation

A cut pipe list report can be output to a file using the CUTPIPELISTFILE command. The information output for each length of pipe consists of:

- Item code
- Item description
- Bore
- Cut length
- Calculated length (ignoring end preparation allowances etc.)
- Special requirements (e.g. pulled bends, loose flanges, field-fit welds)
- Cut piece number
- Batch reference (i.e. the Zone name)

8.6.5 Pipe Wastage Allowance

When cut pipe lengths are calculated, ISODRAFT can include extra material to allow for wastage during cutting and fabrication.

A model is split up into a maximum of ten **wastage areas** and each wastage area is given a **wastage factor** by using the WASTAGE AREA command.

8.6.6 Field-Fit Weld and Loose Flange Allowance

ISODRAFT automatically includes a specified allowance in each calculated cut pipe length associated with a field-fit weld or a loose flange. See [Field-fit Welds and Loose Flanges](#).

8.7 Bolting Information

The BOLTING command specifies the following:

- Whether bolting information is shown on the isometric at all.
- Whether bolt lengths will be included in either or both the item description and item code columns.
- The units flag for bolt dimensions in the Material Control File, which is an ASCII file containing a record of all materials incorporated into the isometric drawing. It is described in more detail in the chapter *Material Control File*, of the [ISODRAFT Reference Manual](#).

The default shows bolting information in the item description and item code columns.

8.8 Material Lists for Split Pipelines

Where a pipeline is split over two or more drawings, a separate material list is created for each drawing. The component serial numbering sequence restarts from 1 on the material list of each drawing sheet.

Note: That you can control how pipelines are split. See [Using the SPOOLER Module](#), which describes how split points are created in a OUTFITTING DESIGN module, and [Splitting Long Pipelines](#), which describes how you can control splitting in ISODRAFT.

8.9 Summary of Commands

The following commands were described in this chapter:

CUTPIPELIST

CUTTINGLIST

DELIMITER	Specifies which character ISODRAFT recognises as the itemcode suffix delimiter.
-----------	---

DTEXTS	Controls the material description part of the item description.
--------	---

EQUIPMENTNUMBERING	Controls numbering of items on materials list for an equipment trim isometric.
--------------------	--

INSTNAME	Allows you to define how instruments are described in the materials list.
----------	---

ITEMCODE	Allows you to label certain types of fitting with their itemcodes on the isometric drawing.
----------	---

KEYS	Allows components with specified SKEYs to be treated like instruments.
------	--

MATERIALLIST	Allows you to control whether the material list is drawn on the isometric and alter the display format if required.
MATLISTFILENAME	Specifies the name of the file into which the materials list can be saved.
SUPPORTFILENAME	Specifies the name of a file into which the schedule for pipe supports is saved.
WASTAGEAREA	Defines wastage areas that can be given different wastage factors for cut pipe lengths.

9 Isometric Drawing

This chapter describes how ISODRAFT outputs isometric drawings, including the default details ISODRAFT uses and how to customise them.

9.1 Setting the Detail and Revision Flags

ISODRAFT can indicate that it has completed processing a pipeline by:

- Setting the detail attributes (DETA) of the branches processed to TRUE.
- Incrementing the revision attribute (REV) of the pipe.

The updates are made only after the pipeline has been detailed successfully. The commands which specify the options required are MARK DETAIL and MARK REVISION. Either or both options can be specified. The attributes are not set by ISODRAFT; an inter-db connection macro is generated to set them in OUTFITTING DESIGN.

If any branches of a pipe have their DETA attributes set to TRUE, ISODRAFT will assume that detailing of that pipe has been completed and will not plot the pipe unless it has been told to ignore the detail flags. The command is MARK IGNORE.

9.2 View Direction

The default isometric view is the North arrow pointing to top left. The ISO command is used to change the view so that the North arrow points to the top right, the bottom right or the top left.

The BOX option plots a box round the north arrow.

9.3 Origins of Coordinate Axes

By default, all spatial coordinates on isometrics are given in the World coordinate system. The AXES command allows you to specify a different coordinate system for the isometric. You can use the origin of any database element which has an origin.

Each owning element (Site, Zone etc.) in a design project has its own coordinate system for locating the elements it owns. The coordinates of an element are usually defined with respect to its owner, and these coordinates are *not* normally the same as World coordinates.

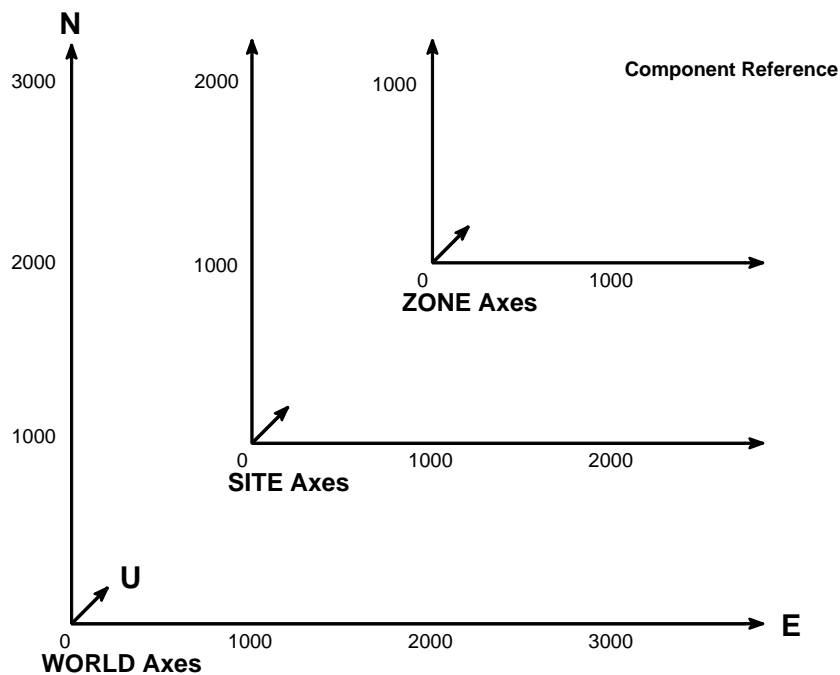


Figure 9:1. Coordinate systems within a design project

With reference to [Figure 9:1.: Coordinate systems within a design project](#), the E/W and N/S coordinates of the piping component are:

E3000	N3000	with respect to the World axes
E2000	N2000	with respect to the Site axes
E1000	N1000	with respect to the Zone axes.

[Figure 9:1.: Coordinate systems within a design project](#) illustrates this in two dimensions, but the same principle applies to the elevation axis.

Spatial coordinates are shown on the drawing, by default, at all pipe termination or connection points. You can control when coordinates are plotted by using the COORDINATES command. You can choose to switch off coordinates of end points which are connected to Nozzles, continued on other Pipes or which have connection type Open, Clos, Vent or Dran.

9.3.1 Elevations

By default, elevations are only shown at intersection points when there is a change of elevation. Elevations can be shown on vertical sections of pipe by using the ELEVATION command.

9.4 Improving Legibility of the Isometric

ISODRAFT tries to balance the amount of information shown on a drawing sheet and its legibility. If the default is not satisfactory, you can improve the legibility by increasing the character size on the drawing area, and by altering the picture scale.

- The character size for the annotating text on the isometric drawing can be specified using the CHARSIZE command. Note that the character size for the material list is specified using the MATERIALLIST command.
- The scale of the isometric area can be changed by using the PICTURESCALE command. Note that this does not scale the whole isometric, but just affects the size of the symbols and annotation relative to the length of pipe shown. The material list is not affected.

9.4.1 Message Layout

You can control how arrowed messages on the isometric are positioned on the isometric using the MESSAGEACCURACY command.

The MESSAGEACCURACY RADIAL option determines the accuracy level of the radial search, based on the number of radial directions searched away from the arrowhead, the number of additional directions for kinked lines, and the length of increments of leader lines. The levels are as follows:

Level	Radial directions searched away from arrowhead	Additional directions or kinked lines	Length increment of leader line
1	18 directions @ 20 degrees	2 directions @ 80 degrees	4mm
2	36 directions @ 10 degrees	4 directions @ 40, 80 degrees	2mm
3	72 directions @ 5 degrees	8 directions @ 20, 40, 80 degrees	1mm

Once all messages have been positioned, the MESSAGEACCURACY ITERATIONS option repositions any messages with crossing leader lines, and all messages with leader lines passing through the text block. You can set a maximum number of iterations, and the process is repeated until one of the following conditions is satisfied:

- No leader lines cross and no messages need to be moved.
- The number of leader lines crossing and messages moving is the same as the previous iteration.
- The maximum number of iterations set by the option is reached.

You should start with accuracy level 1. Higher accuracy levels are achieved at the expense of performance. Increasing the number of iterations is not as expensive as higher accuracy levels, as often an equilibrium is reached before the maximum number of iterations. Thus it is recommended that the effect of increasing iterations be examined before increasing the accuracy level.

9.5 Splitting Long Pipelines

If a pipeline is too long or complex to fit on a single isometric plot, ISODRAFT will automatically split the pipeline over separate drawings. The split may occur in a main line or an off-line section. Each drawing will be identified as DRG 1 OF ..., DRG 2 OF ..., etc. You can control the style of numbering using the MATERIALLIST command.

You can control the complexity of each drawing, or the number of drawings into which the isometric is to be split, by using the SPLIT command.

When a pipe is continued on another drawing, the pipe end will carry the message CONT. ON followed by the identification of the next drawing.

When a pipe is continued from a preceding drawing, the pipe end will carry the message CONT. FROM followed by the identification of the preceding drawing.

ISODRAFT may generate splits in tube. You can use the TUBESPLITS command to generate the message UNACCEPTABLE SPLIT POINT when splits in tube occur, or to stop isometrics with splits in tube being produced at all.

9.5.1 Specifying Split Points

You can specify exactly where pipes should be split by using **split ATTAs** in the OUTFITTING DESIGN database. The principles are described in [Design Requirements](#). If user-defined positions for splits have been created in the design, ISODRAFT will always use these and not calculate its own split points.

Note: Although ISODRAFT will sometimes split isometrics between flanges automatically, it is important that you do *not* position any ATTAs between flanges or flanged fittings. To do so could lead to corruption of the ISODRAFT Material List output.

9.5.2 Repeatability

When a pipe is detailed which needs splitting over more than one sheet, ISODRAFT automatically generates split points. By default this data is not stored, so if the design is altered and the pipe detailed again, the split points may be in different places.

If you use the REPEATABILITY command, sheet split points are written back into the OUTFITTING DESIGN database. This means that when the pipe is detailed again it will be split in the same places. You can reposition the split points if you wish. Repeatability allows you to re-plot single sheets when you have made changes to your design which only affect those sheets. Repeatability is not available for system isometrics or equipment trims. The split points are not set by ISODRAFT; an inter-DB connection macro is generated to set them in OUTFITTING DESIGN.

Restrictions on Repeatability

- Split ATTAs will not be automatically created if some already exist in the pipe.
- Re-ordering branches will destroy the integrity of the repeatability data.

For a major redesign, you should remove repeatability data from the database and regenerate it.

If repeatability has been set, individual spools and sheets can be re-plotted using the DETAIL command. Individual sheets and spools can be entered into the Add and Remove lists using the ADD and REMOVE commands.

9.6 Dimensioning

ISODRAFT can output several types of dimensions with different amounts of detail. The main types of dimensioning are:

- String dimensions (the default).
- Composite dimensions.
- Overall dimensions.
- Support dimensions.

The different types are illustrated in [Figure 9.2.: Support, string and overall dimensions](#).

String dimensions are measured between the following points:

- The start or end of a branch
- Changes of direction (bends, elbows etc.)
- Component connection points (flanges, welds, screwed and compression joints etc., excluding gaskets)

Composite dimensioning ignores dimensions which the fabricator has no control over, particularly the sizes of individual components. Components which are *directly* connected to each other are therefore dimensioned as though they constitute a *single* composite item.

Overall dimensions indicate the lengths of the following sections of a pipeline:

- From the start of a branch to the first change of direction
- From each change of direction to the next
- From the last change of direction to the end of a Branch
- From the start of a falling section skew-box to the end of the falling section skew-box.
- Between Branch connections.

You can specify:

- Whether or not overall dimensions cross Branch connections
- Dimensioning to Valve centrelines

You can specify that overall dimensioning is to **critical components**. This can be used when the exact lengths of all components are not known at the time of design, for example, if the purchase of fittings has been delegated to a local fabricator. Critical dimensions are between points on assemblies of connected components. The points are chosen according to the first rule satisfied from the following list:

- Branch connection, such as Tee or OLET, but excluding tappings on PCOMs and Flanges.
- Change of direction, such as Elbow or Bend.
- Face of Flange or flanged component.
- Centreline of Instrument.
- Centreline of Valve.
- Origin of first component in assembly.

Single components in tube will be dimensioned to their origin; for example, Valves to their centreline.

Critical dimensions can be shown with or without component dimensions.

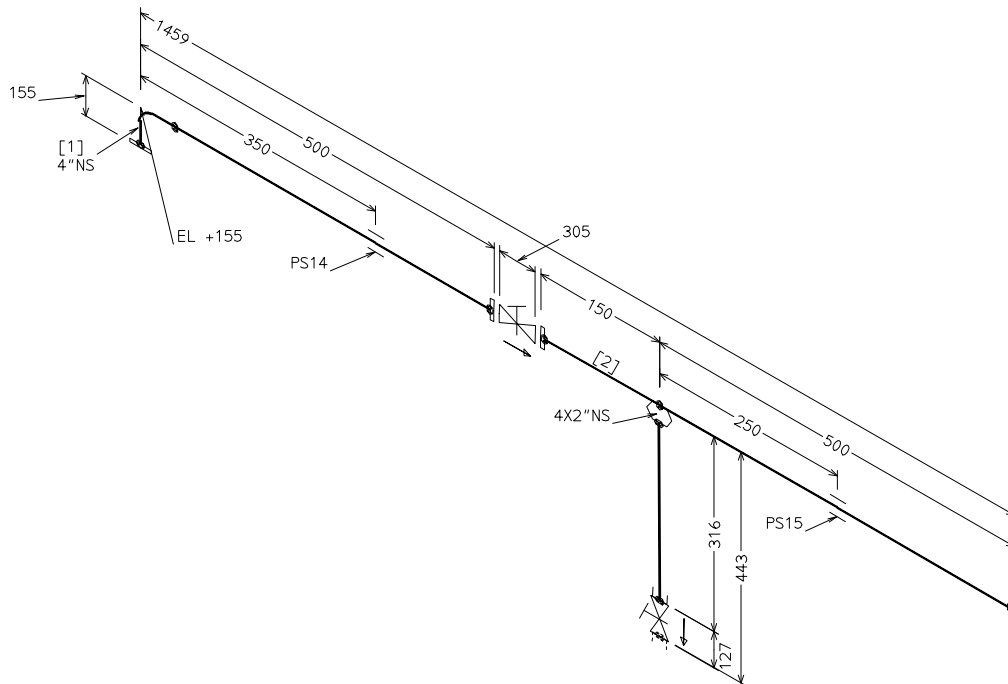


Figure 9.2. Support, string and overall dimensions

The DIMENSIONS command is used to set the dimensioning options. The default settings are:

- String dimensioning on (and, therefore, composite dimensioning off);
- Overall dimensioning off;
- Support dimensioning on;
- Valve centreline dimensioning off;
- Support/hanger dimensions shown alongside other pipeline dimensions.

Isometrics with no dimensions plotted can be produced.

9.6.1 Support/Hanger Dimensions

Support positions can be dimensioned in one of two ways, overall or string. The default is string, and this can be changed using the SUPPORTS command.

Support information may be omitted completely from the isometrics.

By default, **support/hanger dimensions** are positioned with a standout of 6mm and **overall dimensions** are positioned with a standout of 16mm on the *same* side of the pipe as string dimensions (see [Figure 9.2.: Support, string and overall dimensions](#) and [Figure 9.3.: Dimensioning line standout distances \(default standout distances\)](#)).

You can use the DIMENSIONS HANGERS command to position support/hanger dimensions at one standout on the opposite side from string dimensions (see [Figure 9.3.: Dimensioning line standout distances \(default standout distances\)](#)).

9.6.2 Bend Radii

Bend radii will be output, by default, for all pulled bends (that is for all components with SKEYs beginning with PB). Bend radii can be shown either in **distance units** or as **multiples of pipe diameters** (where pipe diameter is taken as the nominal bore of the pipe) by using the BENDRADIUS command. Output of bend radii can also be suppressed.

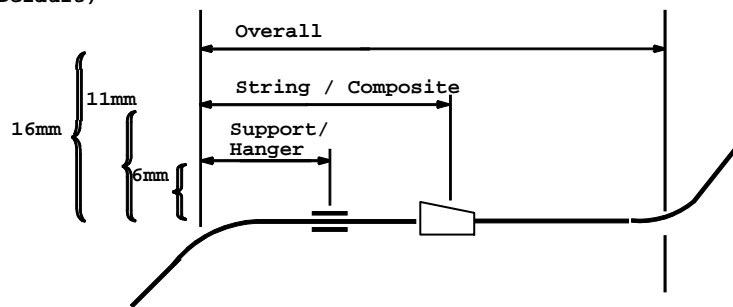
A **standard radius** for bends can be set using the BENDRADIUS STANDARD command. It can be positioned as a note on the isometric plot using the POSITION command. If a standard radius is set, only bends which have radii different from the standard radius will have their radii shown individually. Both the standard radius and any individual radii will be shown in the same format (i.e. in distance units or in pipe diameters).

Note that only the actual value of the standard radius will be output. Any other text, such as:

All pulled bends to be D unless otherwise shown

where the value is positioned in the space immediately before the D, must be plotted by some other method, such as an underlay. See the UNDERLAYPLOTFILE command.

(a) Dimensions Hangers
(Default)



(b) Dimensions Hangers
(Opposite Strings)

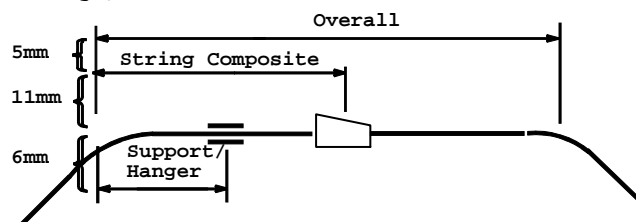


Figure 9.3. Dimensioning line standout distances (default standout distances)

9.6.3 Dimensional Units on Isometrics

The units for output are specified independently from the units for input. By default, ISODRAFT outputs metric lengths and imperial bores, but these can be changed so that output is all imperial units or all metric units using the UNITS command.

You can also choose whether bores are Nominal or Actual using the PRECISION BORES command.

9.6.4 Rounding and Truncation of Dimensions

When dimensioned isometrics are plotted, all dimensions are rounded up or down to the nearest mm or 1/16 inch (depending on the units set by the UNITS command). By default, ISODRAFT carries over the remainder (positive or negative) from a rounded dimension and adds it to the next dimension. Carrying over the rounding remainder reduces cumulative errors over a number of consecutive dimensions, but it can result in identical fittings being shown with different lengths (usually differing by 1mm or 1/16 inch).

If you wish to suppress this carry over effect, you can specify that ISODRAFT **truncates** all dimensions. The remainders from rounding of the individual dimensions will then be discarded rather than added to the next dimension. To specify truncation, use the TRUNCATE command.

9.6.5 Positions of Dimensioning Lines

The default standout distance between string/composite dimensioning lines and the pipe runs to which they apply is 11mm. The default overall and support dimension distances are 16mm and 6mm, respectively. The standout distances can be changed separately for string, overall and support dimensions by using the DIMDIST command. They may not be set to the same value.

9.6.6 Reference Dimensions

If reference dimensions have been set up in the OUTFITTING DESIGN database, they can be switched on using the REFDIMENSIONS command. See [Reference Dimensions](#) for information about how to set up reference dimensions in the OUTFITTING DESIGN database.

9.7 Support Attachment Point Names

By default, the names of support attachment points (that is, all ATTAs for which the ATTY attribute is unset) are not shown on the isometric plot and their SPREFs (specification references) are used as their item codes.

You can use the ATTANAME command to display the OUTFITTING names of support ATTAs on the drawings, and to specify whether their OUTFITTING names or their SPREFs are used as their item codes.

9.8 Tolerances In Isometric Plotting

By default, two components will be plotted as though they have a common axis if the orientation of their axes differs by less than 0.06 degrees, or if their axes are offset by less than 1.00 mm.

The TOLERANCE command allows you to specify a different alignment tolerance, either as an angular difference or as a linear offset.

9.9 Flow Arrows

Two types of flow arrow can be plotted on isometric drawings:

- Arrows alongside in-line components (Valves etc.)
- In-line arrows at specified points in Tube, if these have been defined in the OUTFITTING DESIGN database.

You can control whether each type is plotted, and the inline flow arrows can be scaled, using the FLOWARROWS command.

9.10 Plotted Leg Length

By default, the plotted leg length of Elbows, Tees and Crosses is 9mm. You can change this value by giving the LEGLENGTH command. The minimum leg length allowed is 6mm.

9.11 Skewed and Falling Pipelines

9.11.1 Skewed Lines

Skewed pipelines are shown by **skewboxes** (that is by boxing in the skewed length with dimensioning lines which show the horizontal and vertical differences between the end coordinates). The skewboxes can be full boxes or triangles, as specified by the SKEWBOX command.

If triangles are specified, five further options are available:

- The dimensions can be shown in standard form or in a non-standard form
- The triangle areas can be shown hatched (shaded) or un-hatched
- Swing angles can be displayed in the corners of the triangles
- H and V notes can be shown in the right angle corners of horizontal and vertical triangles
- Overall skewboxes, or skewboxes between branches can be shown

Skewbox triangles can have two styles of dimensioning:

- **Standard dimensioning** shows the horizontal and vertical dimension lines of the triangle standing out from the sides of the triangle.
- **Non-standard dimensioning** shows the dimension characters positioned along the sides of the triangle itself. The default is standard dimensioning.

By default, skewbox triangles are highlighted by parallel-line hatching with a spacing of 3mm between the hatching lines. The hatching can be switched off, or the spacing altered.

You can specify whether triangular skewboxes are shown as overall skewboxes across branches, or as separate skewboxes between branch connections using the OVERALLSKEWS command.

9.11.2 Falling Lines

Falling lines are shown by skewboxes in the same way as other skewed lines. If the amount of fall is less than the current fall limit, the fall is shown by a wedge-shaped symbol on the

pipe showing the direction of fall, and the word FALL followed by the amount of fall in degrees. The symbol is known as a fall arrow.

By default, the fall limit is 5°. This limit can be changed, or expressed in different units using the FALLINGLINES command. The accuracy of reporting the fall amount can be varied.

The fall arrow and slope method can be switched off completely, and *all* falls will then be shown as skewboxes.

9.11.3 Skewed and Falling Lines

By default, skewed and falling lines are showed by 3D skewboxes (that is one skewbox to show the fall and one to show the horizontal skew).

Skewed and falling lines with falls less than the current fall limit can be shown by an arrow on the pipe showing the direction of fall, and the word FALL followed by the amount of fall. A single (2D) skewbox shows the horizontal skew in this case. See the FALLSKEWS command.

You can also suppress skewboxes showing falls for branch legs which have less than a given number of components.

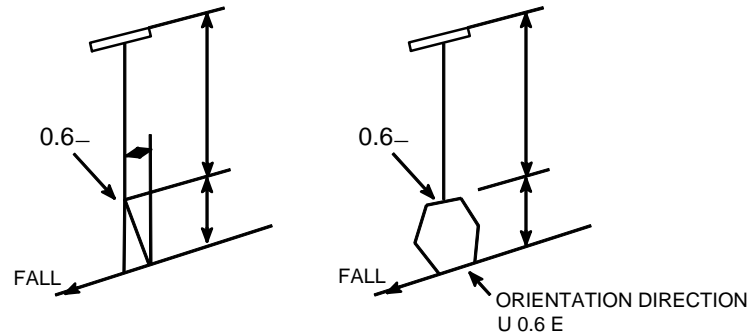
9.11.4 Zero-length Bends at Tees and OLETs

When modelling falling lines containing Butt Weld Tees or OLETs, you can position a zero-length variable-angle bend at the off-line p-point of each Tee or OLET. The use of zero-length bends allows you to connect a vertical Branch to a falling line. To suppress the output of skewbox dimensions at zero-length bends in these circumstances, use the command ZEROLENGTHSKEWS.

The default setting ZEROLENGTHSKEWS ON, which displays Tees and OLETs and the effect of specifying ZEROLENGTHSKEWS OFF are shown in [Figure 9:4.: Suppressing skewboxes at Tees and OLETs](#).

Note: Bends at Tees or OLETs will only be considered as zero-length if they are less than 0.5mm long. Bends with dimensions greater than this will be detailed as normal components.

(a) ZEROLENGTHSKEWS ON(Default)



(b) ZEROLENGTHSKEWS OFF

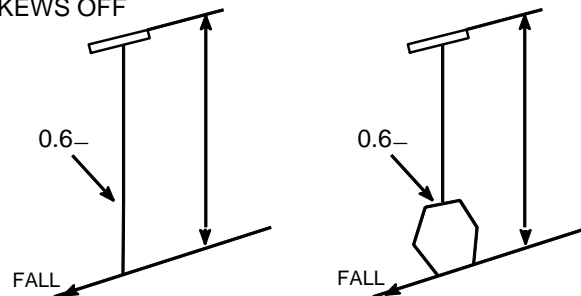


Figure 9.4. Suppressing skewboxes at Tees and OLETs

9.12 Insulation and Tracing

Any pipe which has its Insulation Specification (ISPEC) and/or Heat Tracing Specification (TSPEC) attributes set can be identified on the isometric drawings as shown in [Figure 9.5.: Insulated and traced piping](#).



Figure 9.5. Insulated and traced piping

The start and end points of insulation on a line can be indicated on a drawing as shown in [Figure 9.6.: Insulation Start and Stop symbol example](#).

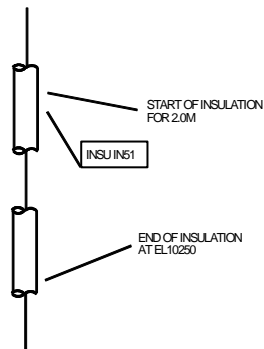


Figure 9.6. *Insulation Start and Stop symbol example*

The insulation specification note will be shown on the drawing at the start points of any insulation.

The name of the tracing specification will be shown in the title block.

Insulation and tracing can be shown either on tubing only, or on tubing and pipe components.

The commands to control the display of tracing and insulation are INSULATIONCONTROL and TRACINGCONTROL.

9.13 Marking Specification Changes

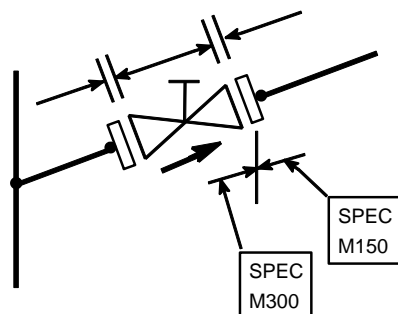
The specification reference for a pipeline will appear in the bottom right-hand corner of the isometric drawing, immediately above the pipeline reference.

Pipeline specification changes are indicated by notes against the corresponding points on the isometric plot. By default, specification changes are noted at component level, but they can be noted at branch level by using the SPECBREAK command.

Whenever a specification change is shown on the drawing, both the original and new specifications at each change point are plotted. The format is similar to that used for dimensioning.

The format is illustrated in [Figure 9.7.: Marking a change in pipeline specification](#).

Insulation and tracing specification changes can also be shown.



Specification change format

Figure 9.7. Marking a change in pipeline specification

Whenever a change of specification is encountered in a pipeline, the information is recorded in the IDF file and a specification change label is output at the appropriate point on the drawing.

Ten types of specification change are catered for:

- Pipeline name (for system isometric only)
- Piping material specification
- Insulation specification
- Tracing specification
- Painting specification
- User defined specifications (five, in ISODRAW only)

A maximum of ten changes are allowed at a single point.

Each specification type has an associated, user-definable ATEXT which specifies the text printed alongside each specification. See the SPECBREAK command in the chapter *Command Reference Section*, of the *ISODRAFT Reference Manual*.

9.14 Plotting Complete System Isometrics

9.14.1 System Isometrics

Normally, ISODRAFT will process each pipe or spool drawing specified in a DETAIL command separately. However, they can be detailed as a complete system isometric. See the DETAIL command for more information.

9.14.2 Equipment Trims for Material Take Off

Unconnected items cannot be drawn automatically as part of a system isometric, vessel or equipment trim. However, drawings consisting of a drawing frame, user-positioned text and a material list can be produced for material take off. The graphical part of the drawing can then be added separately if required.

Groups of components such as blanking flanges, level gauges, relief valves, etc., may be modelled on the basis of one *Pipe* per Nozzle, or per group of Nozzles if connected (such as a level gauge piped between two Nozzles). Disconnected branches in the same pipe on different nozzles cannot be grouped. The collection of pipes may then be detailed as equipment trim.

By default, item numbering in the material list will start at 1 for each Nozzle. To get continuous numbering throughout the sequence of Nozzles, use the EQUIPMENTNUMBERING command.

9.15 Welds

9.15.1 Weld Display

The following weld types have symbols associated with them. They can therefore be shown on isometrics:

WS	Site (erection) weld
WF	Field-fitted weld
WW	Shop (fabrication) weld

By default, all types of weld are shown on isometrics. You can use the WELDTYPE command to specify that:

- Only shop welds will be shown;
- Only site welds will be shown;
- No welds will be shown;
- All welded fittings will be shown as having shop welds.

9.15.2 Weld Numbering

You can assign unique numbers automatically to welds in the SPOOLER module, thereby providing complete weld traceability. Numbering can be applied to an individual pipe, branch or spool drawing. These unique identities are stored in the OUTFITTING DESIGN database, and are available for you to use in ISODRAFT (see the WELDNUMBERS command). If you use this feature, you can also create a table of all numbered welds (according to your own specifications) to be included on the isometric.

If you use weld numbers from the OUTFITTING DESIGN database, temporary weld numbers can be output by ISODRAFT at end points where the weld actually belongs to another drawing. These temporary numbers are also included in the weld table. This will allow the fabricator to identify the end preparation to be applied to the termination joint even though the weld is not to be made for this drawing. Temporary weld numbers will include the weld prefix but will use letters instead of numbers.

If you do not use the automatic weld numbering feature of SPOOLER, you can number the welds on an isometric within ISODRAFT using the WELDNUMBERS command (however, numbers generated this way are not maintained between revisions). You can also use the WELDNUMBERS command to specify that a table of all numbered welds will be drawn immediately above the title block.

Additional welds can be shown on the isometric either in individual balloons or, if there are more than a specified number of them, as an arrowed message in a single ellipse (with the numbers separated by slashes). You specify this by appending `AWELD BALLOON number` to the `WELDNUMBERS` command.

Either with numbers from the `OUTFITTING DESIGN` database, or generated within ISODRAFT, you can specify the size of the characters for the numbers, and whether fabrication or erection welds or both are to be numbered.

Note that by combining the `DIMENSIONS OFF` and `WELDNUMBERS` options it is possible to obtain un-dimensioned isometric drawings with weld numbers on them. These are useful for recording the results of weld radiography inspections.

9.15.3 Weld Identification

Welds which have their `NAME` attribute set in the `OUTFITTING DESIGN` database can have their names plotted against their symbols on the isometric by using the `WELDID` command.

9.16 Item Codes and Name Tags on Isometrics

9.16.1 Item Codes

By default, item codes for piping components are shown only in the material list.

For in-line fittings of generic types `INST`, `VALV`, `FILT`, `PCOM`, `TRAP` or `VENT`, you can specify that item codes are also shown against the corresponding component symbols on the isometric plot by using the `ITEMCODE` command.

9.16.2 Name Tags

By default, component names are not plotted. Fittings of generic type `INST`, `VALV`, `FILT`, `PCOM`, `TRAP`, `VTWA`, `VFWA` and `VENT` can have their `OUTFITTING` names (if they have them) shown against their plotted symbols on the isometric by using the `TAGGING` command. (See also the `KEYS` command.)

9.16.3 User-defined Tag Boxes

You can use the `OUTCOM` command to output piping component attributes and user-defined attributes on the isometric drawing area. You can choose whether the text is boxed or unboxed, and the style of box used. You can also output empty boxes for manual mark-up. Six styles are available:

- A rectangular box with straight ends
- A rectangular box with angled ends
- A rectangular box with round ends
- A triangular box
- A diamond-shaped box
- Unboxed

9.17 Pipe Ends and Connecting Nozzles

9.17.1 Suppression of End Coordinates

The output of end coordinates may be suppressed at locations such as:

- Pipeline continuations to other drawing sheets;
- Nozzle connections;
- End connection types OPEN, CLOS, VENT or DRAN.

To specify whether or not end coordinates are to be shown at one or more types of location, use the COORDINATES command.

9.17.2 Nozzle Continuation Symbols

On combined or erection-only isometrics, dotted nozzles can be plotted at points where connections are made to ship equipment, using the CONTINUATIONSYMBOLS command.

9.17.3 Continuation Notes

You can determine whether continuation notes should be included on the isometric for when drawings continue onto another isometric. The notes can include the names of pipes, drawings or both (as defined by the OUTFITTING DESIGN Spooling application) using the CONTNOTES command. If the termination point on the current isometric is a weld, you can include a description of the connecting component or tube in the notes. The notes will tell you where the continuation is made to, and can include the relevant description.

You can also use the CONTWELDS command to include temporary weld numbers from continuing drawings on the isometric (whether they refer to pipe name, drawing name or both).

9.18 Summary of Commands

The following commands were described in this chapter:

ANGLEACCURACY	Allows you to specify the degree of accuracy for the reporting of angles.
ATTANAME	Specifies the ways in which support ATTAs are identified.
AXES	Specifies the coordinate system to be used for showing 3D positions on drawings.
BENDRADIUS	Specifies when radii of pulled bends will be shown and in what units.
CHARSIZE	Specifies the character size to be used for annotating text on drawings.
CLLENGTHFILE	Specifies that a pipe centreline length summary, with or without an insulation centreline summary, should be output.

CONTINUATIONSYMBOLS	Plots continuation symbols where pipes are connected to nozzles on ship equipment.
CONTNOTES	Specifies continuation notes at the end points on drawings for when the connection goes to another isometric.
COORDINATES	Specifies the drawing locations at which spatial coordinates are shown.
DIMDIST	Specifies the 'standout' distance, between dimensioning lines and pipe/ component centrelines.
DIMENSIONS	Specifies dimensioning style.
ELEVATION	Specifies whether or not elevation coordinates are to be shown.
FALLINGLINES	Defines the maximum rate of fall allowed in a pipeline before it is shown as a skew.
FALLSKEWS	Controls how dimensioning boxes for skewed/falling pipelines are shown.
FLOWARROWS	Allows flow arrows to be shown on isometric plots.
INSULATIONCONTROL	Specifies whether or not insulation is to be plotted.
LEGLength	Allows you to specify the plotted leg length of elbows, tees and crosses.
MESSAGEACCURACY	Controls how arrowed messages on the isometric are positioned.
OUTCOM	Defines the style in which attributes are shown on the isometric drawing area.
OVERALLSKEWS	Allows you to specify whether triangular skewboxes are shown as overall skewboxes across branches, or as separate skewboxes between branch connections.
PICTURESCALE	Allows you to set the scale of the isometric drawing.
REFDIMENSIONS	Switches the display of reference dimensions on and off.
REPEATABILITY	Creates split ATTAs at automatically generated split points.
SKEWBOX	Controls how skewed piping is shown.
SPECBREAK	Specifies the level at which a change of pipeline specification is noted, and which changes are shown.
SPLIT	Controls the number of drawing sheets used when a pipeline is split over more than one sheet.

SUPPORTS	Defines whether support positions are shown and are dimensioned as overall or string dimensions.
TAGGING	Allows you to identify individual components on the isometric plot by means of name tags.
TOLERANCE	Specifies the maximum misalignment between components regarded as having the same axis.
TRACINGCONTROL	Specifies whether or not tracing is to be plotted.
TRUNCATE	Stops rounding remainders of dimensions being added to the next dimension.
TUBESPLITS	Controls how splits in tube are placed.
UNITS	Specifies units of measurement to be used for ISODRAFT's output.
WELDID	Allows you to identify individual welds on the isometric by name.
WELDNUMBERS	Controls numbering the welds on an isometric.
WELDTYPE	Controls which types of end connection are shown on drawings.
ZEROLENGTHSKEWS	Allows you to connect a vertical Branch to a falling line.

10 Drawing Change Highlighting

10.1 Highlighting

The **Change Highlighting** function defines if and how modified data is highlighted on the isometric since a given comparison date. The comparison date is described in [Comparison Date](#).

The Change Highlighting function can be turned ON or OFF.

A Change Highlight colour can be selected and will be used for highlighting all changes.

If highlighting is switched ON, entries in tables and material lists, annotations, labels and dimensions will always be drawn in the highlight colour if they have changed since the comparison date.

Change Highlight Rules can be defined to specify which components are to be represented in the selected change highlight colour.

- If no rules have been defined, then no components will be shown in the highlight colour.
- If ALL components have been selected, but no conditions defined, then ALL components will be in the highlight colour regardless of whether anything has changed.

Using ELBO as an example:

- If ELBO components have been selected, but no conditions defined, then all ELBOs will be in the highlight colour regardless of whether anything has changed.
- If ELBO components have been selected with the MODIFIED() condition, then all ELBOs will be in the highlight colour only if the ELBO has changed.
- If ELBO components have been selected with the MODIFIED(POS) condition, then all ELBOs will be in the highlight colour only if the Position of the ELBO has changed.

Note: Any items deleted after the comparison date cannot be identified on the isometric. (They can be queried.)

You must ensure that the options used on each run are the same. Differing options cannot be controlled by the program.

When labelling grouped components, such as 'F8 L9 G10 B11', in boxed text it will not be possible to isolate the changed item.

You must look at the material list to see which of the grouped components has changed. This is particularly important for bolts and gaskets as these components are never represented on the isometric other than by label.

Change Highlighting options are set using the **Change Highlighting** form from the **Options Create/Modify** functions.

- Checking the **Show Change Highlighting** checkbox sets the Change Highlighting function to ON, unchecking it sets the function to OFF.

- Clicking on the **Colour** button brings up the **Highlight Colour** form. Use this to select the desired highlight colour.
- Use the **Create**, **Delete** and **Clear** buttons to:
 - Create a new Highlight Rule.
 - Delete a selected Highlight Rule.
 - Delete all Highlight Rules.
- Use the **For** drop-down list to select a component type that you want to be associated with the selected Highlight Rule. Use the **Add** button to make the association. Any number of components can be added as required.
- If you want to remove a component type from the selected Highlight Rule, select the component type using the **For** drop-down list, then click the **Remove** button.
- Use the **With** text box to add a qualifier to the component selected in the **For** drop-down list to make it more specific, for example: MODIFIED (POS) or CREATED().
- Click on the **Apply** button to save the current form settings to the **Option** file.
- Click on the **Dismiss** button to close the form without saving any modifications made.

For syntax related to change highlighting refer to the [ISODRAFT Reference Manual](#).

10.2 Comparison Date

It is only by comparing a drawing at two states or sessions that it is possible to determine what has changed. Using the current state of the drawing as one state, you must then reference an earlier state in order to make the comparison. You do this by specifying a Comparison Date: (COMPDATE), that is, the drawing state at a time that you wish to use as a baseline or datum.

If no Comparison Date has been defined, then an error is raised and isometric production is aborted.

The Comparison Date is set from the **Set Comparison Date** form.

The Comparison Date can be set in one of two ways:

- By specifying an actual time and date.
- By referencing a Stamp.

A Stamp is a way of referencing combinations of databases and sessions at specified instances. Stamps are created in ADMIN. For more information on Stamps and how they are created and used refer to the [Administrator Command Reference Manual](#) and the [Administrator User Guide](#).

10.3 Summary of Commands

The following commands were described in this chapter:

SETCOMPDATE	Specifies the Comparison Date to be used to determine what has changed on an isometric.
CHANGE HIGHLIGHTING	Switch ON/OFF, select a colour, define Highlighting Rules for components.

11 Symbol Keys

Each type of component has a symbol which is used to represent it on the isometric drawings. A set of default symbols for a wide range of components is supplied with ISODRAFT. The symbols supplied in the standard library are shown in the chapter *Symbol Keys*, of the *ISODRAFT Reference Manual*. You can query the SKEYs in the symbol library index, but you cannot query symbol attributes and shape definitions.

The symbols are linked to the components they represent in the Catalogue database via the SKEY (symbol key) attribute of a DTEXT element.

You can create new symbols in OUTFITTING DRAFT or other 2D drawing packages. For the forms and menus used to create the symbols in OUTFITTING DRAFT, see the *Administrator Application User Guide*. The commands to create symbols are described in the *Draft User Guide*.

This chapter explains the way ISODRAFT handles the symbols.

11.1 Basic Principles

Any new symbol must be derived from an *existing* symbol which has the required characteristics; however, the existing symbol need not be for a component with the same generic type (GTYPE) as the new one.

Items such as Bends, Elbows, Tees and Crosses, which are drawn as pieces of tube rather than as symbols, cannot be redefined. Other specific components which cannot be redefined are indicated by a 'No' entry in the USER DEFINABLE? column of the SKEY Reference Section in the chapter *Symbol Keys* of the *ISODRAFT Reference Manual*.

You should not redefine the symbols for Instrument Dials (with SKEYs IDFL and IDPL) because their circular shapes are used to store instrument references and are therefore not converted from orthogonal to isometric symbol shapes.

The maximum number of user-defined symbols which may exist at any time is 300.

Many standard symbols consist of a number of separate smaller symbols; for example, the symbol for a valve incorporates a body symbol, a spindle symbol and end symbols. In this case the body and spindle symbols may be separately redefined in OUTFITTING DRAFT, while the end symbols may be modified in PARAGON, as explained in *Cataloguing Requirements*.

Note: Take care that symbols are not so large that they overlap other information, such as dimensioning data, on the drawings. It may be necessary to adjust other aspects of the isometric, such as the standout distance of dimensions, in order to maintain legibility.

11.2 Symbol Attributes

The attributes of ISODRAFT Symbol Templates are set in OUTFITTING DRAFT, where the symbol templates are defined. They are as follows:

Scale *val*

Changes the size of the symbol without re-defining the coordinates of the plotted shape (100 = full size).

ORientation NONE

Specifies a symmetrical fitting which the flow direction is not relevant.

ORientation FLOW

Specifies a fitting for which the flow direction is important, such as check valves.

ORientation REDUcer

Specifies a reducing fitting.

ORientation FLANge

Specifies a fitting which begins with a flange; that is, one which is preceded by a gasket.

FLWA TRUE

Plots flow arrows on the symbol.

FLWA FALSE

Suppresses flow arrows.

DIMension ON

Plots dimensions alongside the symbol.

DIMension OFF

Suppresses the plotting of dimensions.

Fill

Enables default and user-defined symbols to be drawn as filled polygons.

User-defined symbols based on the following SKEYs and defined with ORI FLAN or ORI REDU will be automatically flipped to fit in with the pipeline component sequence.

GTYPE	SKEY	Description
CAP	All	All caps
FLAN	All	All flanges
LJSE	All	All lap joints stub ends
PCOM	PL	Plug
REDU	All	All reducers

User-defined symbols based on the following SKEYs and defined with ORI FLOW will be automatically flipped depending on the flow direction.

GTYPE	SKEY	Description
INST	DR	Rupture disk
INST	XV**	Pressure reducing valve
PCOM	TU**	Tundish (funnel)
VALV	CK**	Check valve
VALV	VX**	Pressure reducing valve
VENT	RD	Rupture disk

11.3 Spindle SKEYS

Spindle SKEYs may be added to any symbol which has a spindle position.

SPINdle ON	retains the original spindle shape.
SPINdle OFF	suppresses the output of spindle shapes and related orientation notes.
SPINdle <i>skey</i>	replaces the original spindle shape by that specified by <i>skey</i> .

Figure 11:1.: The Standard Spindle Shapes and their SKEYs illustrates the 12 standard spindle shapes available, with their corresponding SKEYs.

If the original symbol does not have a spindle, you can set the PURP (Purpose) attribute of any PTAX (axial p-point) to SPIN to show that a spindle is required in the corresponding direction for the current element. If the spindle direction is not orthogonal, a comment giving the actual direction will be output on the isometric plot.

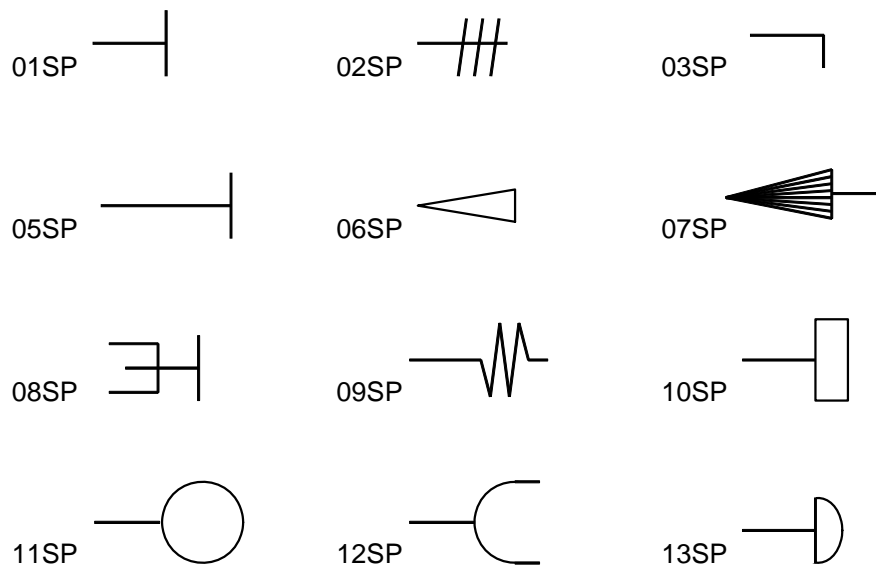


Figure 11.1. The Standard Spindle Shapes and their SKEYs

11.4 Defining a New Hanger

Hanger definition is a special case. There is only one existing hanger SKEY, namely 01HG, which should form the basis for all newly defined hanger shapes.

For hangers which lie along the pipeline and which do not need to be reoriented, the hanger shape can be defined as for a normal fitting. For example, the existing hanger 01HG might be defined as shown in [Figure 11.2.: Defining a hanger](#).

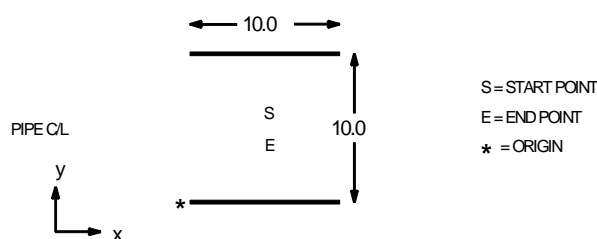


Figure 11.2. Defining a hanger

For hangers that require an orientation (i.e. those that must be drawn above/below, east/west or north/south of the pipeline), the hanger must be defined in two stages. This technique represents a special case of the SPINDLE command described in [Spindle SKEYS](#).

First, the spindle SKEY (99SP) must be defined as the required shape in the normal way, but noting that the spindle must be defined *on its side*; that is, with the axis of the spindle stem in the x direction and the plane of the handwheel through the y direction. See [Figure 11.3.: Defining a spindle key](#).

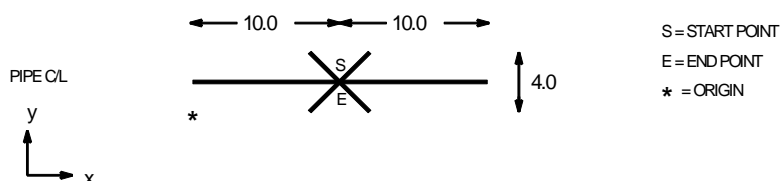


Figure 11:3. Defining a spindle key

Secondly, the hanger SKEY (99HG in this example) must be defined as just a **start point**, a **spindle point**, with an associated spindle SKEY (99SP in this example), and a **finish point**, all of which must be coincident: see [Figure 11:4.: Defining a hanger with a spindle key](#).

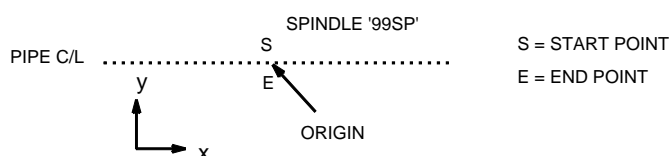


Figure 11:4. Defining a hanger with a spindle key

Note: That the P3 of the ATTA gives the spindle direction of the hanger, and so it must be perpendicular to the line.

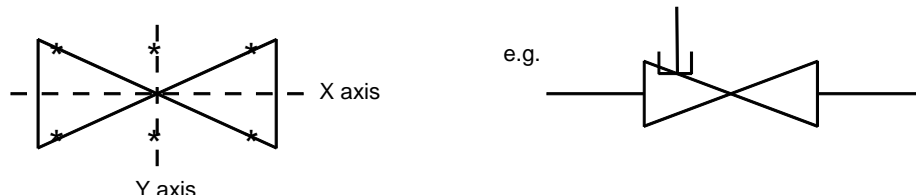
The orientation of the hanger 99HG is retrieved from the database.

11.5 Pipe Components with More Than Four Connections

PCOMs and SHUs may be shown with up to six off-line legs. Flanges and filters can be shown with a single off-line leg. Off-line legs may be regarded as tapping points positioned on the piping component; for example, instrument connections on orifice flanges, vents in valve bodies, etc.

SKEYs FOWN (Weld-neck orifice flange), FOSO (Slip-on orifice flange) and FI** (Inline filter) have automatic tee points. Any other symbols requiring off-line legs must be user-defined.

The links between the 2D symbol representation and the 3D component model are provided by Tee points. A maximum of six tee points may be defined for each symbol. They can be defined in any order, but must be distributed on the fitting as shown:



The following rules for tee point layout must be obeyed:

- No tee points may be positioned on the X axis
- No more than 3 tee points may be positioned above the X axis
- No more than 3 tee points may be positioned below the X axis
- No more than 2 tee points may be positioned on the Y axis
- No more than 2 tee points may be positioned to the left of the Y axis
- No more than 2 tee points may be positioned to the right of the Y axis

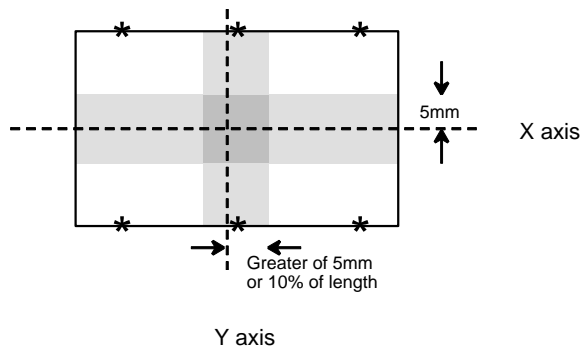
A symbol which has no tee points defined, but which is modelled with fitting taps, will have them drawn from P0.

A symbol which is symmetrical should have tee points defined symmetrically about the symbol axis. This allows any tee leg orientation to find an available tee connection point when detailing takes place. For example, for a circular flange you should allow at least three tee points, even though only one may be needed. ISODRAFT will decide from the tapping point and the symbol definition which tee point should be used.

The default tee point is at the P0 of the symbol. The tee points defined on the X and Y axes will be used if the coordinates given fall on, or within a specified distance of, these axes. Other tee points will be used if the coordinates fall outside the following tolerances:

- The outer tee points in the direction of the Y axis will be used if the coordinates are further along the Y axis than 5mm
- The outer tee points in the direction of the X axis will be used if the coordinates are further along the X axis than 5mm or 10% of the true length (whichever is the greater)

Thus, in the following diagram, coordinates in the shaded areas will select the tee points in the shaded areas and coordinates in the unshaded areas will select tee points in the unshaded area:



In the case of an offset fitting, the centreline is taken to be the line joining the start point to the end point of the symbol. To simplify the generation of tee points on both the X and Y axes, the value of -1 may be used for the XPOS and YPOS in the symbol definition for tee point coordinates. (Note that PCOM and SHU symbols cannot be 'flipped'.)

ISODRAFT incorporates facilities for setting the scale of the tapping leg and for switching tapping leg dimensions and/or coordinates on and off. See the TAPPING command.

11.6 Offset Fittings

The following types of offset fitting may be defined:

Component Type	SKEY
Offset Filter	FO**
Offset Instrument	IO**
Offset Block	BO**
Offset Trap	TO**

In addition, all flanges (except LJSE) may be defined with offsets.

When defining an offset fitting, the start point for the defined shape will be the low point of the offset and the end definition point will be the high point of the offset, thus:



The ORIENTATION NONE option (see Section 11.2) should be used when defining offset fittings.

Normally, when the end condition FL is specified, flanges are generated and attached to the start and end points of the symbol automatically. When offset symbols are user defined, however, flanges are *not* added automatically; they must be built into the symbol or added by using the PSKEY attribute of the relevant p-points (see [Component End Conditions Defined By P-Points](#)). This principle applies also to the following non-offset fittings with flanged connections:

Component Type SKEY

Flame Trap	FTFL
Tundish (or Funnel)	TUFL
Flexible Hose	FXFL

11.7 Angled Fittings

It is possible to define two groups of types of angled fittings.

The first group of types of angled fitting may be defined as follows:

Component Type	SKEY
Angled Filter	FA**
Angled Instrument	IA**
Angled Trap	TA**

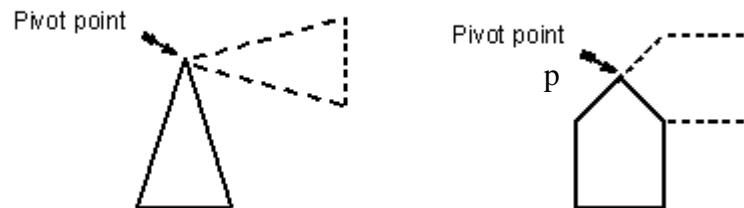
Component Type	SKEY
Angled Valve	AV**
Angled Valve	AX**
Angled Valve	AR**
Angled Instrument Valve	CA**
Angled Instrument Valve	SA**
Angled Instrument Valve	MA**
Angled Instrument Valve	HA**

Definition of these components is subject to the following constraints:

- Both legs (In and Out) must be identical.
- The legs must have a common pivot point and this pivot point must be at a closed end. (This excludes shapes similar to those currently used for the FA** range of SKEYs.)
- The symbol shapes must be symmetrical about the X-axis.

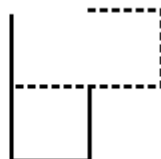
For example:

Valid shapes:



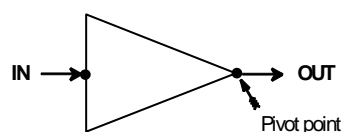
The dotted legs are those which will be duplicated and mirrored automatically.

Invalid shapes:



Unless the legs are symmetrical, it cannot be guaranteed that they will be duplicated in the correct plane when they are mirrored.

When defining an angled fitting, the start point for the defined shape will be the pipe connection point and the end definition point will be the closed (or pivot) point, thus:



A tee point must be positioned on the pivot point.

The ORIENTATION NONE option should be used when defining angled fittings.

The second group of angled fittings is the Y-types, so called because of the angle created between the main leg and the off legs.

Component Type

SKEY

Generic Y-type Cross with user-definable out- and off- legs

X@**

Generic Y-type Tee with user-definable out- and off- legs

Y@**

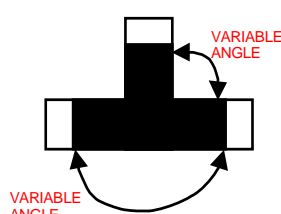
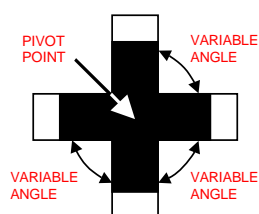
The Y-type Cross and Tee need to be set up in the catalogue before these symbols can be used. Use Paragon to define each symbol - requiring PTSEs (point sets), GMSEs (geometry sets), DTSEs (data sets) and SCOMs of generic type CROS and TEE. It is advised to use parameters and design parameters when setting up the data. Parameters are set on the SCOM and the design parameters are set on the component during design. For the PTSE it is advised to use PTAX or PTPOS for the in leg p-point and PTPOS for each out leg p-point. PTPOS must be used for the out legs as the PTAX element does not support the usage of design parameters.

By default the legs of the Y-type Cross and Tee are defined as variable width, rectangular, filled polygons overlapping at P0. In this way the pivot point will not be apparent.

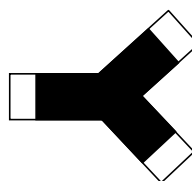
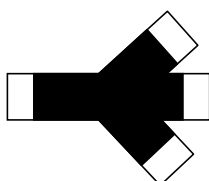
Each leg corresponds to a symbol definition for X1**, X2**, X3**, X4** for the Y-type Cross and Y1**, Y2**, Y3** for the Y-type Tee. The correct symbol is automatically selected by ISODRAFT dependent upon the comparative bore sizes of the connections. The @ is replaced by a digit in the range 1-4 for the Cross and 1-3 for the Tee, with X1** / Y1** being allocated to the largest bore, and X4** / Y3** being allocated to the smallest bore. Each of the legs is redefinable so it is important to bear in mind the bore sizes when redefining the symbol of any leg. The end connections replacing the ** may be: FL (flanged), CP (compression), SC (screwed), BW (butt welded), SW (socket welded) or PL (plain).

Y-TYPE CROSS

Y-TYPE TEE



When the angles have been applied then the symbols appear as shown below.



Flanged connections on generic Y-type fittings are represented on the isometric on the tube only. This is due to the fact that the y-type fitting is user-definable and flanged representation on the component cannot be enforced.

11.8 Universal SKEYs

This facility provides an SKEY suitable for any piping component of generic type PCOM. Fittings with a universal SKEY will be drawn as a 'ghost' symbol, with no attempt made to represent the exact geometry or dimensions on the isometric plot. You will need to attach a OUTFITTING DRAFT drawing, detail plot, or some other method to give the fitting detail. You can use labelling to identify the fitting connection points.

This feature is particularly useful if your designs use non-standard fittings (such as in the food or pharmaceutical industries), since it makes it possible to automatically produce isometrics of piping networks containing *any* piping component. The only limitation will be the clarity of components with many connected branches.

The universal SKEY symbol is represented by a circle with lines from its circumference to each connection point. The radius of this circle (default 10mm) is controlled by a percentage scaling factor, SKeyScale, on SDTE elements. For example, SKeyScale = 110% gives a circle radius of 11mm.

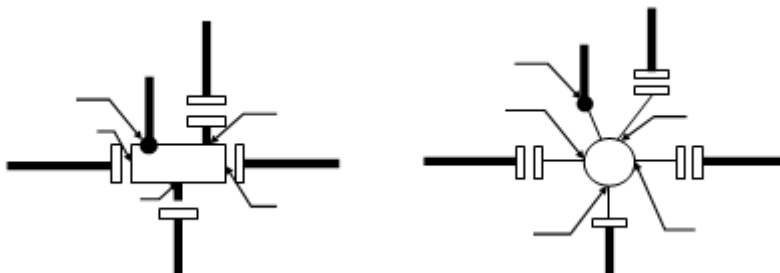
To use this facility, set the SKEY attribute to UNIV and, optionally, set the SKeyScale attribute. To output a detail plot, set the DPFN attribute on the SPCO or piping component in the usual way. Alternatively, set the DPFN (Detail Plot File Name) attribute on the SDTE element, in addition to its SKEY and SKeyScale attributes.

11.8.1 End Connections

The universal (UNIV) SKEY will imply no end connection information. To flag end connections, the PSKEY on each p-point must be set to FL (flanged), CP (compression), SC (screwed), BW (butt welded), SW (socket welded) or PL (plain). If unset, PL will be assumed. For CP, SC, BW or SW, the functionality is the same as for PCOMs with tapped branches; that is, the appropriate end connection symbol will be drawn at the start of the offleg branch.

For explicit SKEYs, tapped branches with PSKEY set to PL or FL are drawn with a short length of tube. For universal SKEYs, the FLSO symbol will be drawn at p-arrive and p-leave if those p-points have PSKEY set to FL, thus:

PCOM with tapped branches Universal SKEY



11.8.2 Tapping Syntax

The TAPPING command, used to control some aspects of branch output for PCOMs, is not applicable to branches associated with universal SKEYs.

11.8.3 Connection Labelling

When fittings are very complicated, the offlegs on the isometric plot cannot always be related easily to the true geometry of the detailed fitting. To aid the fabricator, the relevant p-point number is output at each connection point on the isometric. This number is prefixed with ATEXT 388 (default setting 'P') and is drawn inside an ellipse to distinguish it from other messages. When you use a universal SKEY, you are responsible for ensuring that the detail plot has connection p-points marked in this way. If ATEXT 388 is set to blank, no messages will be output

12 ISODRAFT Transfer File

Before ISODRAFT produces an isometric plot, it processes data from the OUTFITTING DESIGN and Catalogue databases and creates an **intermediate data file (IDF)**. This transfer file is produced as a binary file, which is processed by ISODRAW to create the plot file: you can, however, specify that ISODRAFT will produce an ASCII copy by using the command

```
TRANsferfile filename
```

where *filename* is the name of a file in the operating system directory.

Production of the ASCII transfer file is cancelled by the command

```
TRANsferfilename OFF
```

An ASCII transfer file produced in this way can be processed by the

```
PROCESS TRANSFERFILE filename
```

command to produce a plot file.

The transfer file can provide a useful means for sending information to AVEVA Solutions Ltd for assistance in resolving problems associated with Product User Reports (PURs).

Note: The default transfer file, /trans, will be overwritten whenever a DETAIL or CHECK command is given. Specify a new file name if you want to keep the data.

Note: That transfer files produced by versions of ISODRAFT before Version 10.3 are in binary format only, and cannot be processed by the PROCESS TRANSFERFILE command.

12.1 Preprocessing and Postprocessing Commands

You can execute system commands (for customised reporting, etc.) during the transfer file processing sequences:

- **Preprocessing commands** are system commands which are executed *after* the IDF has been written and *before* ISODRAW has been invoked.
- **Postprocessing commands** are system commands which are executed *after* ISODRAW has finished and *before* control is returned to ISODRAFT.

The PREPROCESS and POSTPROCESS commands each let you define up to five numbered command strings for this purpose. The syntax is:

```
PREPROCESS integer1 'command string1' integer2 'command string2' ...  
POSTPROCESS integer1 'command string1' integer2 'command string2' ...
```

12.2 Summary of Commands

The following commands were described in this chapter:

PROCESS TRANSFERFILE	Instructs ISODRAFT to process a transfer file.
PREPROCESS	Defines up to five preprocessing system commands.
POSTPROCESS	Defines up to five postprocessing system commands.

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