



ALIAS SHALL NOT BE LIABLE FOR TECHNICAL OR EDITORIAL ERRORS OR OMISSIONS CONTAINED HEREIN NOR FOR INCIDENTAL OR CONSEQUENTIAL DAMAGES RESULTING FROM THE FURNISHING, PERFORMANCE OR USE OF THIS MATERIAL.

It is the policy of Alias Limited to enforce any and all rights relating to protection of this information. The information contained in this document is proprietary in nature and may not be reproduced, copied or divulged in whole or in part without the prior written consent of Alias Limited. This document has been provided pursuant to the terms of a contract or confidentiality agreement. Unauthorised distribution or disclosure of its contents is a violation of that contract or agreement and persons doing so may be liable for penalties as provided by law.

COPYRIGHT

Copyright © 1991-2001 Alias Limited Industrial Software Products.

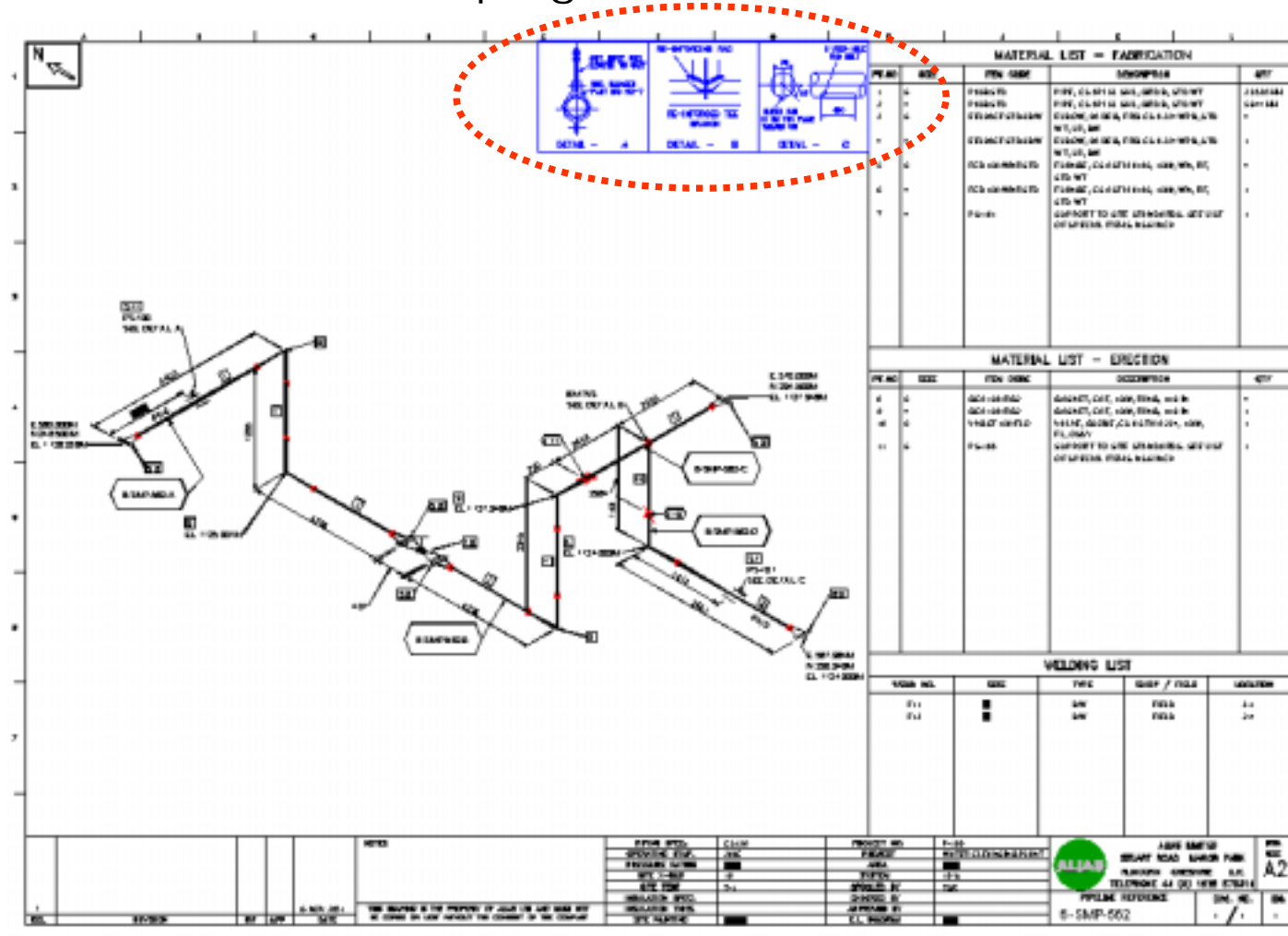
Stuart Road,
Manor Park,
Runcorn,
Cheshire,
WA7 1TS,
United Kingdom.
Telephone: +44 (0) 1928 579311
Fax: +44 (0) 1928 579389
E-mail: info@alias.ltd.uk
Web: www.alias.ltd.uk

TRADEMARKS

I-Sketch, I-Sketch Classic, I-Sketch Field, I-Convert, I-Export, I-Run, PLANTGEN, SPOOLGEN and ISOGEN are registered trademarks of Alias Limited.

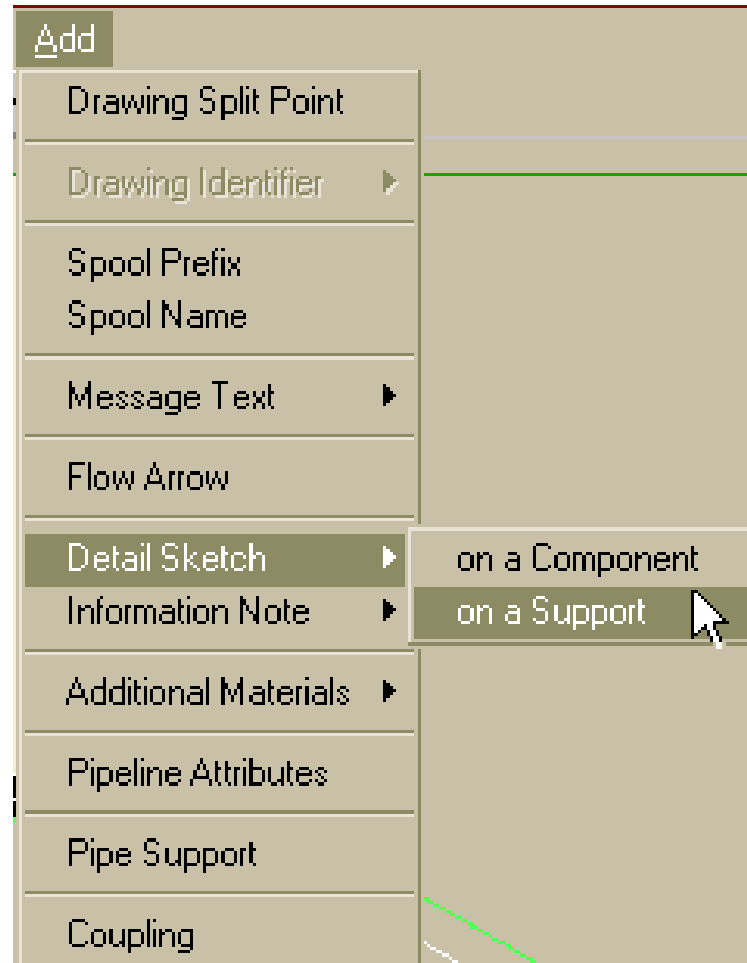
All other trademarks are the property of their respective owners.

This feature Permits User's Detail Sketches to be generated on the Piping Isometric like this



Basic Features

- The Detail Sketches are mini drawings prepared by the User in a graphics system like AutoCAD or MicroStation
- The permitted File Format is DXF, DWG or DGN (FOR DWG SEE SLIDE 27)
- For AutoCAD each Detail Sketch is held in an individual DXF file whereas for MicroStation they're held collectively in a Cell Library
- The Detail Sketches are assigned to a Drawing Layer / Drawing Level
- In Spoolgen Probing - Detail Sketch filenames are entered in the pipeline at the point needing the information - An Automatic Cross Reference is then generated between that point & the Detail Sketch graphics
- They are automatically positioned on the isometric by Spoolgen

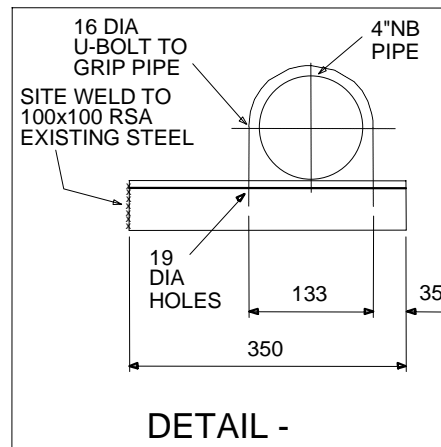
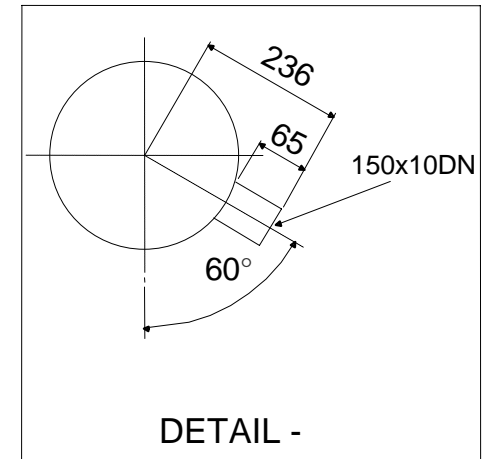
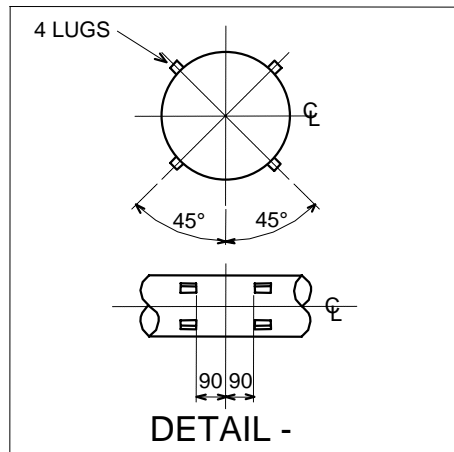


This is
where the
Sketches
are added
in Spoolgen
Probing

Typical Uses :-

- Pipe Supports / Hangers
- Branch Connections
- Support Lugs
- Special Welds
- etc.

Some Sample Sketches





File Format Restrictions

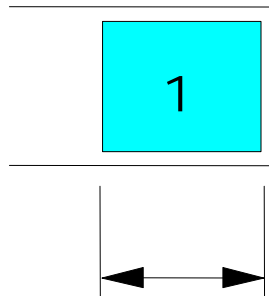
- On a given isometric - all Detail Sketches must be of a consistent format
- If Detail Sketch files are DXF then the Spoolgen plot file must be DXF
- DXF sketches must be saved in AutoCAD Release 12 format (to 6 decimal places) and must not contain Blocks
- See slide 27 for DWG information

Defining the DXF Units (Not applicable to MicroStation users)

This is done to inform Spoolgen whether mm or Inch dimensions have been used in the construction of the Detail Sketches

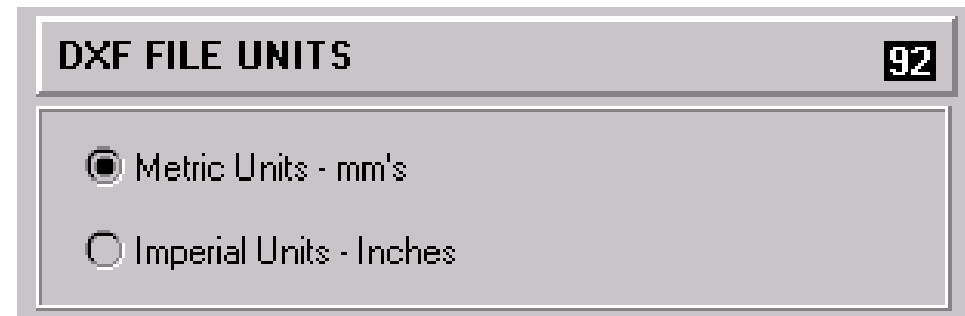
It's done directly in O.S. 92 of an .OPT file or by using the Options Editor

1 O.S.92



Value 0 = Metric Units (mm's)
Value 1 = Imperial (inch)

2 Using the Options Editor



On the Plotted Isometric page -
select one of these options

The value set here is also used for the
Spoolgen DXF plot file output units and the
DXF underlay file units

The Data Definition File (DDF) - Is used to control the following aspects of Detail Sketches -

- File Format
- Sketch Size

And, for the Cross Reference Identifying Character :-

- The Position on the Sketch
- Text Height
- Text Weight (MicroStation only)
- Drawing Layer / Level

The Drawing Definition File (DDF) Contd.

Detail Sketch Section of DDF input file - sample data :-

DETAIL-SKETCH

FILE-FORMAT DXF

SKETCH-SIZE 50 50

CROSS-REFERENCE 35 4 ALPHA

TEXT-HEIGHT 2.8

TEXT-THICKNESS 2

DRAWING-LAYER 40



Metric
data

DETAIL-SKETCH

FILE-FORMAT DXF

SKETCH-SIZE 2 2

CROSS-REFERENCE 1.5 0.065 ALPHA

TEXT-HEIGHT .11

TEXT-THICKNESS 2

DRAWING-LAYER 40



Imperial
data



The Drawing Definition File (DDF) Contd.

File Format Parameters

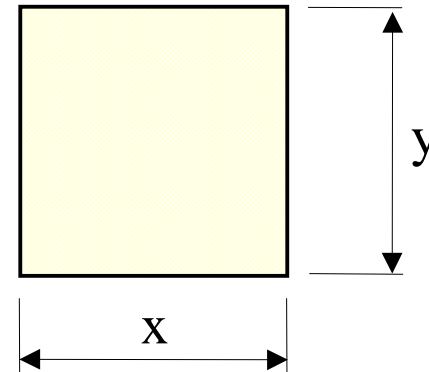
FILE-FORMAT *data*

where *data* = DXF or DGN

The Drawing Definition File (DDF) Contd.

File Format Parameters

SKETCH-SI ZE x y



Where x (horizontal) and y (vertical) are the size of the sketch border sizes in mm's or Inches

Values may be integer (e.g. 50) or decimal (e.g. 50.5) for Metric or 2 or 1.97 for Inches

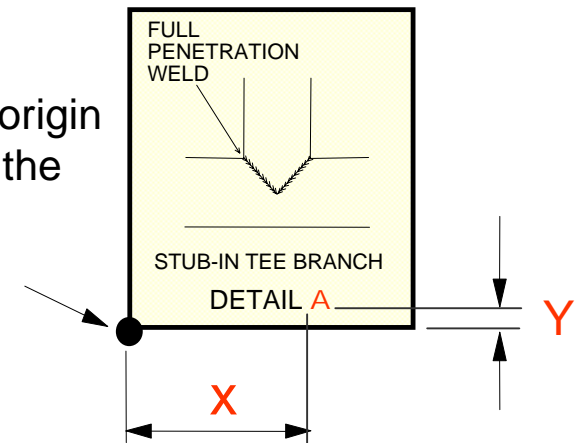
The Drawing Definition File (DDF)

.... Contd.

File Format Parameters

CROSS-REFERENCE *x y type*

The Sketch origin
is always at the
bottom
LH corner



Where **X** (horizontal) and **Y** (vertical) are the distances in mm's from the Sketch Origin Point to the bottom LH corner of the identification text (letter **A** in the above example).

Values may be integer (e.g. 35) or decimal (e.g. 35.5) for Metric or 1.5 or 2 for Inches

type may be either ALPHA or NUMBER

ALPHA generates letters - A B C etc.

NUMBER generates numbers - 1 2 3 etc



The Drawing Definition File (DDF) Contd.

File Format Parameters

TEXT-THICKNESS *value*

Where *value* is the text thickness of the Cross-Reference character expressed as an integer number - e.g. 2

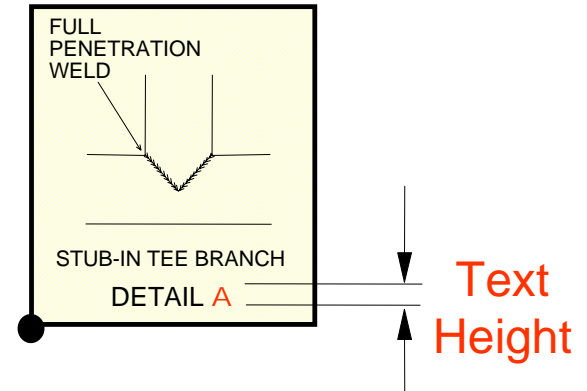
Text Thickness is a valid parameter for MicroStation where it's called Text WEIGHT - but it's not used in AutoCAD

The Drawing Definition File (DDF)

....Contd

File Format Parameters

TEXT-HEIGHT *value*



Where value is the Text Height of the Cross-Reference character

Values may be integer (e.g. 3) or decimal (e.g. 2.8) for Metric or .11 for Inches

If Text Height is not specified here then the value contained in O.S. 4 will be used

The Drawing Definition File (DDF) Contd.

File Format Parameters

DRAWI NG-LAYER *value*

This is the Drawing Layer for the Cross Reference character only

- where value is an integer number valid for the target graphics system - e.g. 40

(The Drawing Layer for the Sketch itself is set in AutoCAD in the normal way)

In the absence of **DRAWI NG-LAYER** information then Layer 1 is used by default

In MicroStation - Level is used not Layer - but the command used is still

DRAWI NG-LAYER

Detail Sketch File Directories

Detail Sketch Files may be held in a variety of ways -

AutoCAD

For AutoCAD each Detail Sketch is held in an individual DXF file

These files are held in a Sketch Library / Directory

The Sketch Library / Directory may be kept -

- Within an individual Project
- In a Top Level Directory on a PC for common access by all Projects on that PC
- On a Server for access by all Users on the network

The required method is selected when creating a Project using NEWPROJ

MicroStation

The Detail Sketches are held collectively in a Cell Library

Location Of Sketches On The Isometric

Detail Sketches may be positioned on the isometric in 3 different ways -

1. Across the top of the drawing
2. Positioned locally around the isometric in free spaces
3. Output to a completely separate isometric containing only Detail Sketches

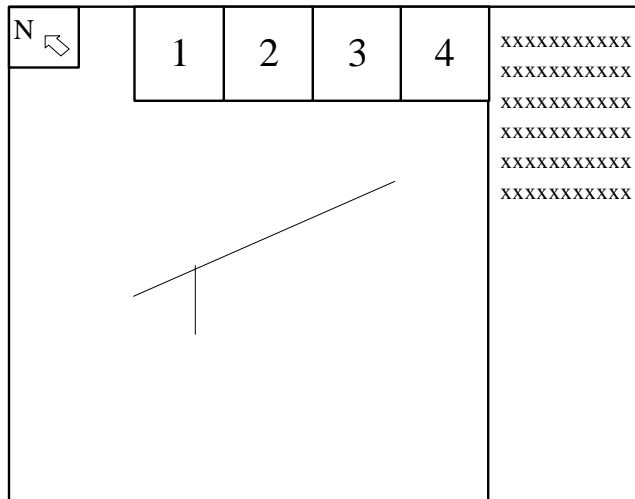
The required method is User determined and is specified in the DDF

Location Of Sketches On The Isometric

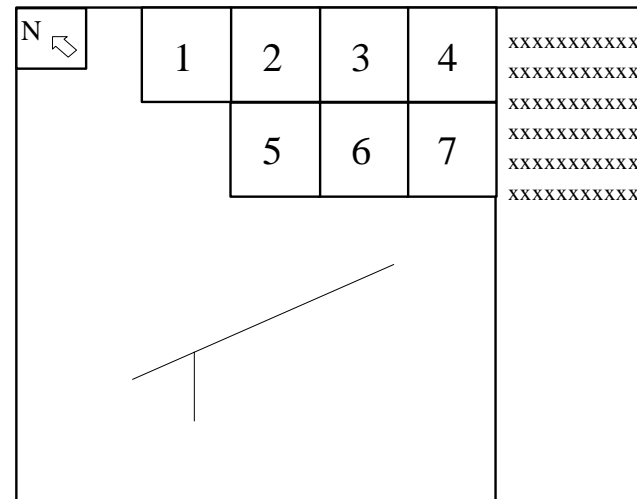
Method 1 - Across the top of the drawing

The actual Sketches location is determined automatically by Spoolgen and will be done away from the North arrow

Typical Layouts

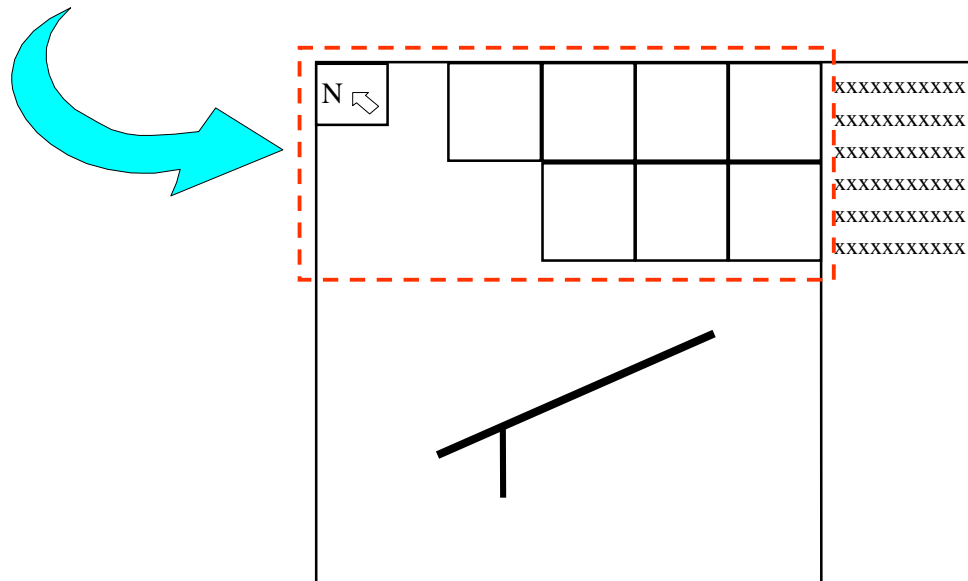


Single Row



Double Row

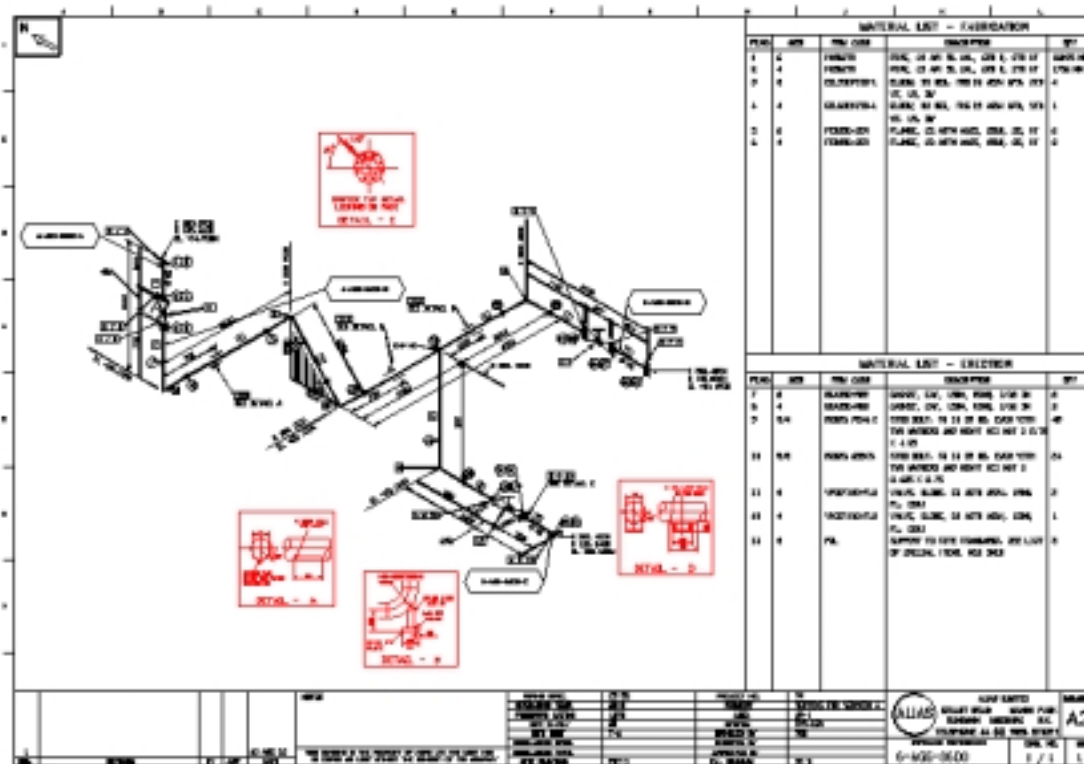
When Method 1 is used an invisible Reserved Area is created around the block - like this -



The Sketches Area automatically becomes a Reserved Area that
Spoolgen will not draw in
This Reserved Area extends across the whole width of the isometric
drawing section

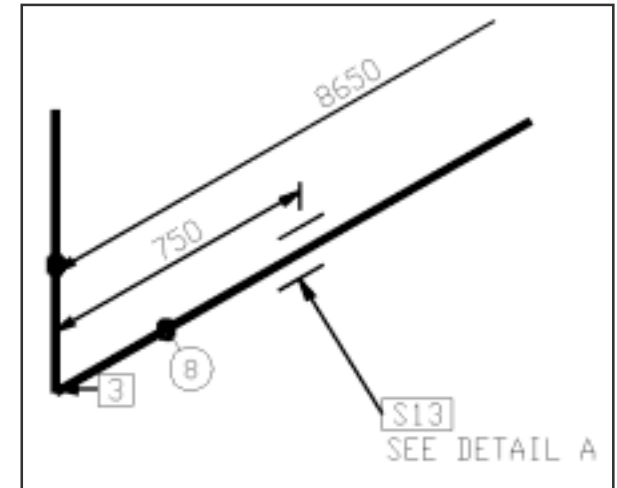
Location Of Sketches On The Isometric

Method 2 - Locally positioned around the isometric in free spaces



Locally Positioned Detail Sketches -

- Each Sketch is positioned in so called 'white space' - close to the relevant component
- Arrowed Lines between Sketches and Pipeline components will **not** be generated - but the cross reference message '**SEE DETAIL . . .**' will be output pointing to the component
- In the case of duplicate occurrences, only one copy of each unique Detail Sketch will be output on a single isometric - but the '**SEE DETAIL . . .**' cross reference message for that Sketch will be generated for each occurrence
- To conform to the basic Isogen requirement to generate clear isometrics - no Message Lines nor non-Sketch related text will be permitted to encroach into the confines of a Detail Sketch outline border



Locally Positioned Detail Sketches -

Are switched on by the addition of the **SKETCH-POSITION-LOCAL** command in the DETAIL-SKETCH section of the Data Definition File (DDF) - like this -

DETAIL-SKETCH

SKETCH-POSITION-LOCAL

FILE-FORMAT DXF

SKETCH-SIZE 50 50

CROSS-REFERENCE 35 1.85 ALPHA

TEXT-HEIGHT 2.8

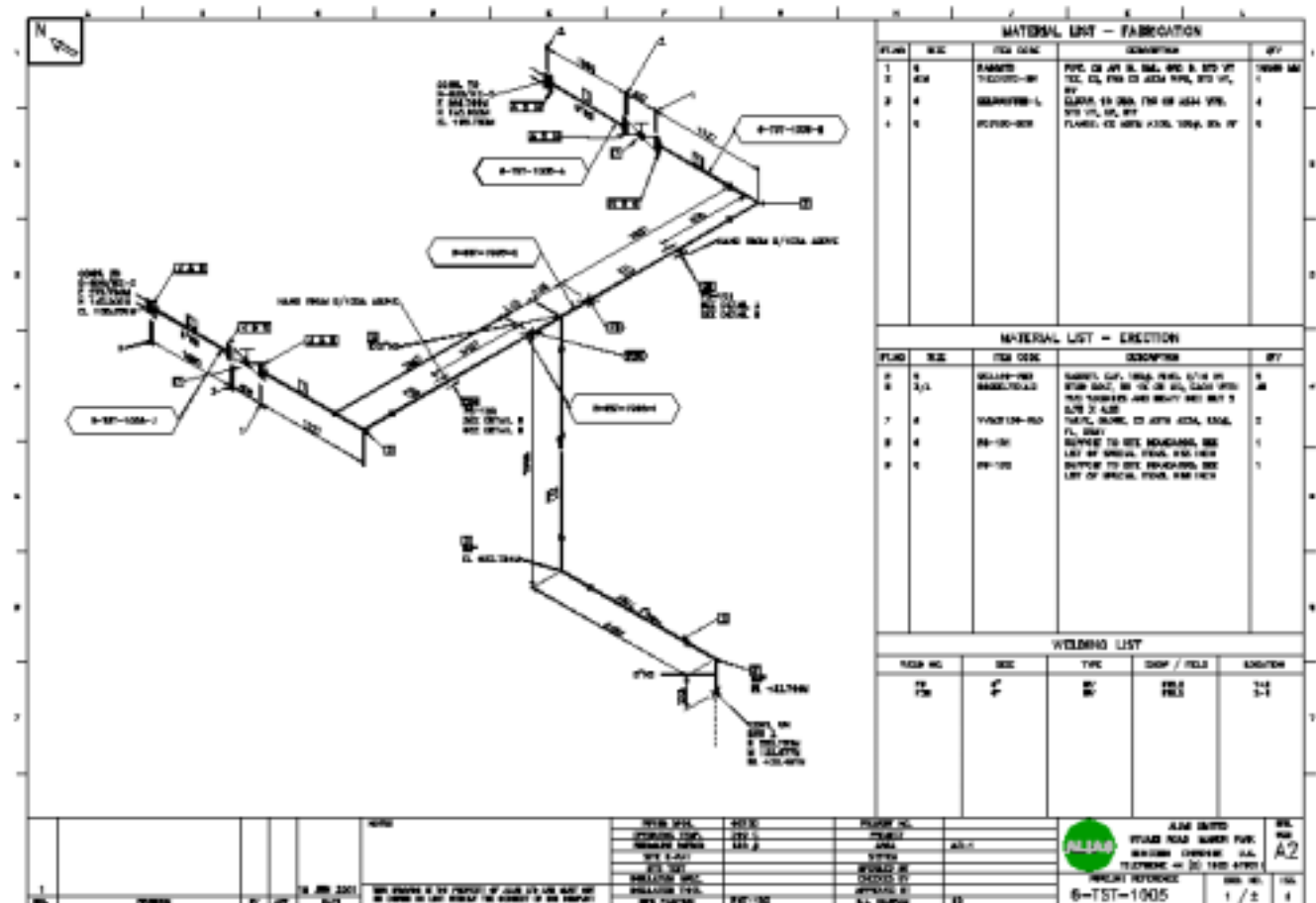
DRAWING-LAYER 40

Location Of Sketches On The Isometric

Method 3 - Output on a separate isometric containing only Detail Sketches

With this method the Detail Sketches are generated on a separate dummy drawing

This is Drg 1
which does
not contain
any Sketches



This is Drg 1A
- the dummy
overflow
drawing

It contains
just the Detail
Sketches

With
Positional
Method 3 the
Sketches are
always placed
in line across
the top of the
isometric

Detail Sketches

MATERIAL LIST - FABRICATION				
ITEM	SIZE	REQ. QTY.	DESCRIPTION	QTY.

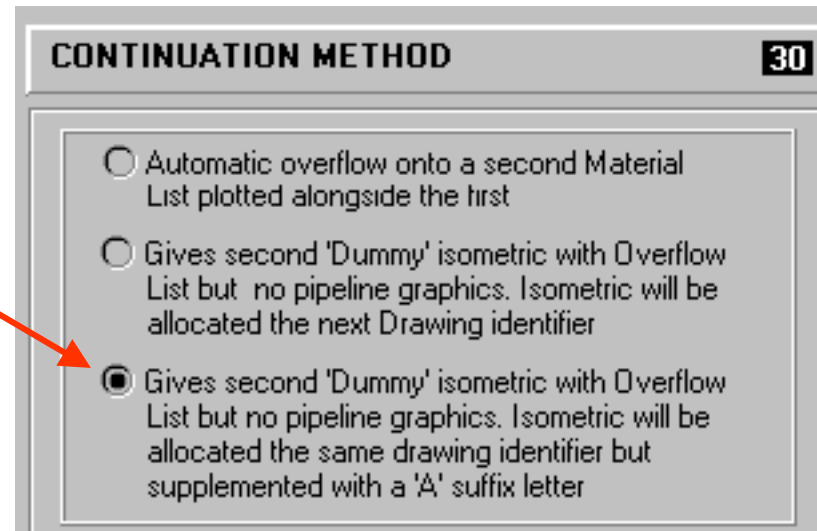
MATERIAL LIST - ERECTION				
ITEM	SIZE	REQ. QTY.	DESCRIPTION	QTY.

WELDING LIST				
WELD	WELD	WELD	WELD / WELD	LOCATION

PROJECT NO.		PROJECT NAME	
DESIGNER		LOCATION	
CHECKED BY		DATE	
APPROVED BY			

Requirements to set up Positional Method 3

1. In the Options Editor make this setting for O.S. 30



2. Make this entry in the DETAIL-SKETCH section of the DDF -
SKETCH-POSITION-OVERFLOW

AText

There is one AText for use with Sketches -

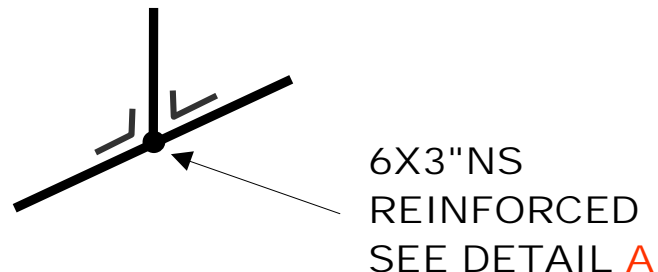
-456 SEE DETAIL ?

This is the default setting and the ? character is used for the insertion of the Cross Reference character - e.g.

SEE DETAIL **A** (Alpha system)

SEE DETAIL **1** (Numeric system)

This text is plotted at the pipeline Sketch location as shown below





DWG Detail Sketches and Information Notes

An enhancement made in Spoolgen 4.10.0 allowed the user to create a DWG output and in order for the system to place Detail Sketches onto the finished isometrics certain additional steps must now be followed.

Creating DWG sketches

1. The sketch is drawn in the usual way and saved as a DWG.
2. Ensure that the entry in the relevant DDF file shows the following

DETAIL-SKETCH

FILE-FORMAT DWG

SKETCH-SIZE 50 50

CROSS-REFERENCE 38 1.65 ALPHA

TEXT-HEIGHT 3

DRAWING-LAYER 40

INFORMATION-NOTE

FILE-FORMAT DWG

NOTE-SIZE 36 16

INTRODUCED IN 4.10.0



Using existing DXF sketches

1. Open the existing DXF sketch and save it as a DWG.

Note. This must be in the same AutoCAD format as the backing sheet into which it will be placed.

2. When entering the sketch reference replace the DXF format name with DWG. For instance under the old system a sketch would be entered as SK1.DXF because we are now using DWG sketches we simply enter SK1.DWG.

3. Ensure that the entry in the relevant DDF file shows DWG (see previous slide).