SIEMENS

COMOS

Process Isometrics

Operating Manual

09/2014
A5E32078384-AA

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Table of contents

Trademarks

1

Trademarks

Registered trademark: COMOS®

Introduction 2

Aim

COMOS Isometrics is a tool for creating isometric pipe drawings that are required for production and documentation of industrial pipes.

Pipe spec concept

If you place a pipe run on an isometric report, its components are assigned to a pipe spec. The pipe spec defines the properties, such as dimensions or material characteristics, for the components in the pipe spec including bolt sets. Pipe spec administration is the responsibility of an administrator and it is managed with the COMOS PipeSpec Designer product before COMOS Isometrics is used.

Each engineering object represents a real component with all its specified properties in an isometric report. The use of manufacturer devices on the working area is based on the function of the COMOS PipeSpec Manager. Pipe spec data are filtered based on the selected parameters "Nominal diameter" and "Pipe spec", for example, and valid components are offered for placement.

The specification of components in the parts catalog also includes the geometric properties.

Object-oriented technology

COMOS Isometrics fully supports the object-oriented technology of COMOS. This means you also have access to the standard functionalities of COMOS such as evaluation of objects by queries and reports. The representation on the isometry is always a representation of the current 3D properties of the components.

Areas of application

Depending on the configuration, isometries can reflect different aspects of a plant lifecycle.

You can, for example, create construction isometries with integrated parts list. You can also divide them into different production spools, if necessary. Each spool receives its own isometric report and a parts list in the process. You can also transfer production data to programs for production machines (such as bending machines).

You can create assembly isometries for assemblies that include relevant data for pipe installation at a later time.

You can also create the following isometries:

- Compilation isometries from different individual isometries
- Coating isometries
- Inspection isometries for representation of inspection cases and results
- Maintenance isometries for description of maintenance processes

All these reports are based on the same objects created in COMOS.

Creating the project structure

3

3.1 Organization of isometric pipe engineering

Basics

For isometric engineering of pipes you need an engineering object with the respective defaults. You create the objects for isometric pipe engineering in hierarchical order in the Navigator under a "Subunit" in the "Units" tab.

The following layers are available to structure the objects of isometric pipe engineering:

- Folder
- Pipe
- Pipe section

Folder

In the "A40 Pipes" folder, you create pipes and evaluating reports about pipe parts that are used or not used.

Pipe

The "Pipe" object is a structure object. You create isometric reports under this object in which you engineer the pipe.

A pipe consists of at least one pipe section. You usually create a pipe section after drawing the pipe isometry. You can also create a pipe section separately in the Navigator.

Alternatively, you can create a document folder below the pipe. You can create isometric reports, such as for shared revision management, under this document folder.

Pipe section

The "Pipe section" object is a structure object. The components of a pipe are stored under this object. Each component is assigned to exactly one pipe section. The pipe section is referred to as owner with respect to the component. If a vessel nozzle or pump nozzle is not connected to a vessel or a pump, the pipe section is the owner.

See also

Creating and opening an isometric report (Page 16)

3.2 Creating an engineering project

Requirement

You have the necessary rights for creating an engineering project.

Procedure

1. Start COMOS.
   You can find additional information on this topic in the "COMOS Platform Operation" manual, keyword "Starting COMOS".
2. Create a engineering project.
   You can find additional information on this topic in the "COMOS Platform Administration" manual, keyword "Creating a new project".
3. In the project properties click the "..." button in the category "Project data" in the field "Project structure".
   The "Select project structure for <project name>" window opens.
4. Select the node "@30 > M00 > A20 > A10 > A10 Project presetting, general".
5. Confirm your selection with "OK".

Result

You have now created a new engineering project that you can select from the list of available projects.

3.3 Creating a unit and subunit

Procedure

To create a new unit with a new subunit below, proceed as follows:

1. In the "Units" tab of the Navigator, right-click to select the project node of your engineering project.
2. Select the "New > Plant (general)" command from the context menu of the project node.
   The plant is created in the Navigator.
3. Click the "New > Unit" command in the context menu of the plant.
   The unit is created in the Navigator.
4. Click the "New > Subunit" command in the context menu of the unit.
   The subunit is created in the Navigator.

3.4 Creating a "Pipe" object

Requirement

A unit and at least one subunit have been created. There is an automatically created "A40 Pipes" folder below the subunit in the Navigator. Underneath this folder, you can create any number of pipes.

Procedure

1. Right-click the "A40 Pipes" object to select it.
2. Click the "New > A10 Pipe" command in the context menu.

Result

The "Pipe" object is created in the Navigator. Such an object is required to create an isometric report.

Engineering with COMOS Isometrics

4

4.1 Workflow

Requirement

- The administrator has defined pipe specs.
- You have created the project structure. See chapter Creating the project structure (Page 11).

Task description

To create a pipe isometry based on predefined pipe specs below a pipe, proceed as follows:

- Creating an isometric report.
  See also chapter Creating and opening an isometric report (Page 16).
- Drawing the pipe run.
  See also chapter Drawing the pipe run (Page 26).
- Placing components.
  See also chapter Inserting the component on the pipe run (Page 41).
- Labeling components (optional).
  See also chapter Creating tag symbols (Page 54).
- Dimensioning pipes.
  See also chapter Automatic dimensioning of pipe run (Page 62).
- Creating spool isometries (optional)
  See also chapter Creating spool isometric drawings (Page 82).
- Creating compilation isometries (optional)
  See also chapter Placing an isometric report in a compilation isometric drawing (Page 86).

4.2 Isometric reports

4.2.1 Isometric report types

Basics

An isometric report is an interactive report, the type of which determines the functionality. You specify the type of isometric report in the isometric report template.

The following types can be distinguished:

- Isometric report
- Spool isometric report
- Compilation isometric report

Isometric report

In the isometric report, you create the pipe engineering based on an isometric representation.

Spool isometric report

A spool isometry divides an isometric pipe engineering into individual production units. You define the individual production units in the isometric report with spool marks. You display spool isometries in production reports.

See also chapter Basics of spool isometries (Page 78).

Compilation isometric report

You place several isometric reports in a compilation isometric report for documentation purposes.

See also chapter Basics of compilation isometries (Page 84).

4.2.2 Creating and opening an isometric report

You create an isometric report in the "Units" tab below a pipe in the Navigator. The available isometric report templates are displayed in the context menu. The editing options depend on the type of isometric report template used.

Requirements

The project structure has been fully created. See chapter Creating the project structure (Page 11).

Procedure

To create an isometric report, proceed as follows:

1. Open the "Units" tab of the Navigator.
2. Select the "New > <Isometric drawing>" command from the context menu of the pipe. The following formats are available for selection:
   - "Isometric drawing DIN A3"
   - "Isometric drawing DIN A3 with BOM"
   - "Isometric drawing DIN A2"
   - "Isometric drawing DIN A2 with BOM"

You create a production report with a spool isometry from an existing isometric report. See also chapter Creating spool isometric drawings (Page 82).

Result

The isometric report is created below the pipe. To open an isometric report, double-click the object.

See also

Organization of isometric pipe engineering (Page 11)

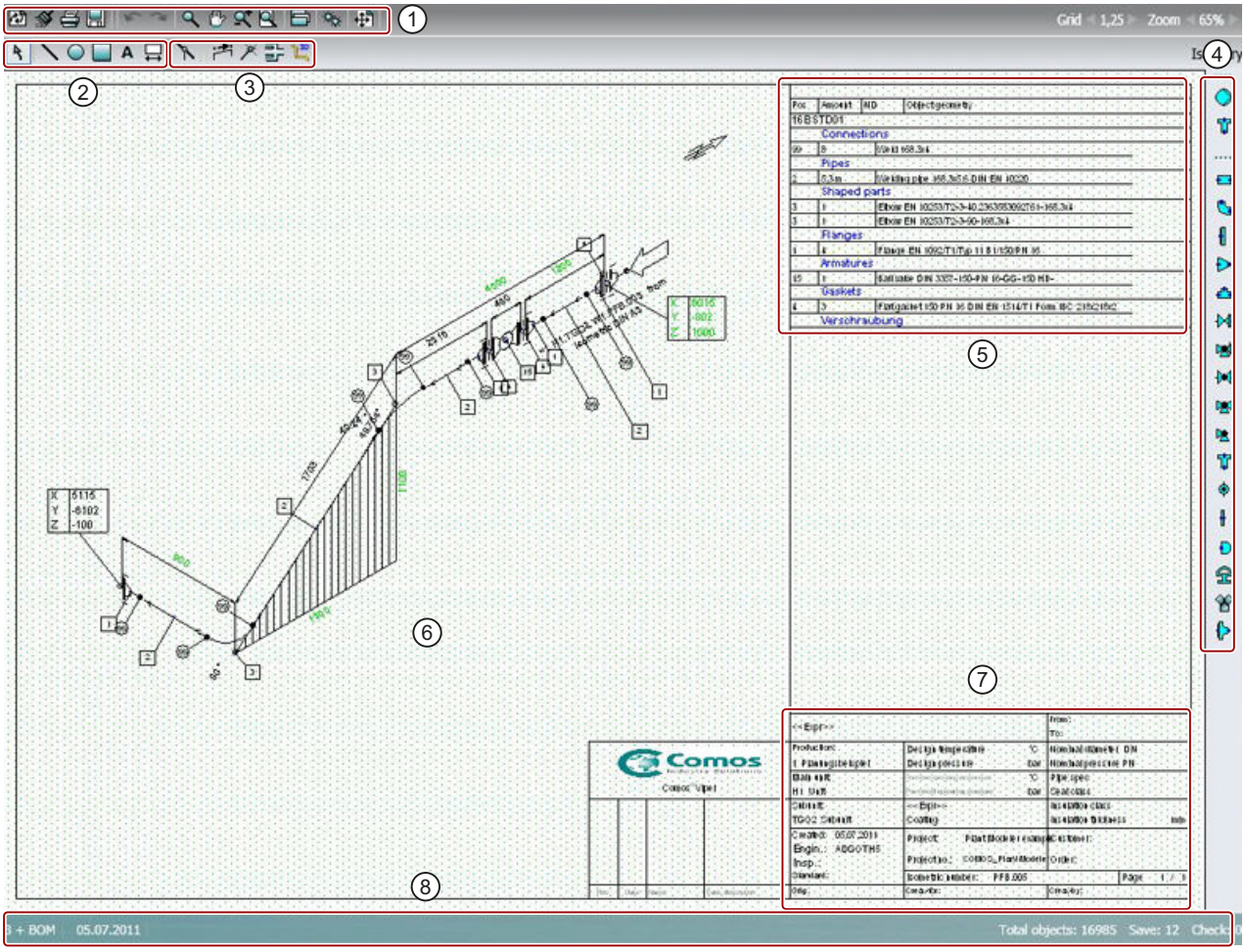
Structure of an isometric report (Page 17)

4.2.3 Structure of an isometric report

In the isometric report, you create the pipe engineering based on an isometric representation. An isometric report is an interactive report In COMOS. This means you have all editing options of an interactive report in an isometric report.

Overview

The following figure shows an isometric report with integrated bill of material:



① Toolbar
② Default toolbar
③ "Isometry" toolbar
④ Symbol bar
⑤ Integrated bill of material
⑥ Working area
⑦ Drawing header
⑧ Status bar

"Isometry" toolbar

The following table describes the buttons on the "Isometry" toolbar:

Button	Function	Description
	"Dimension"	Creates an individual dimension between two connectors or a coordinate dimension.
	"Connection"	Draws a pipe run.
	"Calculate"	Calculates the isometry based on components and their dimensions.
	"Create dimensioning and labeling"	Adds labeling symbols as well as automatic dimensions for all levels.

Button	Function	Description
	"Mapping P&ID to 3D"	Opens the window "P&ID objects" and lists all objects from the P&ID in flow direction that are associated with the pipe. If there are no objects from a P&ID, the window remains empty.
	"3D read-only"	Opens the window "3D view" and displays the objects on the isometry in space.

Working area

The working area is the actual area in which the isometry is displayed and labeled. The pipe run as well as the pipe components are drawn or placed here. You can add text or graphical elements to an isometry with the text tool or the line and elbow tool.

The following can also be displayed here:

- Dimensions
- Pipe numbers
- Auxiliary symbols
- Text symbols such as position numbers
- Component data

Symbol bar

You can place components on the isometric report via the symbol bar by means of drag&drop. Depending on the pipe spec and nominal diameter that were set, the icons in the symbol bar belonging to a component can be set to active or inactive, depending on whether or not they are available in the pipe spec.

If required, the symbol bar can be changed and extended. Additional information on this topic can be found in the "COMOS Platform Operation" manual, keyword "Editing the symbol bar".

If you select the icon of a component type in the symbol bar, you can make additional settings in the toolbar using additional buttons:

- Type of insertion point
- Pipe spec
- Nominal diameter
- Object-specific properties

The component of a component type that you can place on the isometry with drag&drop depends on the pipe spec settings of the administrator and on whether you have specified preferred components.

See also chapter Changing symbol bar for specific isometric drawings (Page 105).

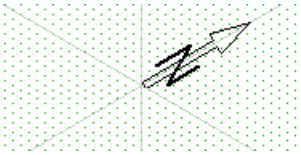
4.2.4 Coordinate cursor and grid

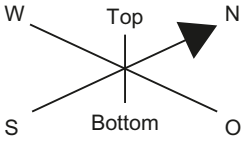
Drawing on the working area is always conducted along the axes of an oriented coordinate axis and on the set grid.

You can only drop components on the points given by the grid. In some cases, the grid is left automatically.

Example: If a pipe was placed in a special direction, possible connectors may be located outside the grid. To place components on the pipe afterwards, turn off the grid temporarily by pressing the <Shift> key.

- The size of the grid can be set in the standard toolbar.
- By default, the alignment of the crosshair cursor is as follows:





Orientation of the north arrow at 30°

The North-South axis corresponds to the Y axis, the East-West axis to the X axis, and the Up-Down axis to the Z axis.

North arrow

The direction "plan north" is specified beforehand in the report template. The crosshair serves as an orientation help and is especially helpful for setting special directions.

See also chapter Checking the "SYSISO" tab (Page 104).

The rotation angle of the north arrow can only be changed on a blank isometric report. See also chapter Changing the north arrow (Page 20).

If you use an axes folder, settings for the north arrow made there will override other settings.

See also

Creating unit grids (Page 50)

4.2.5 Changing the north arrow

A north arrow is located on isometric reports for orientation purposes. See also chapter Coordinate cursor and grid (Page 20).

You can change the rotation angle as well as the appearance of the north arrow for reports.

Requirement

- You have created a report.
- The isometric report is closed.
- Objects have not been saved on the report yet.

Procedure

1. Select the required isometric report in the "Units" tab of the Navigator.
2. Select the "Properties" command in the context menu.
3. Open the "Attributes > SYSISO" tab.
4. Make the required changes:
   - To change the rotation angle, click the associated selection button in the "Rotation angle for north arrow" field and select the new rotation angle.
   - To change the appearance of the north arrow, click the "North arrow" field and select a different north arrow representation.
5. Save your entries.

Result

The selected changes are applied to the selected report.

You can only place isometries with the same north arrow orientation in a compilation report.

4.2.6 Create report net diagram

Requirement

Ghostscript and Graphvis from AT&T, for example, must be installed in order to create a PDF from a PostScript file.

Procedure

1. In the report, open the context menu of an object with a reference flag which is used on more than one isometry.
2. Select the "Options > Create report net diagram" command.
   The "Create report net diagram" window appears.
3. If the associated reports are to be displayed in an XML tree, select the "XML output" option.

4. If the connected reports should be output in a net diagram, enable the "Output network plan" option.
   Fill out the displayed fields.
   - "Converter utility": Path to the EXE file of the converter utility. Specify the parameters for the converter utility in the field below.
     Example:
     "C:\Program Files\Graphviz2.26.3\bin\dot.exe"
     "-Tps “{O}’ -o “{1)""
   - "Postscript command line tool": Path to the EXE file of the postscript command line tool. In the field below it, enter the parameters for the postscript command line tool.
     Example:
     "D:\Programs\gs\gs8.71\bin\gswin32c.exe"
     "-q -dBATCH -dNOPAUSE -dPDFFitPage -dCompatibilityLevel#1.4 -dPDFSETTINGS#/screen -sDEVICE=pdfwrite "-sOutputFile={1}" "{O}'"
   - "Converter output file":
     Path and file name of the net diagram to be created. Possible data types are PNG, Postscript, and PDF. The data type that can be output depends on the specified converter. If the required software is installed, you can specify the PDF data type.
     Example:
     "D:\TEMP\dot\digraph1.pdf"
5. Click on the "Start" button.

Result

If you have activated the "XML output" option, the XML file is displayed by default in Microsoft Internet Explorer.

If you have activated the "Output network plan" option, the net diagram is saved in a file with the appropriate data type in the specified directory.

4.3 Designing the pipe run

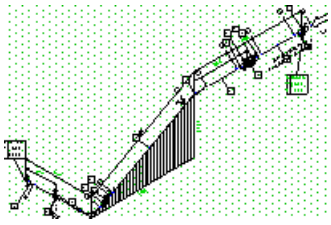
4.3.1 Isometric drawing concept

Isometric pipe engineering takes place in the isometric report based on a three-dimensional coordinate system.

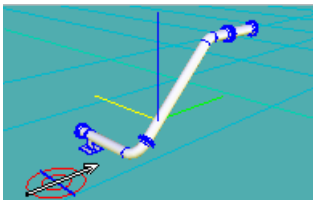
Pipe engineering in isometric view

The pipe run is displayed schematically in the isometric view of the isometric report. All objects used in isometric pipe engineering have 3D properties. New components are placed in the form of graphic symbols on the pipe run of the isometric report.

Use the 3D view of the pipe run to check the entered values.



You specify the required dimensions as well as the position in space using coordinates, dimensioning and angles.



A 3D model is calculated from isometric pipe engineering in the 3D view based on the entered dimensions.

See also

Basics for editing (Page 23)

Components with dynamic connectors (Page 44)

4.3.2 Basics for editing

Basics

Each isometric report is based on a three-dimensional coordinate system. You draw the pipe run in this coordinate system by adding points in the isometric report with the connection tool. The points are automatically connected by lines. The lines will always follow one of the three main axes in X, Y or Z direction.

Flow direction in pipe runs

The flow direction in pipe runs is indicated by an arrow. The flow direction is usually the drawing direction.

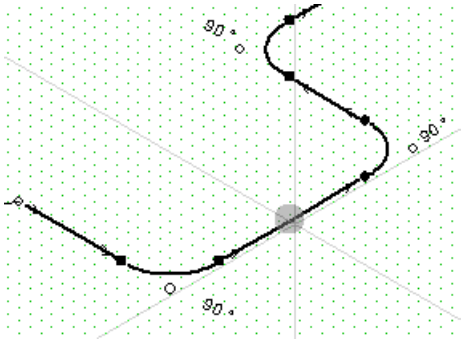
You can adapt the flow direction. See also section Editing the flow direction (Page 30).

Pipes or components with contradicting flow direction are marked as "inconsistent".

Branches in pipe runs

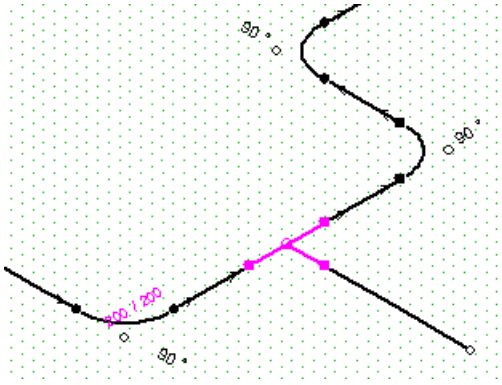
You can insert branches in a pipe run. The branch points are displayed when you move the mouse above the pipe run. Draw the continuing pipe run starting at the branch point.

The following figure shows a branch point in the pipe run:



The branch table of the pipe spec determines the component that is installed in the pipe run for the branch.

The following figure shows a t-piece automatically added in the pipe run:



Dimensions and calculations

You can create dimensions on different layers, by default 3. Dimensions are either created for the entire isometric drawing on the report using the context menu or manually for individual components using a dimensioning tool. See also section Dimensions (Page 59).

You usually use default pipe lengths during construction. Assign the actual pipe lengths once you have created the dimensioning symbols.

Dimensioning is followed by the calculation of the pipe lengths. The calculation transfers the values to the pipe system. See also section Calculating the pipe run on a report (Page 75).

Slopes in pipe runs

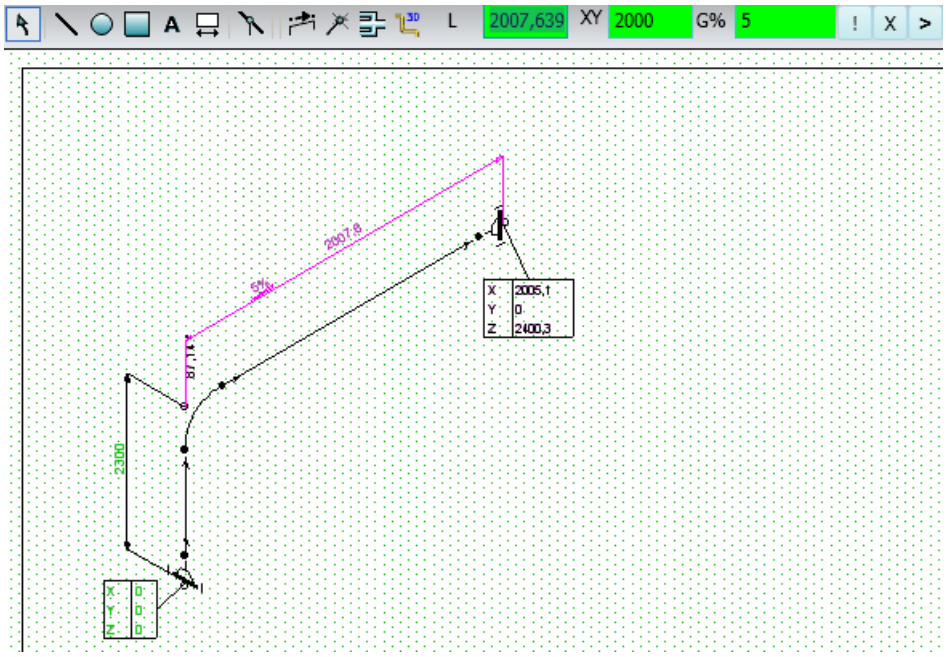
You define slopes up to 17.632% (corresponds to 10°) for a pipe in X/Y direction when dimensioning the main layer. See also section Editing slopes (Page 31).

Dimensioning determines the direction with which the slope is added:

- If you have created automatic dimensioning, the drawing direction of the pipe run is used.
- If you have created manual dimensioning, the drawing direction of the dimensioning is used.

The slope is always displayed at the end of the dimensioning.

The following figure shows a pipe run with a slope of 5%. The arrows in the pipe run indicate the flow direction:



You can display slopes greater than 17.632% by drawing special directions. See also section Create slopes and special directions (Page 27).

See also

Drawing the pipe run (Page 26)

Connectors (Page 25)

4.3.3 Connectors

Basics

The connectors at an isometric symbol indicate at which point of the component you can connect other parts.

We distinguish between the following types of connectors:

- Logical connectors
- Physical connectors
- Dynamic connectors

Logical connectors

Logical connectors correspond to the connections at the engineering objects. You connect the components in the isometric report using the connectors.

Physical connectors

You connect graphical report elements, such as dimensions, to the physical connectors. Physical connectors are determined by the respective connection type and define the physical end of a component.

If one of two connected components has the connection type "generic", the physical end of this component is set to match the connection type of the other component. If both connection types have the value "Pipe end generic" or "Transition bend", a welding seam is set by default if not defined otherwise.

If you want to dimension physical connectors (displayed in orange), keep the <Ctrl> or <Shift> key pressed during selection.

Dynamic connectors

You connect branches or components to dynamic connectors. Dynamic connectors are created dynamically and are not predefined in the base object. Dynamic connectors are also created in the database.

See also

Components with dynamic connectors (Page 44)

Basics for editing (Page 23)

Example configuration (Page 110)

Individual dimensioning of a pipe run (Page 63)

4.3.4 Drawing the pipe run

Basics

You draw a pipe run with the connection tool along the X, Y or Z axis. The drawing direction also determines the flow direction in the pipe run. If possible, draw the pipe run in the intended flow direction. Allow sufficient space for components.

Press the [Esc] button to exit the connection tool and cancel polygon creation.

Requirement

An isometric report is open.

Procedure

1. Select the "Connection" button in the toolbar of the isometric report.
   The selection boxes for pipe spec and nominal diameter are displayed in the toolbar.
2. In the "Pipe spec" box, select the pipe spec and in the "Nominal size 1" box select the nominal diameter used for the pipe run.
   The selected pipe spec specifies the components you are using and how they are installed according to the COMOS pipe spec concept. This makes facilitates automatic construction.
   See also chapter Introduction (Page 9).
3. To determine the starting point of a pipe run, click on the required position in the working area or on a free connector to continue an existing pipe.
   An isometric crosshair appears at the mouse pointer.
4. To determine the intermediate points of the pipe run, click on the required positions in the working area.
   All intermediate points along the main axes are connected by a line beginning at the starting point.
5. Press <Backspace> to delete the last intermediate point.
   The intermediate point is deleted when you exit the connection tool.
6. Right-click in the working area to finish polygon creation or click an existing free connector ion as termination point.
   - If an existing pipe was extended, the new pipe run is created.
   - If no existing pipe was extended, the "Select owner" window opens. The pipe is already selected in the "Units" tab located in the window. See also chapter Organization of isometric pipe engineering (Page 11).
     In the "Select owner" window, you either select an existing pipe section below the pipe or you create a new pipe section.
     Click "OK" to save your entries and close the "Select owner" window.

Result

The pipe run is created and automatically equipped with pipe components from the selected pipe spec. The pipe components are connected according to the pipe spec definition. The pipe components are created under the pipe section in the Navigator.

4.3.5 Create slopes and special directions

To draw a slope or a special direction, define a starting point and a target point in the space when drawing the pipe run. You create the target point using auxiliary points. Auxiliary points are only visible during creation of special directions.

- You define a two-dimensional special direction using one auxiliary point (layer jump).
- You define a three-dimensional special direction using two auxiliary points (space jump).

Definition

- Special directions deviate from the run of a main axis by more than 17.632%. Special directions are displayed on isometries deviating from the run of the main axes.
- Slopes are a type of special direction which deviate from the run of the main axes by less than 17.632%. Slopes are demonstrated on isometries as main axes. They only detect slopes based on their dimension.

Procedure

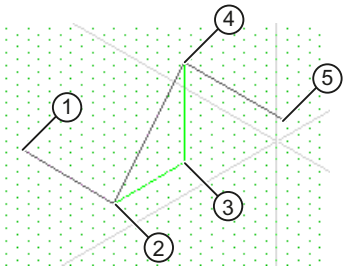
To create a slope or a special direction, proceed as follows:

1. Use the "Connection" tool to draw a pipe run or just a starting point.
2. While drawing hold the <V> key down until you have set one or two auxiliary points with the "Connection" tool.

Result

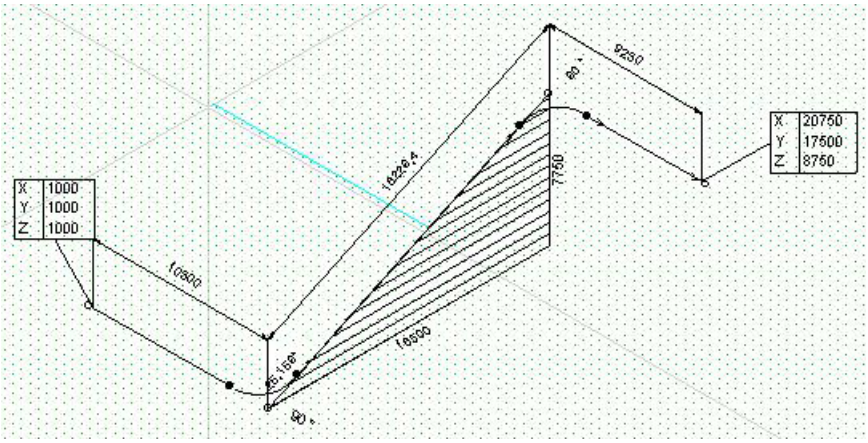
The auxiliary points are connected to green auxiliary lines beginning at the starting point and indicate the position of the target point.

The following figure shows the construction of a simple space offset with the help of an auxiliary point.



① Starting point of the pipe run
② First intermediate point and starting point of the special direction (space offset): <v> is pressed.
③ Auxiliary point of the special direction (space offset): <v> is released.
④ Second intermediate point and end point of the special direction (space offset)
⑤ End point of the pipe run

The following figure shows the result with the resulting special direction:



If you place a branch at a space offset, four additional special directions are available. These special directions run at an angle of 90° to the space offset and are highlighted in color during drawing. With 90° branches at special directions, these directions are shown in turquoise.

If you have drawn a special direction in the pipe run, the special direction is saved. If you draw another pipe run in the same special direction, the special direction is offered as a yellow line. If you delete all pipe components of the original special direction, the special direction is no longer offered.

See also

Editing slopes (Page 31)

4.3.6 Editing the pipe run

4.3.6.1 Principle

You can always continue a pipe run at the beginning and end on an isometric report. To do so, select the connection tool and click the start or end point of the pipe run.

Requirement

- An isometric report is open.
- A pipe run is configured.

4.3.6.2 Editing the flow direction

Basics

- Flow direction
  The flow direction is usually specified by the drawing direction with the connection tool. The flow direction is subject to physical laws and therefore applies to the entire pipe run. A flow direction can be defined in components, such as pumps.
  The original flow direction is retained when you separate a pipe.
  You use the context menu of a component to set, change or delete the flow direction. You decide in the context menu if you want to change the flow direction only for the selected component or for the entire pipe run.
- Drawings without flow direction
  If a flow direction is not defined in drawings, you can specify a flow direction with the context menu.
- Flow direction arrows
  Flow direction arrows are drawn for each pipe. Flow direction arrows are not printed.
- Automatic alignment of components
  If components have a specified flow direction through their IO connectors, they are immediately installed correctly into the pipe. If the flow direction of the pipe changes, the components turn around as well.
- Tees
  If T-piece s are installed, an undefined flow direction "Neutral" is initially preset for the end that is not connected.

Procedure

To adjust the flow direction in the pipe run, proceed as follows:

1. Select one or more components.
2. Select one of the following menu commands in the context menu of the selection under "Options > Flow direction":
   - "Set > Pipe run"
     This command creates a flow direction for this pipe run. If there is no flow direction, it is created during setting.
   - "Set > Selected components"
     This command creates a flow direction for the selected components of a pipe run.
   - "Delete > Pipe run"
     This command sets the flow direction of a pipe run to neutral.
   - "Delete > Selected components"
     This command sets the flow direction of selected components to neutral.
   - "Change > Pipe run"
     This command reverses the existing flow direction of a pipe run. If the flow direction was neutral, it remains so after the command is executed.
   - "Change > Selected components"
     This command reverses the existing flow direction for the selected components of a pipe run. If the flow direction was neutral, it remains so after the command is executed.

4.3.6.3 Placing a branch in the pipe run

The branch table of the pipe spec determines which component is installed as the branch in the pipe run.

Procedure

To place a branch in a pipe run, proceed as follows:

1. Select the connection tool in the toolbar of the isometric report.
   The selection boxes for "Pipe spec" and "Nominal diameter" are displayed in the toolbar.
2. Select the "Pipe spec" and the "Nominal diameter" used for the pipe run and activate the "Fix nominal diameter" option, if necessary.
3. Click the required branch point in the pipe run.
4. Draw the pipe run.
5. Select the owner of the pipe branch in the "Owner" window.

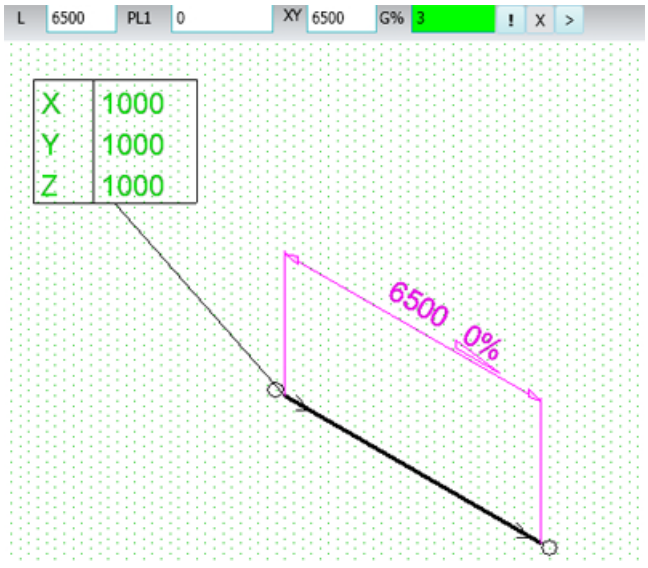
4.3.6.4 Editing slopes

You can only create slope dimensions on level 1. You can only assign main dimensions to a slope for this reason.

To create an angle of up to 10°, the pipe run must run in X/Y direction.

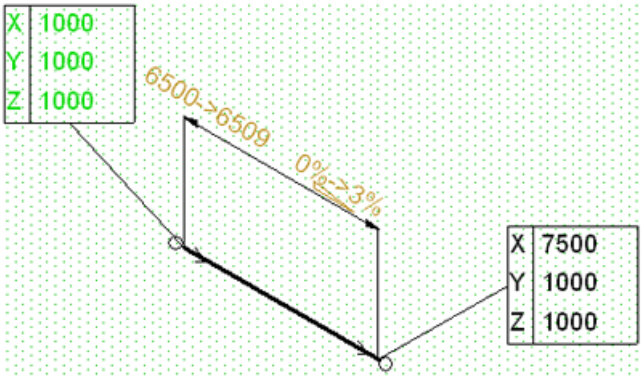
Procedure

1. Click the dimensioning of the pipe run.
   The "G%" field is shown in the toolbar.
2. Enter a value in the "G%" field of the toolbar greater than zero to change the slope.

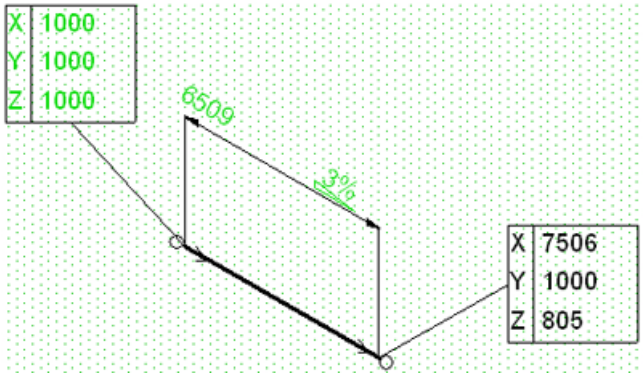


3. Click ! on the toolbar to apply the value.
   The "Slope assignment" window opens.
4. In the "Slope assignment" window activate either
   - None of the listed options
   - "Assign slope to complete pipeline"
   - "Apply settings from current session"
     If this option is activated, you can only turn off this functionality by exiting COMOS.
5. Click the "OK" button in the "Slope assignment" window to confirm your entries.
   The changes in the dimensions are calculated and displayed in an ochre font.

6. To specify the position in space, select a coordinate dimension and click ! in the toolbar.
   The selected coordinate dimension is displayed in a green font.

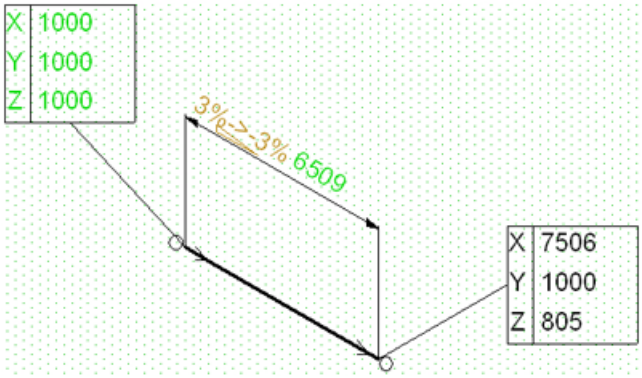


7. Click the "Calculate" button.
   The calculated dimension is applied and displayed in a green font.

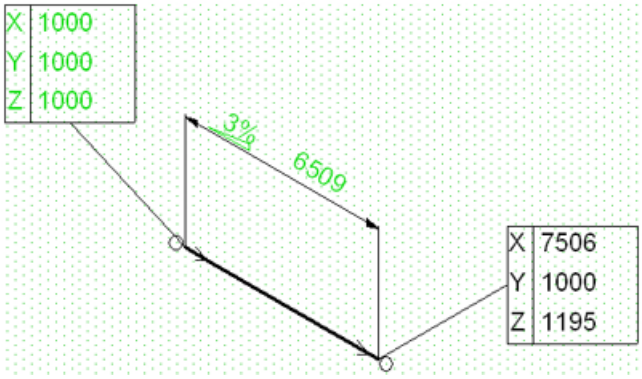


Reversing the slope direction

1. If you want to reverse the slope direction, select the required dimension and enter a negative value in the "G%" field.
2. Click ! on the toolbar to apply the value.
   The change in the slope direction is calculated and displayed in an ochre font.



3. Click the "Calculate" button.
   The change in slope direction is displayed in a green font.



See also

Automatic dimensioning of pipe run (Page 62)

Individual dimensioning of a pipe run (Page 63)

4.3.6.5 Replacing elbows with bends

Requirement

The used pipe spec includes bends.

Procedure

1. Select one or more elbows on an isometric report.
2. Open the context menu of a elbow.
3. Select the command "Options > Replace bows with bends".

4.3.6.6 Moving the pipe run

Procedure

To move a pipe run to another isometric report, proceed as follows:

1. Select one or more random components in the pipe run.
2. Move the components with drag&drop to the required position.

Result

The pipe run is adapted to the new positions.

4.3.6.7 Centering a pipe run on an isometric report

Procedure

To center individual components or several pipe runs in an isometric report, proceed as follows:

1. Make sure that no component is selected.
2. From the context menu of the working area, select the command "Options > Center drawing".

All components are centered in the isometric report. The distances between components remain the same.

See also

Continuing a pipe run on another isometric report (Page 35)

4.3.7 Continuing a pipe run on another isometric report

If there is not enough space for configuration of a pipe run on an isometric report, continue the pipe run on another isometric report.

Regardless of this approach, you still have the option of displaying the entire pipe run in connected form in a compilation report at a later time. See also chapter Compilation isometric drawings (Page 84).

Requirements

- An isometric report with a configured pipe run is open.
- Another isometric report is open on which you want to continue the pipe run.

Procedure

To continue a pipe run on another isometric report, proceed as follows:

1. Select the object at the end of the pipe run on the first isometric report.
2. In the context menu, click the command "Navigate > Object".
   The respective end piece is selected in the Navigator.
3. Select the tab with the second isometric report.
4. Use drag&drop to move the selected end piece from the Navigator to the second isometric report.
5. Confirm the query whether you want to place the existing object on the original report several times.
6. Continue with engineering the pipe run.

Result

The continuation of the pipe run is displayed in both isometric reports by references. See also chapter Creating tag symbols (Page 54).

4.4 Using components

4.4.1 Display of components

Components are displayed as two-dimensional symbols on an isometric report. Each component must have a symbol for the drawing type "M25_P1" for this purpose.

Components whose associated 3D object was locked are displayed in blue. See also chapter Locking or releasing 3D objects (Page 74).

Alignment of components

As long as components are rotationally symmetric, the alignment of the component with regard to the position in space is irrelevant. If you rotate such a component symbol on an isometric report, the rotation does not have an effect on the display in the 3D preview.

For components, such as T-pieces or multi-way valves, the alignment of the component also determines the relative position of its component connections in space. In this case, the rotation angle of the component is relevant for 3D engineering. If you rotate such a component, the rotation angle relative to the neighboring component is also displayed in the status bar. Pipe runs on rotated connections of these components run along the rotation angle. The special direction for a pipe run in the rotation angle is created automatically.

See also

Components with dynamic connectors (Page 44)

Inserting the component on the pipe run (Page 41)

4.4.2 Changing preferred components

Basics

The individual components are grouped according to component types in the symbol bar next to the user interface of an isometric report. See also section Isometric reports (Page 15). The components that are allocated to a component type and can be installed for the selected pipe spec and nominal diameter can be seen with a right click on the respective icon in the symbol bar.

The component displayed in bold is the one which is placed by default on an isometry using drag&drop (default preferred component). Default preferred components are likewise used when additional components are to be created automatically as well, as soon as you place a component on a pipe run, connect pipe runs with one another, or if the course of the pipe run requires so. The default preferred components for individual pipe specs are defined based on the nominal diameter in the Pipe Spec administration using the PipeSpec Designer module.

You can find more information on this topic in the "PipeSpec Designer" manual, keyword "Preselected components".

Changing preferred components as user

If you want to use another component as preferred component of the same component type instead of the default preferred component, proceed as follows:

1. Select the required component symbol.
2. In the context menu, click on another available component in the displayed list.
   The component you clicked is displayed as new preferred component in bold.

To identify your change in the current COMOS session, the component is also marked with a check mark.

Limited existence of changed preferred components

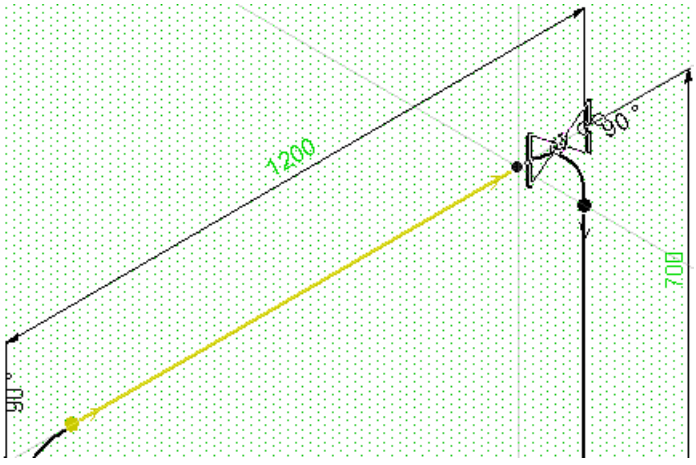
- When you shut down COMOS and restart it, the original preferred components are loaded once again.
- The preferred components are also reset after changing the pipe spec.
- An administrator can make permanent changes with the PipeSpec Designer module.

4.4.3 Insertion position and pipe length

You can influence the pipe length when you place components on a pipe or between a pipe and another component.

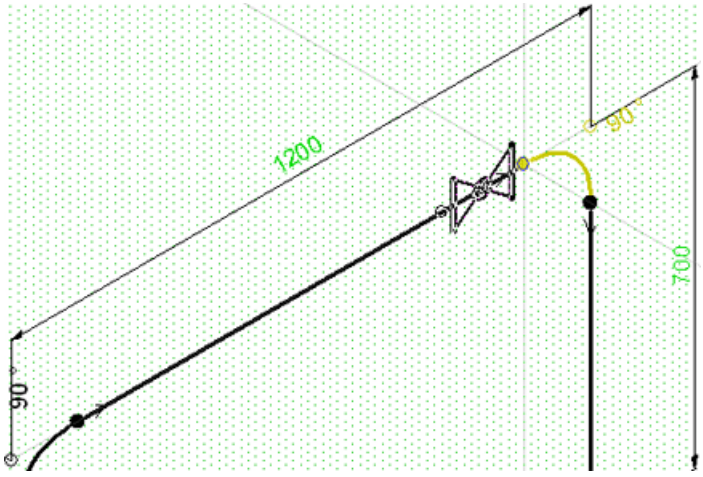
If you move a component above a pipe run while pressing the right mouse button, components available for installation are highlighted.

Highlighting pipe and placing components at one end of the pipe

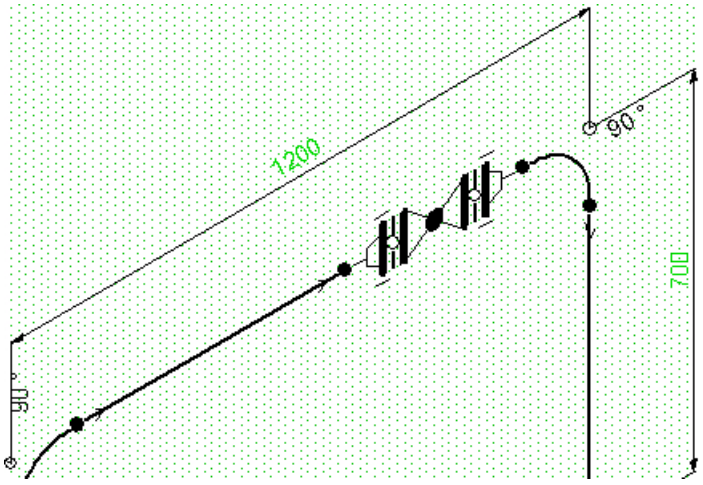


If you highlight a pipe and then place components with a click at one end of the pipe, the components are inserted at the end of the pipe. The pipe length remains the same. The elbow is moved:

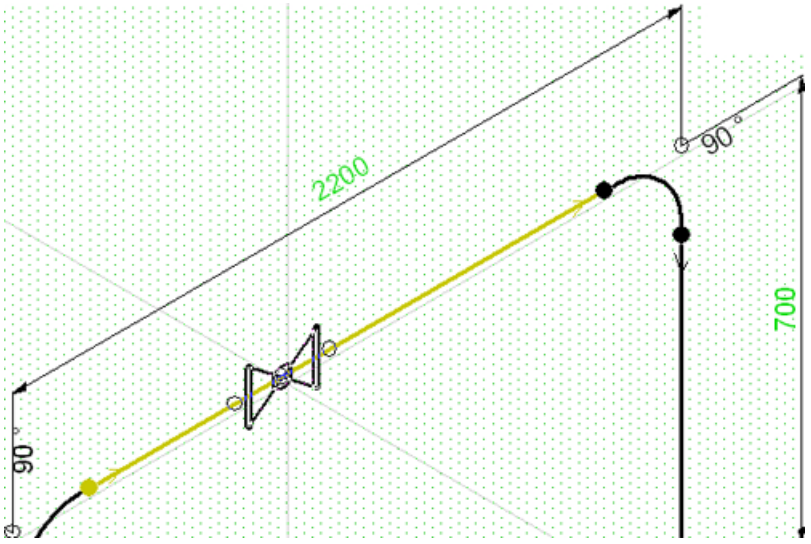
Highlighting elbow and placing components at one end of the pipe



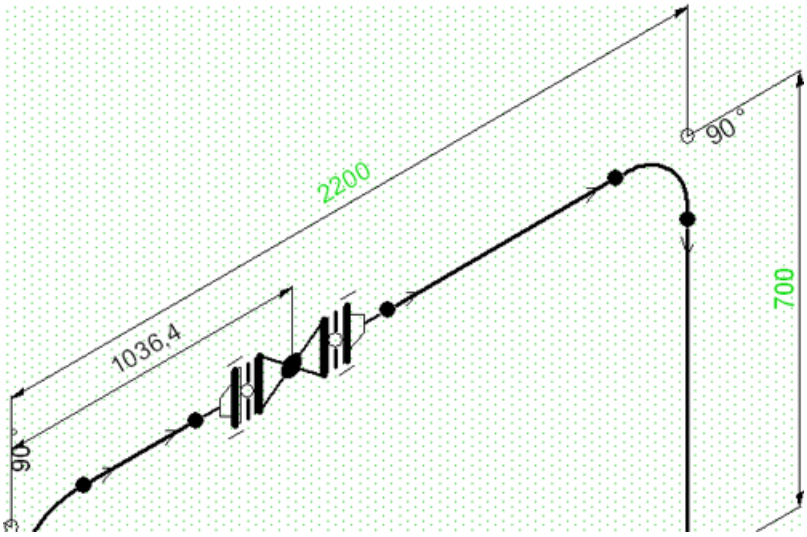
If you highlight an elbow and then place components with a click at the neighboring end of the pipe, the pipe is shortened:



Highlighting pipe and placing components approximately at the center of the pipe



If you highlight a pipe and then place components with a click approximately at the center of the pipe, components are placed relative to the insertion point and the pipe is replaced by two new ones. The original total length remains the same:



Undershooting the minimum length of a pipe

If you undershoot of the minimum length of a pipe by placing components, the "Place on the pipe" window opens. Answer the question "The pipe is too short to place the component. Do you want to lengthen the pipe?" by clicking the buttons "Yes", "No" or "Cancel".

4.4.4 Inserting the component on the pipe run

The defined components that are available to you are determined according to the pipe spec concept by selecting the pipe spec. See also chapter Introduction (Page 9).

Requirement

- An isometric report is open.
- A pipe run has been created.

Procedure

To insert a component into a pipe run, proceed as follows:

1. If the component you want to insert is connected to a component that is located between the spool marks, delete the spool marks first.
2. You have the following options to select the required component:
   - Click the component symbol in the symbol bar and move the mouse pointer on the isometric report.
     The symbol of the component is displayed at the mouse pointer.
     Additional lists and edit fields are displayed in the toolbar depending on the selected component. Here you can change the configuration of the component, if necessary. See also chapter Component-specific toolbar (Page 133).
   - If you want to first change the preferred component associated with the component symbol, specify a different preferred component in the context menu of the symbol. See also chapter Changing preferred components (Page 37).
   - If you have the right to access base objects, you can also drag the required component from the "Base objects" tab to the isometric report with drag&drop.
3. Position the mouse pointer on the required insertion position.
   - If the component can be placed in the pipe run, the pipe run is highlighted in color. The component is automatically aligned in the pipe run.
   - If "Automatic" has been set as insertion point, you can change the mounting direction in some components by pressing the <Ctrl> key.
   - If you want to place the component outside the pipe run, the "Select owner" window opens in which you can select an existing pipe section or create a new pipe section.
   - To influence the pipe length, move the new component above the pipe run with drag&drop. The component at which or in which mounting is permitted will be highlighted. The pipe length can change depending on the insertion location. See also chapter Insertion position and pipe length (Page 38).

4. To place the component, click the desired insertion position.
   COMOS checks whether the pipe and the component have the same nominal diameter:

The "Fix nominal diameter" option is activated in the component-specific toolbar.	The "Fix nominal diameter" option is not activated in the component-specific toolbar.
• Component with one nominal diameter If the nominal diameter of the component does not match the nominal diameter of the pipe, COMOS automatically installs reducers. • Component with two nominal diameters If neither of the two nominal diameters of the component matches the nominal diameter of the pipe, you cannot place the component. If only one nominal diameter of the component matches the nominal diameter of the pipe, the "Place reducer on pipe" window opens. You can make additional settings in this window.	The nominal diameter of the component to be installed is adapted to the nominal diameter of the pipe.

5. If the "Place reducer on pipe" window opens, activate one of the three listed options in this window:
   - "Do not adjust the pipe run"
     The reducer is created with the set nominal diameters, but is not connected to the pipe run at the database end.
   - "Adjust the pipe run"
     The nominal diameter of the pipe run is adjusted.
   - "Generate a counter reducer"
     A counter reducer is placed.

Result

The component is inserted into the pipe run and created as object in the Navigator. The component is automatically assigned to the pipe section which contains the corresponding pipe run.

The following objects can only be installed on an open end:

- Vessel nozzles and other objects with only one connector (except supports)
- Pipe elbows or other components with a change of direction

See also

Display of components (Page 36)

Deleting a component (Page 49)

Creating an isometric drawing from existing 3D data (Page 93)

4.4.5 Components with dynamic connectors

Basics

You can dynamically connect additional components in addition to defined connectors in the following components:

- Pipes
- Elbows, bends
- Tees
- X-pipe
- Y-pipe
- Blank flanges
- Flanges
- Covers
- Reducers

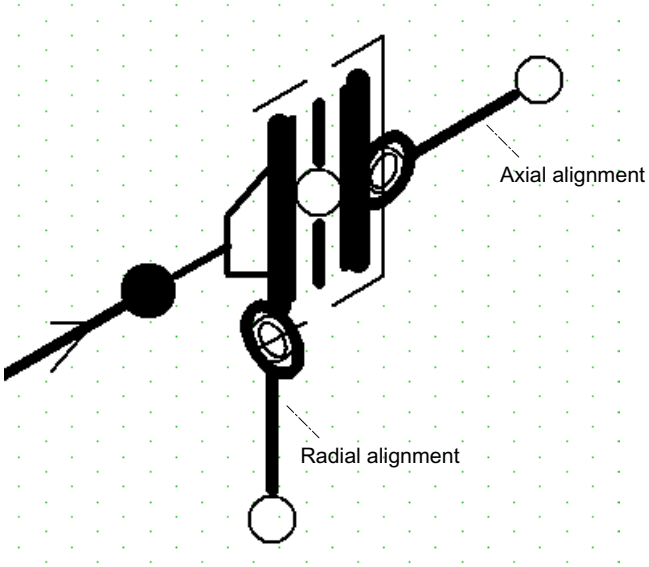
Principle

You define additional connectors with "dynamic connectors". These are automatically created at the permitted locations.

Alignment

You specify the alignment for each dynamic connector that is used to connect the component to another component.

The following figure shows possible alignments for dynamic connectors:

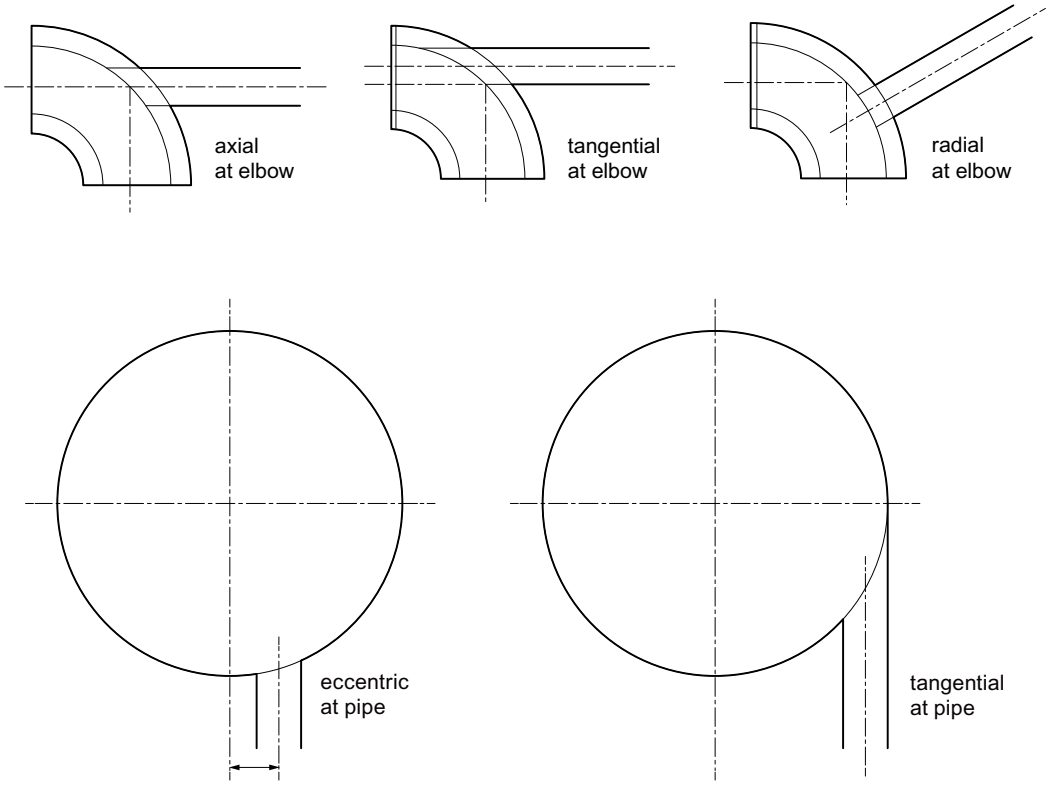


- Axial alignment: The component is connected at the end of a component, for example.
- Radial alignment: The component is connected on the shell of a component, for example.

Dynamic connections are always possible at pipes.

For all other components, the directions in which dynamic connections are permitted depends on the specified properties of the isometric symbol. See also section Connectors (Page 25).

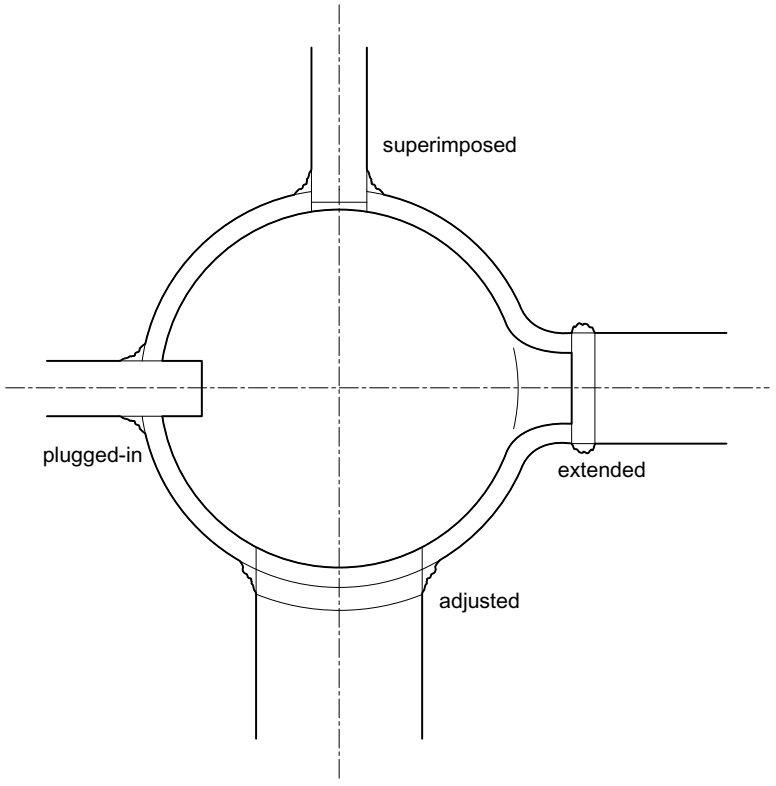
Example illustration of frequently used branch types



The properties specified for the respective isometric symbol determine which branches are possible for elbows. This includes placing the relevant symbol parts on layer 110 or 111 to 114.

Branches on pipes are always possible, regardless if they are radial, eccentric or tangential.

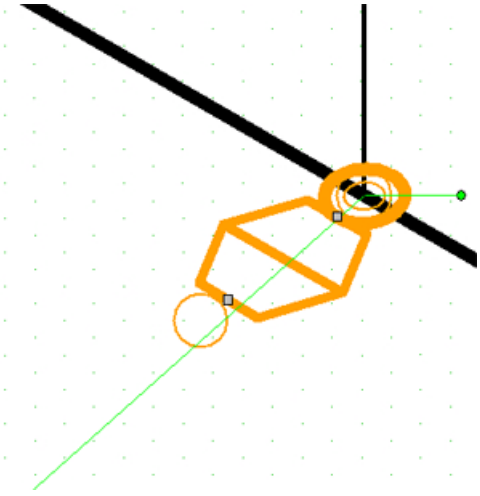
Weld shapes of connectors



If a certain weld is specified in the pipe spec administration for a nominal diameter combination, this weld with the respective weld form will always be used for a specific nominal diameter. See also section Changing preferred components (Page 37).

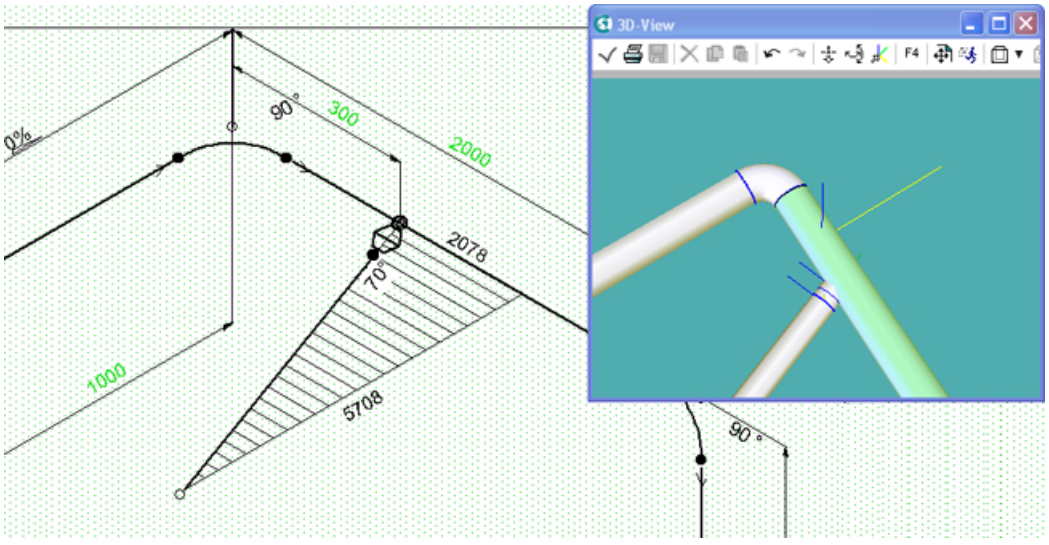
Rotating dynamically connected components at a later time

If no additional components have been connected to a dynamically connected component, you can rotate the component in 15° increments using the handles. The current angle is displayed in the status bar during the rotation.



Changing the connection angle at a later time

If you select a dynamically connected component, the connection data of this component are displayed. You can change the connection data. Depending on the component, other setting options are offered on the specific toolbar of the isometric drawing. Contact your administrator if you have questions regarding configuration of the "Y00T00011.Y00A02625" attribute. The following figure shows the status of a change in the connection angle at a later time. The change has an effect on the isometric drawing as well as the 3D model.



See also

Isometric drawing concept (Page 22)

Display of components (Page 36)

Creating symbols (Page 107)

4.4.6 Grouping components

You can group several components. You can remove individual components from a group. The commands for working with groups is available in the context menu under "Grouping".

Procedure

To group components, proceed as follows:

1. Select the components in the isometric report.
2. Select the respective command under "Grouping" in the context menu of the selection. The selected components are grouped.

4.4.7 Changing the bill of material relevance of an object

If you are planning an isometric pipe run, all pipes and pipe parts are by default considered in the bill of materials. For pipe parts which you are able to select on the isometry you may switch off the BOM relevance.

Mounting parts are always displayed on bills of material if the owner is displayed.

Procedure

1. Select the object.
2. Select the respective command under "Bill of material relevant" in the context menu of the selection.

Result

The object is no longer included in the bill of material. It is indicated by a dotted line in the isometric report.

4.4.8 Connecting or disconnecting components

If components are on the same axis, you can connect the components to each other.

Procedure in typcial case

1. Select both components.
2. Select the "Options > Connect" command in the context menu of the selection.

Result

The two components are connected. To disconnect components, select the command "Options > Disconnect" in the context menu.

You can also connect the components with drag&drop. To do so, drag the component connector to the connector of the other component.

4.4.9 Exchanging welds and gaskets

You exchange welds and gaskets with drag&drop by dragging the new weld or gasket onto the existing one.

4.4.10 Creating generic pipe ends

There is a switch between circumferential ends and plug welds for pipes.

Bends have the "transition bend" connector form. Connector forms of the continuing pipes also have this value.

The length of the bending pipe and length of continuing pipes are added up for the bill of materials.

4.4.11 Editing component data

You enter component data either using the properties tree or in the properties in the "Attributes" tab.

4.4.12 Deleting a component

You can always modify or delete components on construction isometries. If you have generated spools from one of these isometries, the isometry is locked against further processing by default.

In this case, first delete the spool marks and confirm that the associated document should be deleted.

Procedure

To delete a component from a pipe run, proceed as follows:

1. Select the component.
2. There are two delete commands available in the context menu of the selection:
   - To only delete the component graphically, select the command "Delete".
     The object is still available in the Navigator along with its 3D information.
   - To delete the component as an object, select the command "Delete (Object)".
3. If the component was placed in a pipe run, select in the "Delete" window how to proceed with the gap that will be created.
   - "Close gap": The two pipe runs are connected.
   - "Fill the gap and close the following pipe": One of the two pipes is deleted.
   - "Do not fill the gap": The component is removed from the pipe run.
     If the pipe runs also exhibit different nominal diameters, you are prompted to select a process:
   - "Do not adjust nominal diameters"
   - "Adjust nominal diameters to smaller/larger nominal diameter"
     You can also specify that components are automatically replaced by components with matching nominal diameter.

Result

The component is deleted in the isometric report as well as the database regardless of the selected command. If you deleted the component from a pipe run, the result also depends on your selection in the "Delete" window.

4.5 Creating unit grids

If you want to use configurable grid axes on your isometric reports, you have to use a unit grid. You can create a unit grid for each plant, unit or subunit for which there is a "Coordinate system definition" tab. The axes of the unit will show through in the subunit and the ones of the plant will be visible in the unit. You can specify data on the coordinate system of the higher-level plant part in the "Delta distance" field in the "Coordinate system definition" tab under the "Zero point" control group.

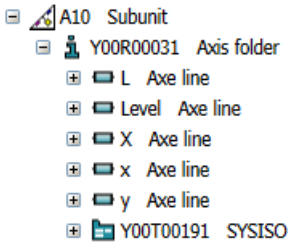
Procedure

1. Select the node in the "Units" tab below which you want to use a unit grid in the isometric report. You can choose from:
   - Plant
   - Unit
   - Subunit
2. Open the properties of the selected object.

3. Open the "Attributes > Coordinate system definition" tab.
4. Click the "Axis lines editor" button.
5. The "New axis" window opens.
   - To create a grid line with the X value, activate the option "X value" in the "New grid line" control group. Enter individual settings in the "Name" and "Position" fields, if necessary, and click the "Create" button.
     If you do not make individual settings, default settings are used.
   - To create a grid line with the Y value, activate the option "Y value" in the "New grid line" control group. Enter individual settings in the "Name" and "Position" fields, if necessary, and click the "Create" button.
   - To create a grid line with the level, activate the option "Level" in the "New grid line" control group. Enter individual settings in the "Name" and "Position" fields, if necessary, and click the "Create" button.
6. Save your entries.
7. To generate the unit grid, click the "Generate grid" button in the "Global position" tab.

Result

After using the axes on an isometry an "Axis folder" folder is created in the Navigator below the selected object (plant, unit or subunit).



You can use drag&drop to drag individual axes onto an isometric report. See also section Using grid axes (Page 67).

4.6 Position numbers

You have several options to influence the generation of position numbers.

4.6.1 Assigning position numbers from the engineering project

Overview

If you have specific requirements on the numbering of bills of materials and want to set corresponding references on the report, specify the generation of position numbers from your engineering project.

Example

Each similar pipe component is to receive a unique position number, regardless on how many reports the pipe component is used.

Technical implementation

An "Y00A00081 position number" attribute is assigned to each component. The range of numbers is handled with this attribute. See also chapter Edit with tag symbols (Page 115).

Procedure

1. Select the node in the "Units" tab of the Navigator in which the respective isometric report is located.
   You can select any number of nodes, even down to the root node for the entire project.
2. Select the "Set position numbers" command in the context menu.

Result

Position numbers are assigned to the components in the isometric report below the selected node according to the arrangement in the Navigator. The Navigator is also the hierarchical start object for assignment of position numbers. Any existing position numbers are retained.

Display on reports

To display position numbers on reports, you also have to create tag symbols on the reports. To do so, select the command "Tag symbols > Create > Position numbers" in the context menu on the user interface of an isometric report. See also chapter Creating tag symbols (Page 54).

This step can be automated by an administrator in the options script of a report template. See also chapter Calling and editing the options script of a report template (Page 105).

See also

Flags (Page 58)

4.6.2 Deleting position numbers in the engineering project

Target

If you want to delete many position numbers from isometric reports, you can do so in the Navigator. The respective reports do not have to be open for this purpose.

Example

You have created new reports after setting position numbers from the Navigator and want to assign new numbers for all reports after having deleted position numbers.

Procedure

1. Select a node in the "Units" tab of the Navigator in which the respective isometric reports are located.
   You can select any number of nodes, even down to the root node for the entire project.
2. Select the "Delete position numbers" command in the context menu.

Result

- The position numbers in reports are deleted below the selected node in the Navigator.
- Position numbers that you have assigned to the report with the context menu are deleted as well. Position flags are retained.
- If position numbers are preset on the base object of an engineering object, these are now inherited.

4.6.3 Creating tag symbols with position numbers on reports

You create graphic elements, such as flags, by creating tag symbols with the context menu of a report.

The value of the "Y00A00081 position number" attribute is also read out and position numbers are displayed in this way. A "Y00A00081 position number" attribute belongs to each component. The range of numbers is handled with this attribute.

The procedure for displaying position numbers is described in the explanation of tag symbols. See also chapter Creating tag symbols (Page 54).

4.6.4 Tag symbols with position numbers from base objects

If you have the right to access base objects, you can also drag tag symbols from the "Base objects" tab to your report with drag&drop.

These tag symbols are also called flags. They can contain freely configurable content, which means they can also display position numbers. Contact your administrator if you have questions about configuration.

The procedure for using flags is described in the explanation of flags. See also section Flags (Page 58).

4.7 Tag symbols on a report

4.7.1 Basics

You label components, welding points, and gaskets in the isometric report with numbers. The numbers are displayed in tag symbols. The contents of these tag symbols are read directly from the respective component properties.

You have the following options for automatic creation of tag symbols:

- Preconfigured tag symbols including position numbers for components, welding points and gaskets via a context menu on the isometric report. See also section Creating tag symbols (Page 54).
- If you have access to base objects, additional tag symbols (flags) are available. Your administrator can create additional tag symbols and display any required contents or evaluations contained in them. See also section Flags (Page 58).

4.7.2 Creating tag symbols

Properties of tag symbols

If different components exhibit the same properties, only one number is assigned per property. Tag symbols are not objects, which means they are not displayed in the Navigator. If the property in the component has no value, the tag symbol remains empty.

The following table shows the default display of tag symbols in the isometric report. The administrator can change or extend tag symbols.

Symbol	Description
	Position number
	Welding point
	Gasket

You can also enter texts with the toolbar of the isometric report and label components individually. Text fields added this way are not connected to the component.

- Flags that are created with the context menu are positioned so that they do not overlap any existing texts, if possible (including existing position flags).
- The flag content is updated when associated attributes are changed.
- An empty flag is displayed for deleted position numbers.

Requirement

- The isometric report is open.
- A pipe run is configured.

Procedure

To label components automatically with tag symbols, proceed as follows:

1. Specify which components you want to label:
   - If you want to label only one of several pipe runs, select the start or end component of the pipe run.
   - If you want to label all components on an isometric report, right-click an empty space in the report user interface.
2. Select one of the following commands in the context menu of the selection under "Tag symbols > Create":
   - "All tag symbols"
   - "Position numbers"
   - "Welding points"
   - "Gaskets"
   - "<report-dependent graphical text>"
     See also chapter "Options script of the report templates (Page 95)", section "Variable "GraphicalTexts"".

Result

The tag symbols are created. If there are position numbers, they are displayed in tag symbols.
See also chapter Assigning position numbers from the engineering project (Page 51).

If there are no position numbers, they are created for reports. If you use the other menu commands, "Position numbers", "Welding points" or "Gaskets, the "Value" is read from the "Y00A00081 position number" attribute. The value is specified with the options script for all other graphical texts.

Additional auxiliary symbols and tag symbols:

See also chapter Using flags (Page 58).

Different pipe specs: Displaying connection information

If two pipes with different pipe specs are used in an isometric report, you can add connection information in the form of a label.

1. Select one of the components at the junction.
2. Select the "Options > Display connection info > Yes" command in the context menu of the selection.

Sheet references

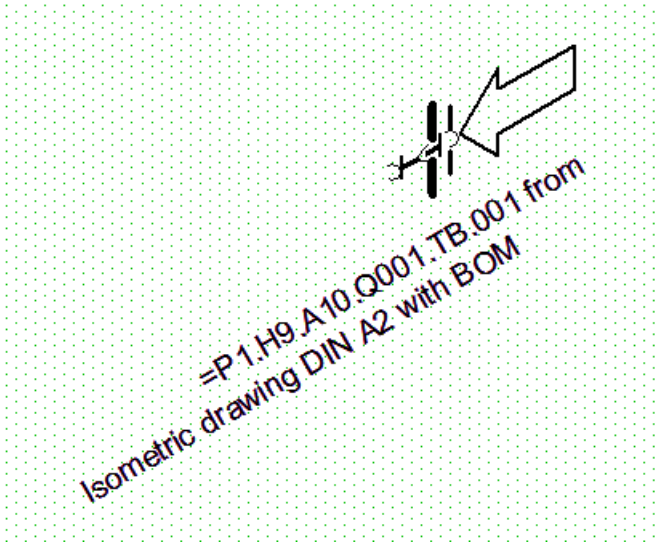
If the corresponding base data exist, sheet references are created automatically.

If a component is displayed on more than one isometries and if it is connected in each of them, the sheet references are created automatically. See also chapter Continuing a pipe run on another isometric report (Page 35). The display can depend on the flow direction.

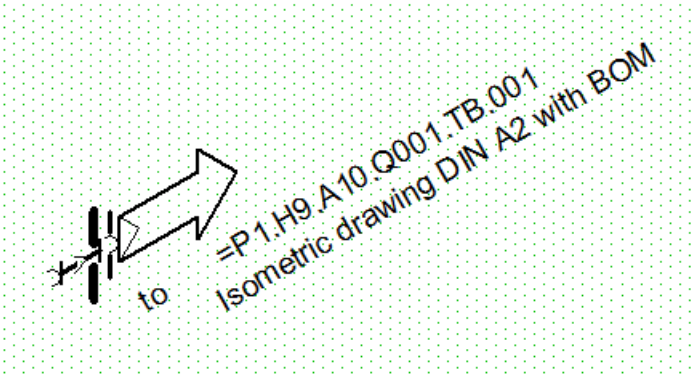
Configuration of sheet references:

"Standard tables > Y10 > M20 > A10 > Y10M20N00010 > Y10M20N00010A02 Sheet reference for pipes, multiple pages"

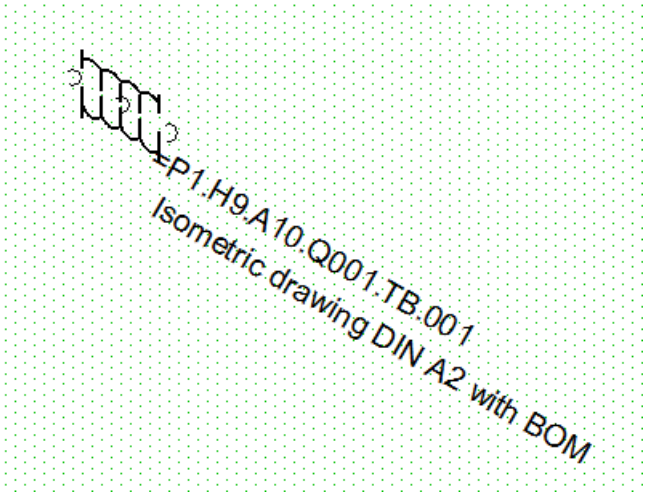
Sheet reference at inlet



Sheet reference at outlet



Sheet reference with neutral flow direction



If no additional component is connected on the other report, no sheet reference is created.

See also

Deleting a component (Page 49)

Editing slopes (Page 31)

Basics (Page 54)

Flags (Page 58)

4.7.3 Deleting tag symbols

Procedure

- To delete an individual tag symbol, select it and select the command "Delete" in the context menu.
- To delete tag symbols by category, select one of the following menu commands in the context menu of the selection under "Tag symbols > Delete":
  - "All tag symbols"
  - "Position numbers"
  - "Welding points"
  - "Gaskets"

Result

The respective tag symbols are deleted depending on the selected command. Deleting tag symbols does not affect numbering.

Even if numbers are no longer visible on the isometric report due to a lack of tag symbols, they are retained at the objects.

4.7.4 Flags

4.7.4.1 Basics

Flags are usually graphical elements on the working area that can be connected to a predefined string, such as position numbers, or can display evaluations.

Flags are administered as base objects in COMOS and can be placed on an isometric report. By placing a flag on an isometric report, no new engineering objects are created in the "Units" tab.

The following objects can only exist once for each component:

- Position flag
- Gasket
- Weld

They cannot be docked to another component.

4.7.4.2 Using flags

Requirement

You have the right to access base objects.

Setting flags

1. Open the "Base objects" tab of the Navigator.
   Select a prepared flag in the database:
   "@10 > A20 > A30 > A20 > A10 > A10 Isometries tag symbols"
   Flags may be but do not have to be assigned to specific component types, depending on their purpose. Contact your administrator if you have questions about configuration. See also chapter "ISO" tab: (Page 115).
2. Drag the flag to the required position in the isometric report using drag&drop.
   If the flag can be docked to a specific component, the component is highlighted in color.
3. Place the flag by clicking the required position.

Deleting flags

To delete flags, click <Del> after selection or select the "Delete" command in the context menu.

4.8 Dimensions

4.8.1 Basics

You use dimensions to specify the length of the pipe run including components. The complete dimensioning of the pipe run is the basis for final calculation of the isometric drawing.

The dimensioning of a pipe run includes the following:

- Start point and position points
- Length and space offset dimensions

Principle

The length of the drawn pipe run does not have an effect on dimensioning. Only the default values of dimensioning are used for calculation of the isometric drawing.

See also

Calculating the pipe run on a report (Page 75)

Dimensioning types (Page 59)

Creating dimensions (Page 61)

4.8.2 Dimensioning types

The dimensions are divided into the following types:

- Length dimensions
- Space offset dimensions
- Coordinate dimensions
- Distance dimensions

Dimensions are typed as follows:

- Type 1: Space offset dimensions
- Type 2: System length
- Type 3: Position dimensions
- Type 4: Length dimension of a component or component dimension
- Type 5: Length dimension of connections, such as welds or gaskets

The types are relevant for creating automatic dimensions for pipes and components. In the properties, you assign the types physical connectors in between which the length is dimensioned.

Length dimensions

Length dimensioning determines the length of pipes, components, and connections, such as welds or gaskets. You also use length dimensioning for position dimensioning: Position dimensioning determines the relative position of a component in a pipe.

Space offset dimensions

Space offset dimensioning determines the length and angles of space offsets. Space offset dimensioning draws in the catheti and angle used to create the space offset. Space offset dimensioning is of dimensioning type "1".

To clearly determine space offset dimensioning, the following values are required for a 2D space offset, for example:

- Length of catheti
  Or:
- Angle and length of a cathetus

Coordinate dimensions

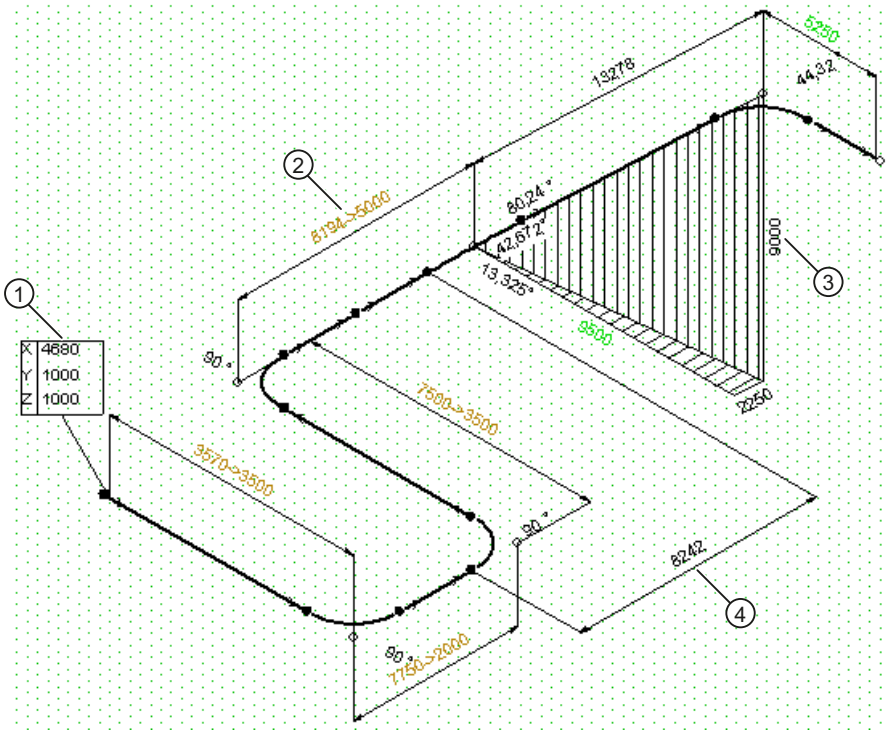
Coordinate dimensioning determines the position of the connectors at the components and pipes. The position coordinates are displayed in a coordinate box.

Distance dimensions

Distance dimensions determine the distance between two connectors. You specify the distance to be calculated based on the position of the two connectors.

Example

The following picture shows an isometric pipe run with default dimensions:



① Coordinate dimensions
② Length dimensions
③ Space offset dimensions
④ Distance dimensions

See also

Creating dimensions (Page 61)

Basics (Page 59)

4.8.3 Creating dimensions

You create dimensions in the layers of the isometric report.

You have the following options to create dimensions:

- Automatic dimensioning of pipe run
  See also section Automatic dimensioning of pipe run (Page 62).
- Individual dimensioning of a pipe run
  See also section Individual dimensioning of a pipe run (Page 63).

Dimension levels are assigned to specific drawing levels.

You show or hide levels with the "Levels" button.

See also

Dimensioning types (Page 59)

Basics (Page 59)

4.8.3.1 Automatic dimensioning of pipe run

Introduction

You create the basis for calculation of the isometry with automatic dimensioning of the main lengths and the position.

With automatic dimensioning of the pipe run, dimensioning lines are added and dimensioned with default values from the database. Adjust the values according to your drawing defaults. Depending on the complexity of the pipe run, you also adapt the alignment of the dimensioning lines.

The following dimensions exist:

- Dimensions of the main level (level 1)
  Creates dimensions for pipe lengths, space offsets as well as coordinate dimensions of the start and end point of the pipe run. The length dimensions are created in such a way that there are as few overlaps as possible with the pipes.
  The length dimensioning is aligned as follows by default:
  - The dimensioning line lies parallel to the pipe.
  - For pipe runs on the X/Y plane, the length dimensioning is aligned in the Z plane.
- Dimensions of the part level 2 (level 2)
  Creates position dimensions for components. A requirement for this is that you have created dimensions of the main layer and that there are components.
- Dimensions of the part level 3 (level 3)
  Creates dimensions for component lengths and connections, e.g., welds between pipes. A requirement for this is that you have created dimensions of the main layer.

Requirement

- The isometric report is open.
- Pipes and components are added.

Procedure

To create main dimensions, proceed as follows:

1. You have the following options to determine the pipe runs to be dimensioned:
   - To dimension all pipe runs on an isometric report, click a free space in the working area. All pipe runs are dimensioned.
   - To specify dimensioning for individual pipe runs, select one or more objects in the required pipe runs.
     Only the pipe runs with the selected objects are dimensioned.
2. To create main dimensions, select the command "Dimensions > Create > Main dimensions" in the context menu of the working area.
   The dimensions of the main layer are entered with default values.
3. If you also want to create part dimensions, select the command "Dimensions > Create > Part dimensions level 2" or "Dimenions > Create > Part dimensions level 3".

Result

Dimensions are added and assigned the default lengths.

Adjust these lengths according to your requirements. See also chapter Editing slopes (Page 31).

Note

Generating dimensions and labeling symbols at the same time

If you want to create main dimensions, part dimensions and labeling symbols for all pipe runs in the isometric report at the same time, click the "Create dimensioning and labeling" button in the toolbar.

See also

Creating dimensions (Page 61)

4.8.3.2 Individual dimensioning of a pipe run

Introduction

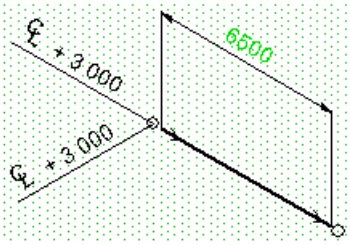
Use the isometry to measure individual dimensions in order to dimension the distances between any two connectors or to add a coordinate dimension. Individual dimensions are based on the same layer concept as automatic dimensions. This means you assign each individual dimension to a level.

Requirement

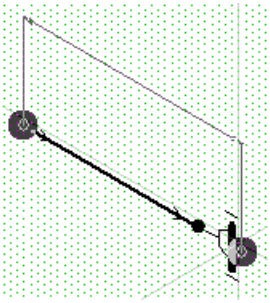
- The isometric report is open.
- A pipe run is configured.

Inserting dimension of the global height of a pipe axis (Centerline dimensioning)

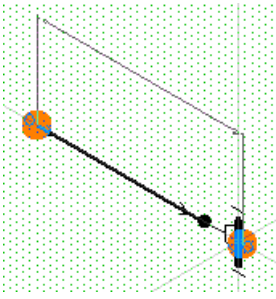
1. Click on a connector, press the <Ctrl> key and move the cursor to the pipe axis at the same time.
2. Click the left mouse button while pressing the <Ctrl> key.
3. The "Centerline" dimension line is displayed. The Z value differs from the value in the coordinate box: The value displayed is "Z + Y00T00279.Y00A04314" from the axis folder.



4. To dimension the distance between two points using logical connectors, click both connectors whose distance you want to dimension.

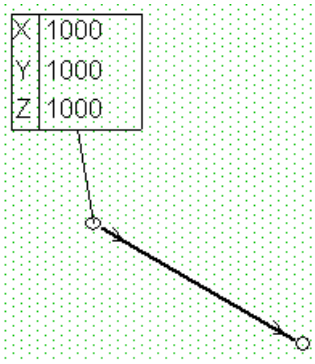


5. To dimension the distance between two points using physical connectors, press the <Ctrl> or <Shift> key and click both connectors whose distance you want to dimension.



Inserting coordinate dimensions

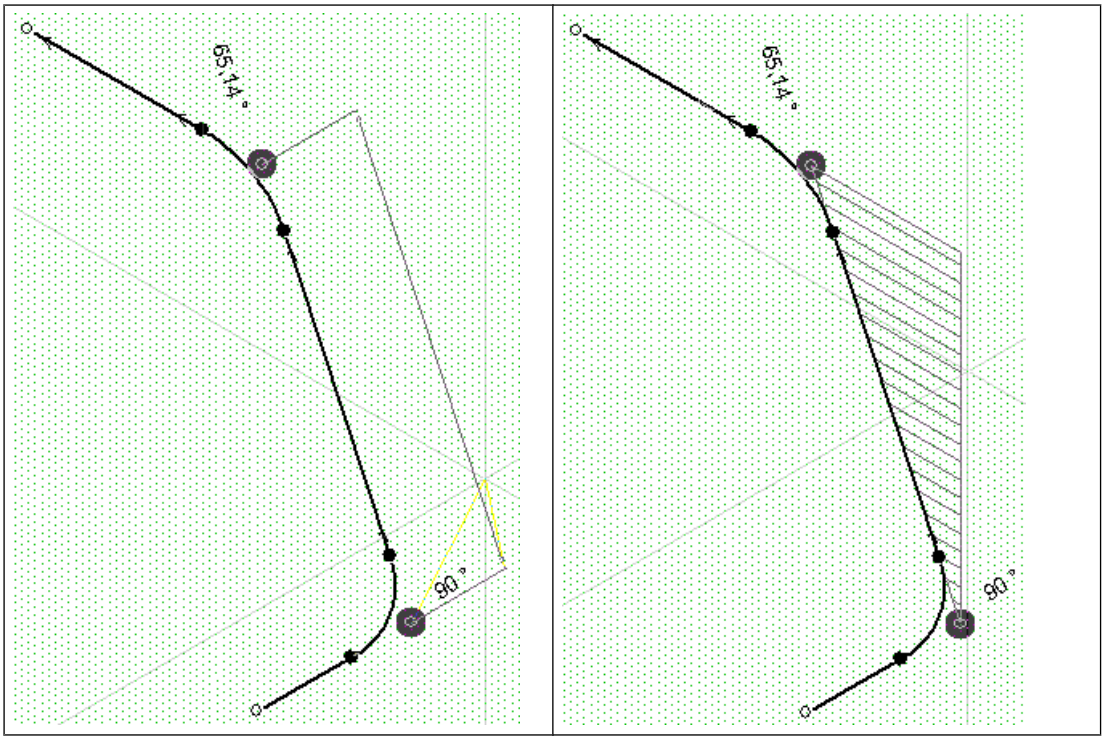
1. Click the "Dimensioning" tool.
2. Click on a connector in the isometry.
   The coordinate box is displayed at the cursor tip.
3. Click the position at which you want to insert the coordinate box and move the mouse. Place the coordinate box with another click. The coordinate reference system is identical to that of the subunit.



Space offset dimensioning

If you want to check if there is a special direction, click both connectors, press the <Ctrl> key and move the cursor.

You create a hatched triangle.

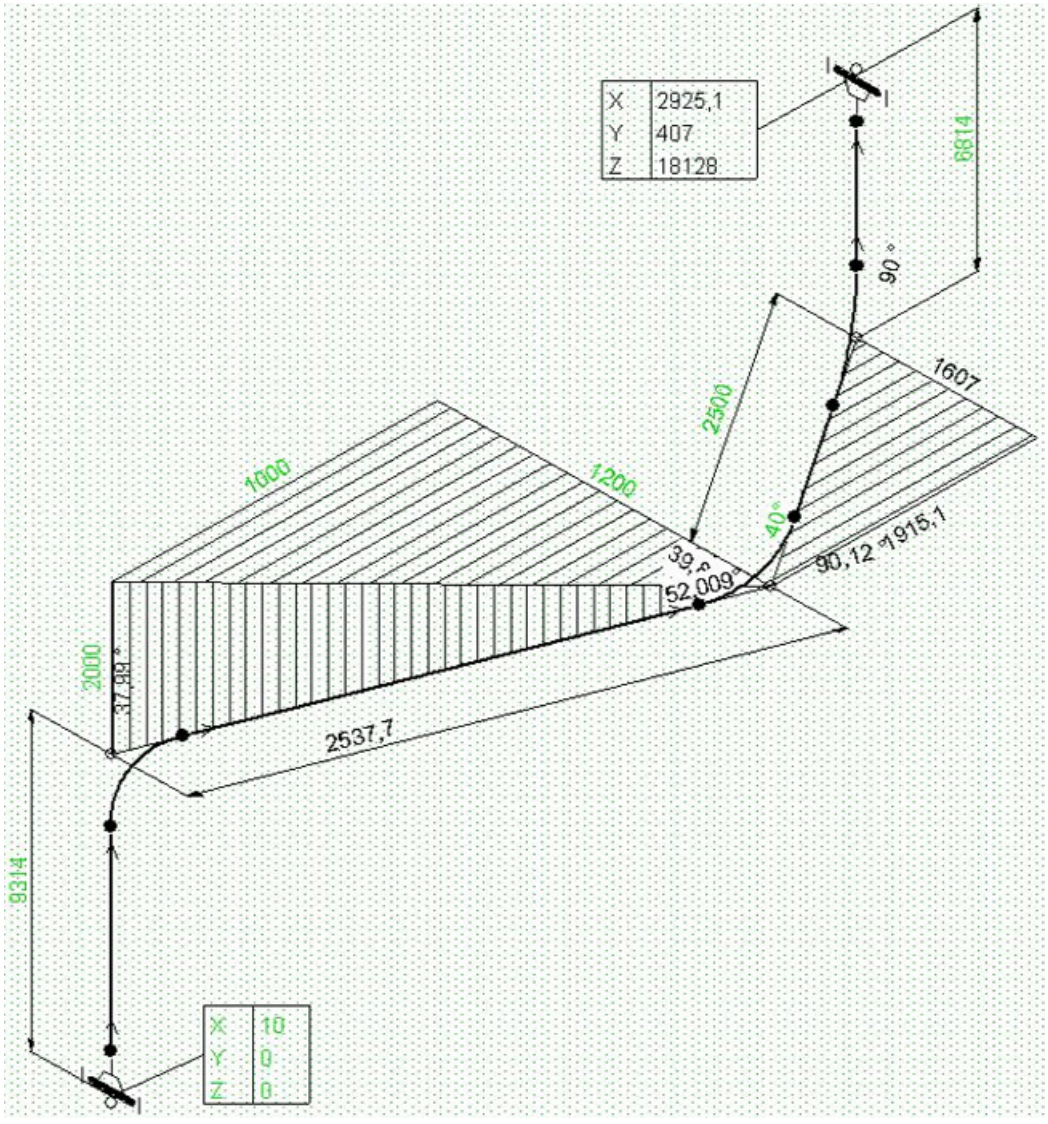


Space jump dimensioning

Pipes cannot only run at an angle in the plane, but also in space.

To display multi-axis dimensions, proceed as follows:

1. Click the "Dimensioning" button.
2. Select the start and end point of the required dimensioning.
3. Keep the <Ctrl> key pressed and move the mouse button until the display of the hatched triangles meets your requirements.
4. You place the triangles with a mouse click.



Distance dimensions

You use distance dimensioning to determine the distance between two connectors that are not on a main axis or special direction.

By default, the direction with the greatest distance on the isometric drawing is offered in distance dimensioning.

To create dimensions in other directions, activate the <Ctrl> or <Shift> keys. The position of the selected connectors determines if and which additional dimensioning directions are possible:

- <Ctrl>: Activates distance dimensioning in the direction of the second largest distance between dimensioning points.
- <Ctrl+Shift>: Activates distance dimensioning in the direction of the shortest distance between dimensioning points.

Result

The dimensioning is added. They have the default dimensions. Adjust the dimensions according to your needs so that you can calculate a valid isometric drawing. See also section Adapting dimensions (Page 71).

See also

Creating dimensions (Page 61)

Connectors (Page 25)

4.8.3.3 Creating slope dimensions

Requirement

You have drawn a slope

Procedure

1. Select the dimensioning of the pipe run.
2. Select the "Options > Show slope" command in the context menu of the selection. The slope is shown at the dimension.
3. Click the "Create dimensioning and labeling" button.

4.8.3.4 Using grid axes

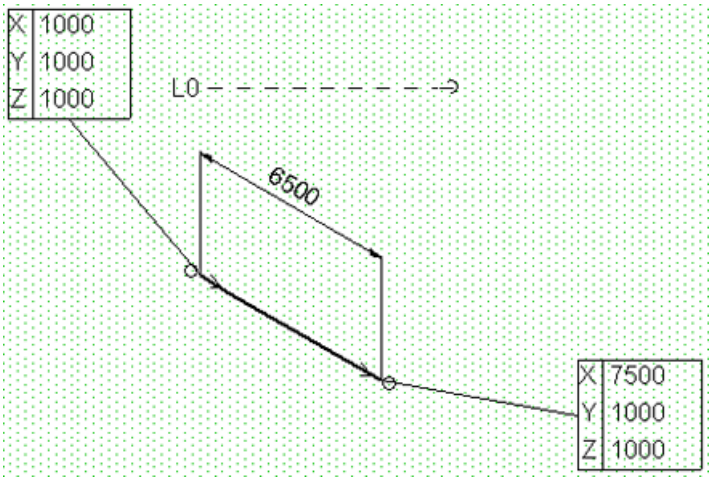
With grid axes on isometric reports you can add reference lines of your choice, name them and dimension them.

Requirement

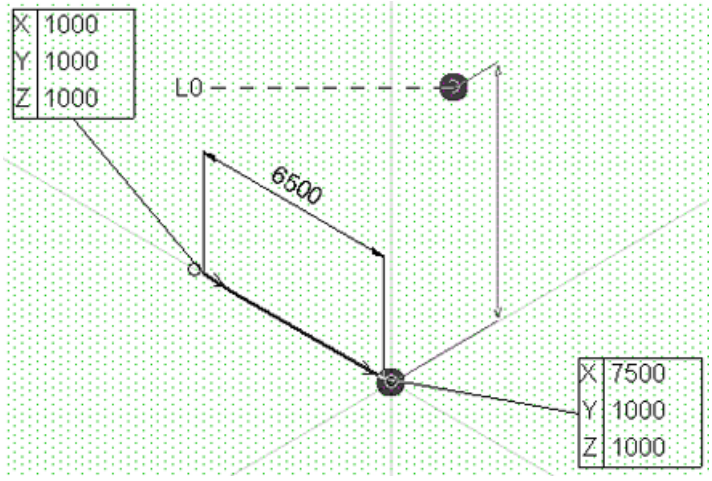
- You have a unit grid. See also chapter Creating unit grids (Page 50).
  After creation of a unit grid, there is a folder "Y00R00031 Axes folder" in the "Units" tab of the Navigator. It is located below a plant, a unit or a subunit.
- The isometric report is open.

Procedure

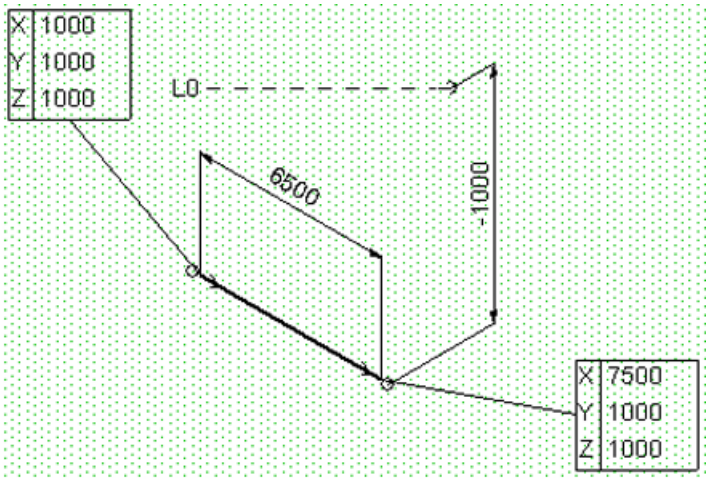
1. From the "Axes folder" in the "Units" tab of the Navigator drag the required grid axis onto your isometric report.



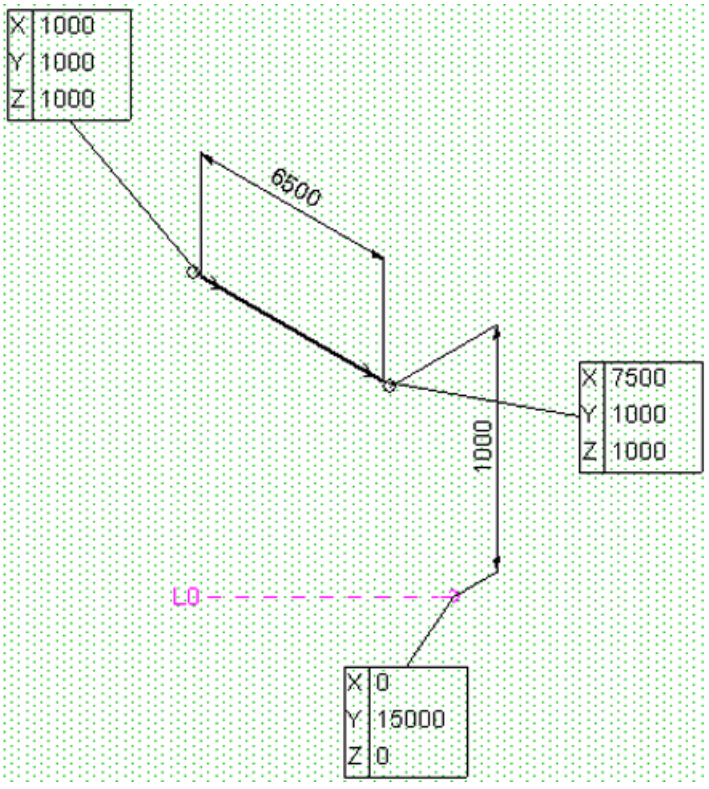
2. To display a reference dimension to the axis, click the "Dimensioning" button to dimension distances between any two connectors.
3. Connect the connector of an axis to the connector on the required pipe with your mouse.



Depending on the selected axis, the dimension is displayed as distance in the plane (X, Y axis) or distance of the height (Z axis).
If the dimension value is displayed as negative value, the display on the isometry does not match the actual arrangement in space but is shown reversed:



In such cases, select the axis and move it across the axis of the pipe to the other page:

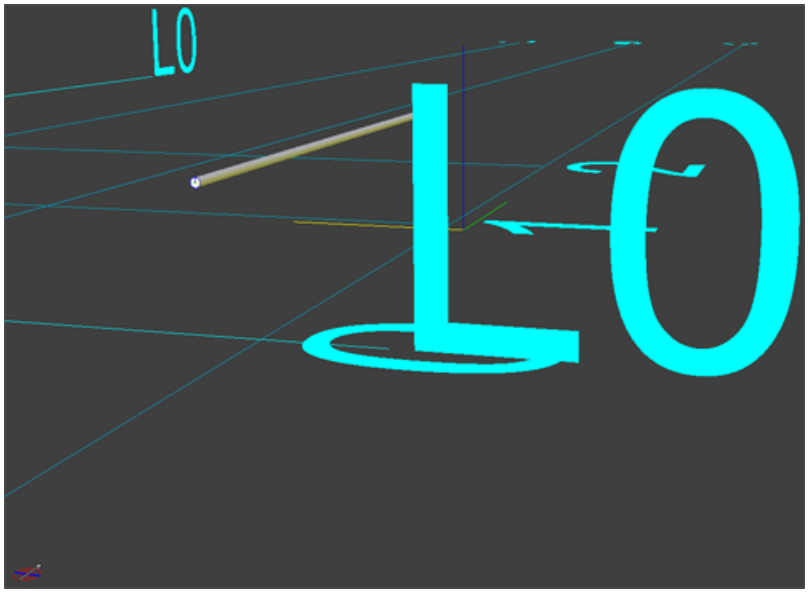


The dimension value is now displayed as a positive value.

Result

You have added a grid axis to your isometric report.

Display after clicking the "3D read-only" button:



4.8.4 Adapting dimensions

4.8.4.1 Basics

You adapt dimensions as follows:

- Changing values
  When you adapt the values of lengths or angles, the changes have an effect on the calculation of the isometric drawing.
- Change the alignment of the dimensions
  When you change the alignment of dimensions, the changes only have an effect on the isometric report.
- Deleting dimensions
  When you delete a dimension, you can no longer calculate the isometry.

You can only enter decimal places for length and angle dimensions. By default, two decimal places are displayed. The decimal places are only displayed in the toolbar in the respective edit field. The value is rounded to be displayed as an integer on the dimensioning line.

4.8.4.2 Editing dimensions

Requirements

- The isometric report is open.
- The pipe run is dimensioned.

Editing dimension values

You change dimension values in the toolbar. You change the values for lengths or angle depending on the selected object.

1. Select the dimension.
2. Enter the required value(s) in the toolbar.
3. Click the ! button on the toolbar to apply the changed values.

The changed values are checked for consistency and entered on the dimensioning line. If a dimension borders the selected dimension on the same level, it is automatically selected. To skip the dimension, click the ">" symbol in the toolbar.

Adapting dimensions with the same slope

To adapt dimensions with the same slope, click the slope pipe and make an entry in the "XY" or "L" field.

When you enter values here, the hypotenuse or the ancathetus is calculated.

"XY" stands for the ancathetus and represents the projected pipe length.

"L" stands for the hypotenuse and represents the actual pipe length.

Change the alignment of the dimensions

The following table shows the different handles you can use to change the alignment of a dimension with drag&drop:

	Rotates the dimensioning line around the axis of the dimensioning object. Optionally changes the distance between dimensioning line and dimensioning object.
	Rotates the dimensioning line and all components of the main dimension with their part dimensions around the axis of the dimensioning object.
	Changes the position of the dimension value. Optionally changes the distance between dimensioning line and dimensioning object.
	Changes the distance between dimensioning line and dimensioning object.

To change the alignment of the dimensioning, follow these steps:

1. Click the dimensioning line twice.
   The grab points are displayed.
2. Perform the required action.
3. To exit alignment, click a free space in the working area.

The alignment of the dimension is changed. The alignment of components can also change depending on the configuration of components.

Deleting dimensions

1. Select the dimension.
2. Select "Delete" in the context menu of the selection.
3. To delete all dimensions or the dimensions of a level, select the respective command in the context menu of the working area under "Dimensions > Delete".

One or more dimensions are deleted depending on the selected command. To calculate the isometry, set the dimensions once again.

Setting adjusting lengths

1. Select the length dimension of a component or component leg.
2. Enter the desired value in the "L" field of the toolbar.
3. Click the ! button on the toolbar to apply the value.
   The adjusting length is entered in the dimension.
   The adjusting length is adapted to the pipe length in the parts list.

Viewing or changing available adjusting lengths

To view or change available adjusting lengths of a component, you can select the required component on the isometric report and navigate to the object with the context menu. In the object properties, change to the "Attributes > Production" tab in which the available adjusting lengths are administered.

4.8.4.3 Changing the type of slope input

The slope is displayed as percentage, degree or ratio information on an isometric report. The default setting is percentage. You can change the type of slope input for each report.

Procedure

1. Select the required isometric report in the "Units" tab of the Navigator.
2. Select the "Properties" command in the context menu.
3. Open the "Attributes > SYSISO" tab.
4. Click the respective selection button in the "Gradient input type" field and select the required new display. The settings "Percentage", "Degree" or "Ratio" are available.
5. Save your entries.

Result

The selected changes are applied to the selected report.

4.8.5 Locking or releasing 3D objects

To protect your coordinate calculations in the form of 3D objects, lock assigned 3D objects on an isometric report. This also protects your calculated isometric drawings, which evaluate 3D objects.

- Any changes you make to the isometric display at a later time have no effect on locked 3D objects.
- You can release locked 3D objects.

Procedure

To lock 3D objects, proceed as follows:

1. Select the required objects on the isometric report.
2. In the context menu, select the "Options > 3D object status > Locked" command. Locked objects are displayed in blue on the report.

To release 3D objects, proceed as follows:

1. Select the required objects on the isometric report.
2. In the context menu, select the "Options > 3D object status > Released" command. Released objects are displayed without blue on the report.

4.8.6 Calculating isometries

4.8.6.1 Overview of isometries that can be calculated

You can calculate the following isometries:

- The isometry of a selected pipe run, regardless of additional separate pipe runs on an open isometric report.
- The isometry of all pipe runs on an open isometric report.
- The isometry when using multi-page/connected isometric reports.

A 3D consistency check is performed before calculation of the isometry. All affected connectors are checked for their relative position and direction and corrected, if necessary.

The isometry is calculated after successful consistency check. The calculated values are displayed at the dimensions.

4.8.6.2 Calculating the pipe run on a report

Requirement

- An isometric report is open.
- The dimensions are added.
  - The pipe run has a starting coordinate.
  - All pipes have length dimensions.
  - All space offsets are clearly identified.
  - All variable component length are dimensioned.
- The dimensions have plausible values.

Procedure

- In order to calculate all pipe runs on the open isometric report, click "Calculate" without having selected a component or pipe run beforehand.
- In order to calculate a specific pipe run on the open isometric report, select any component of the pipe run in question. Click the "Calculate" button in the toolbar of the isometric report.

Result

The isometry of the pipe run connected to the selected component is calculated on the open isometric report.

The pipe run is checked for consistency. See also chapter Consistency check (Page 76).

See also

Basics (Page 59)

Dimensioning types (Page 59)

Creating dimensions (Page 61)

4.8.6.3 Calculating pipe runs on multi-page/connected reports

Requirement

See also section Calculating the pipe run on a report (Page 75).

Procedure

1. Select a component that has been placed multiple times.
2. Select the command "Options > Calculate connected documents" in the context menu.
   The "Calculate documents" window opens. See also section "Calculate documents" window (Page 136)
   The isometry of the pipe run connected to the selected component is calculated on the open isometric report.
   The calculation is continued on other connected documents starting with the selected component.
   - If there are no errors, the calculated objects are saved in the connected documents. A success message is output and shown in the "Protocol" area.
   - If errors occur, the affected document is shown and the note "Error" is written on the document in the "Calculate documents" window.
     The document is opened.
3. Correct any errors and click "Continue". To refresh the "Calculate documents" window, start a new calculation at the same start object on the start document.
4. To ignore the error and continue with the calculation on the next document, click "Ignore".
5. To interrupt the calculation, click "Pause". Everything that could be calculated up to this point is saved. To resume the calculation, click "Continue".
   When you save the start document, you are also saving a list of all documents calculated up to this point.
6. Click on the "Finished" button once the calculation is successfully completed.

Result

The isometry of the pipe run connected to the selected component is calculated on the open isometric report.

The pipe run is checked for consistency. See also section Consistency check (Page 76).

4.8.6.4 Consistency check

If the consistency check was not successful, the respective components are shown in orange.

The following errors are typical:

- The part dimensioning of a component is missing (under-dimensioning).
- Dimensions are contradictory (over-dimensioning).

The isometry is calculated after successful consistency check. The calculated values are displayed at the dimensions.

- Green:
  Correct default values
- Black:
  From values without default.
- Brown:
  Contradictory default values that could not be considered in the calculation.

4.8.6.5 Special form gap dimensioning

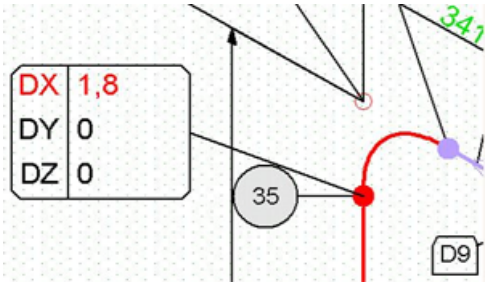
If two components are not connected to each other correctly, the incorrect connection is identified as inconsistent in red in the isometric drawing. In this case, the isometry cannot be calculated.

You can set a tolerance for gaps so that an inconsistency only arises from a certain gap size. See also section Stipulating the tolerance for gaps (Page 101).

Procedure

If you still want to calculate the isometry, proceed as follows:

1. Click the "Dimensioning" button.
2. Select the problematic connection.
3. You define a permitted gap by placing the coordinate box with pressed <Shift> key.
   You can then calculate the isometry but the gag remains in 3D.



4.8.7 Displaying calculated documents

When you display the calculated documents, you receive an overview of all documents that are connected to the document at which a calculation of the documents was executed.

Requirement

You have executed the "Calculate connected documents" command for the isometric drawing.

Procedure

1. Select an isometric drawing in the Navigator on which you have executed the "Calculate connected documents" command.
2. Select the "Show calculated documents" command in the context command.
   A window opens in which all connected isometric drawings are listed.
3. To navigate to a document, select the "Navigate" command in the context menu.

4. To release 3D objects of an isometric drawing that were locked by the calculation, select the command "3D Objects > Release 3D objects" in the context menu.
   The 3D objects can be adapted during a calculation.
5. To lock 3D objects on an isometric drawing, select the command "3D Objects > Lock 3D objects" in the context menu.
   The 3D objects can no longer be adapted during a calculation.

See also

Locking or releasing 3D objects (Page 74)

4.9 Spool isometric drawings

4.9.1 Basics of spool isometries

Introduction

Spool isometric drawings divide an isometric pipe engineering into individual production units. You define production units by setting spool marks in the isometric pipe engineering. You can create a production report for each spool isometric drawing. The production report uses the templates you are using for isometric pipe engineering.

Application

You use spool isometric drawings, for example, to hand over individual production units of an entire pipe engineering to a service provider for construction.

Editing options

You use a spool isometric drawing exclusively for displaying individual production units of a configured isometric pipe engineering. If you set spool marks, this production unit is blocked for editing. You are then limited to the following editing options:

- Adapting adjusting length of placed components.
- Adding labeling and graphics.
- Adapting tag symbols.

But you can release individual objects of a production unit. You can also adapt the pipe spec and the nominal diameter for released objects.

You should still change data relevant for engineering only in the isometric report even if they can be released. Then reset the spool marks.

4.9.2 Spool marks

Introduction

You use spool marks to define the start and end of a production unit. You create spool isometric drawing based on spool marks in a separate step.

Setting spool marks

Spool marks are set automatically on the basis of predefined rules. The rules determine which components are considered "stop components". You use a pipe or a component as starting point. Then, a search is performed for the first "stop component" in all available directions.

The following rules apply to the search of stop components:

- If the starting point is already defined as stop component, the search is canceled. Spool marks are set at the physical connectors.
- If a stop component is not found in a search direction, the spool mark is automatically set at the beginning or the end of the pipe run.

If you are dragging flags from the Navigator to the report with drag&drop, automatic spool functions are not supported and no spool document is created.

Moving spool marks

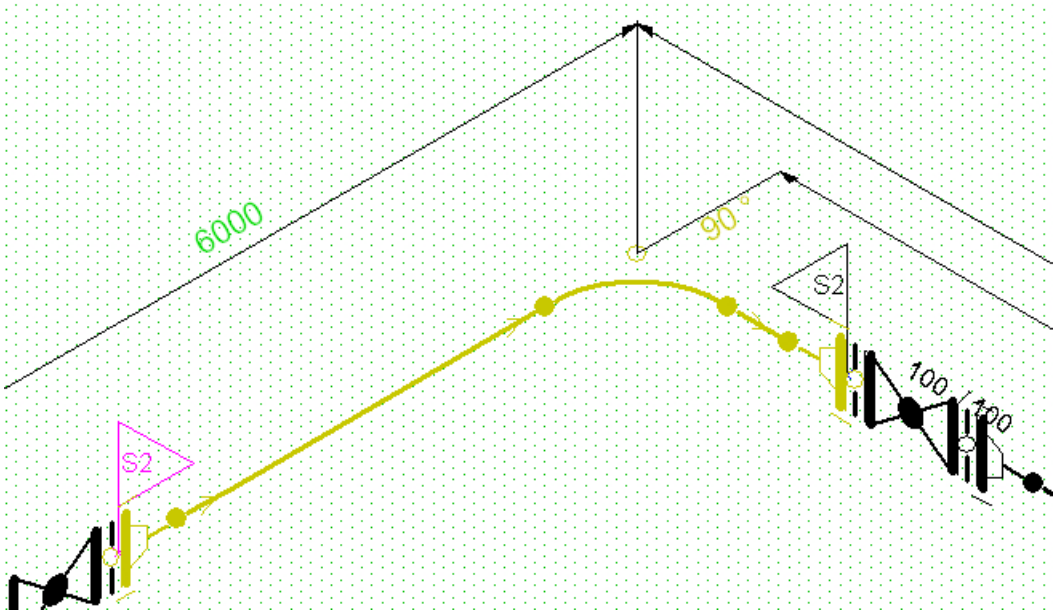
If there is no spool document, you can move the automatically set spool marks using drag&drop.

The system then prevents open ends being set within production units or spool marks being set outside the limits.

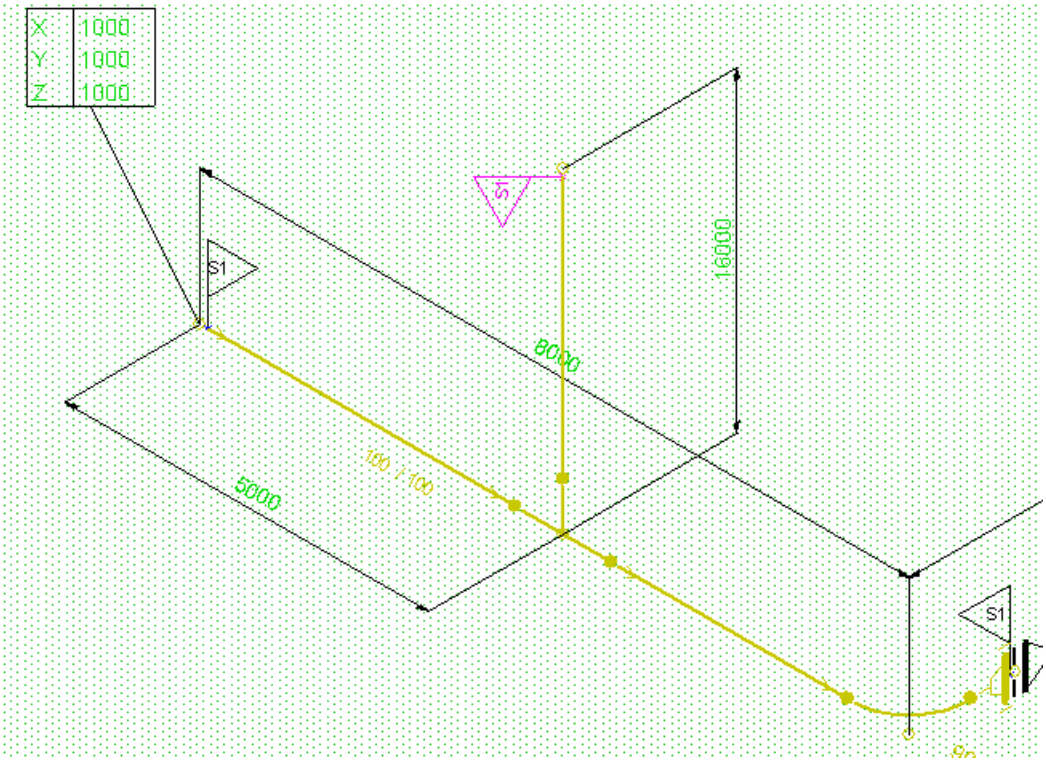
Objects outside a spool can still be edited.

Example

The following figure shows automatically set spool marks. Starting point is the highlighted pipe run. Flanges are defined as "stop component". You can extend the production unit to the right with the 2-way valve. You cannot integrate the 3-way valve into this production unit because it has an additional connector. No spool mark has been created for this connector because a "stop component" (flange) was reached beforehand:



The following figure shows automatically set spool marks on the same pipe run. Starting point is the highlighted 3-way valve. Because flanges were defined as "stop components" and the open, third connector is considered the "end of the pipe run", only the 3-way valve is defined as production unit. Yon can extend the production unit to the right and the left because no open ends are created at any location:



See also

Creating spool isometric drawings (Page 82)

Basics of spool isometries (Page 78)

Connection objects (Page 81)

4.9.3 Connection objects

Introduction

When you create a spool, transitions are created between the spool and the remaining isometric drawing. The transitions are usually welds or gaskets. These transitions are referred to as "connection objects" in COMOS.

Connection objects and bills of materials

When you set spool marks, the created connection objects are by default assigned to the bill of material of the isometric report. You can change this assignment in the context menu of a spool mark.

You specify for connection objects in which bill of material they are to be listed:

- Bill of materials of the isometric report
- Bill of materials of the production report

This assignment ensures that objects are not contained twice in part lists.

See also

Editing spool marks (Page 83)

4.9.4 Creating spool isometric drawings

Introduction

You can create a separate production report for each spool isometric drawing. The production report uses the template you are have used for the isometric report.

Requirements

- The isometric report is open.
- The pipe run is dimensioned and calculated.

Procedure

To create a spool isometric drawing, proceed as follows:

1. Select a pipe run or a component that is to be included in the spool isometric drawing.
2. Select the "Spool > Set spool marks" command in the context menu of the selection. The spool marks are set after confirmation. The area is locked for editing.
3. To extend an area framed by spool marks, move the spool marks with drag&drop, if necessary.
4. Select the spool mark at the beginning or the end.
5. Select the "Spool > Create document" command in the context menu of the selection.

Result

The production report is created below the isometric report in the Navigator. The spool isometric drawing is created and centered in this production report.

The production report includes the following in addition to the spool isometric drawing:

- All components and pipes within the spool marks.
- All dimensions.
- Tag symbols for position numbers.

Releasing objects within a spool

1. Select the required objects.
2. Select the "Options > Release" command in the context menu of the selection.

You can also change the pipe spec and the nominal diameter for released objects.

See also

Basics of spool isometries (Page 78)

Spool marks (Page 79)

Editing spool marks (Page 83)

Structure of an isometric report (Page 17)

4.9.5 Editing spool marks

Requirement

The isometric report is open.

Changing spool mark properties

You change the name and the description in the properties of a spool mark. You can also define a spool mark as "bending spool". In this case, the length of the spool is also calculated and displayed in meters in the bill of material.

The changes in the properties of a spool mark also have an effect on all spool marks of the spool.

1. Select the spool mark.
2. Select the "Spool > Properties" command in the context menu of the selection.
   The "Spool properties" window opens.
3. Make the required changes.

Adding a connection object to the spool

1. Select the spool mark whose component you want to add to the spool.
2. Select the "Spool <Component> add to spool" command in the context menu of the selection.

If the isometric report contains a bill of material, the component is removed from the bill of material. The component is added to the bill of material of the corresponding production report.

Removing a connection object from the spool

To remove a connection object from the spool, proceed as follows:

1. Select the spool mark whose component you want to remove from the spool.
2. Select the "Spool <Component> remove from spool" command in the context menu of the selection.

The component is removed from the bill of material of the production document. If the isometric report contains a bill of material, the component is added to the bill of material.

Highlighting a spool

1. Select the spool mark.
2. Select the "Spool > Highlight" command in the context menu of the selection.

The spool is highlighted in color. The highlighting of the spool remains in place until the next action in the isometric report.

Deleting spool marks

When you delete a spool mark, the entire spool is removed.

1. Select the spool mark.
2. Select the "Spool > Delete" command in the context menu of the selection.
3. Confirm the following message

The spool document is being removed.

The pipe section is released for processing.

See also

Basics of spool isometries (Page 78)

Creating spool isometric drawings (Page 82)

Connection objects (Page 81)

4.10 Compilation isometric drawings

4.10.1 Basics of compilation isometries

Purpose

Compilation isometric drawings are used to display and print several isometric drawings in a common report. Compilation isometric drawings are also referred to as overview or construction isometric drawings.

You can use compilation isometric drawings, for example, for heating circuit isometric drawings, calculation isometric drawings or various loop isometric drawings, such as print loops or press loops.

Application

You have distributed an isometric pipe engineering on several isometric reports for editing. To print out the pipe engineering completely, you place the isometric report on a compilation report.

Rules for compilations

The following rules apply to compilations:

- The alignments of the isometric report have to match the alignments in the compilation.
- You can place an isometric report in different compilations.
- You cannot place an isometric report multiple times in a compilation.
- You can place a compilation into another compilation but not break it apart.

See also

Creating a compilation isometric drawing (Page 85)

Placing an isometric report in a compilation isometric drawing (Page 86)

Breaking apart an isometric report in a compilation isometric drawing (Page 87)

Editing compilation isometric drawings (Page 88)

4.10.2 Creating a compilation isometric drawing

Introduction

A compilation report based on the same report template as an isometry. If a compilation report contains a bill of material, all pipes and components included in the placed isometric reports are listed in it.

This means the objects are always placed several times, at least once in the compilation isometric drawing and once in the individual isometric drawing.

The compilation isometries are configured by default in such a way that the dimensions and labels are not adapted when you break apart placed isometric reports.

Procedure

1. Create an isometry as usual. See also section Creating and opening an isometric report (Page 16).
2. Open the "Attributes > SYSISO" tab in the properties of the isometry.
3. Select the "Compilation" entry from the "Isometry document type" list:

See also

Basics of compilation isometries (Page 84)

4.10.3 Placing an isometric report in a compilation isometric drawing

Requirements

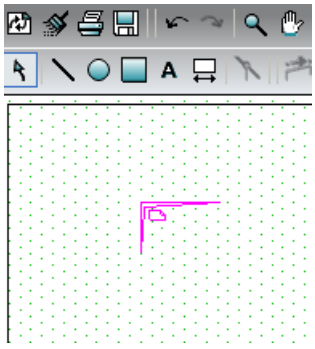
- The "Units" tab is open in the Navigator.
- Individual isometric reports are configured.
- A compilation report has been created.
- The planar north arrow must point in the same direction for all involved isometric reports.

Procedure

1. Open the compilation document.
2. Use drag&drop to move the isometric report from the Navigator to the compilation document.

Result

The isometric report is placed as graybox in the compilation document:



If the compilation document includes a bill of material, it is updated.

Any changes to the source documents at a later time do not have an effect on the isometric drawings placed in a compilation document.

See also

Basics of compilation isometries (Page 84)

Editing compilation isometric drawings (Page 88)

Breaking apart an isometric report in a compilation isometric drawing (Page 87)

4.10.4 Breaking apart an isometric report in a compilation isometric drawing

Target

If you want to display related isometric drawings on a compilation report in connected form or create dimensions or tag symbols on the placed isometric drawings at a later time, you need to break apart the grayboxes of the placed isometric drawings.

Only the graphical information of the pipes is applied in the compilation document.

Requirement

- The compilation report is open.
- The required isometric reports are placed on the compilation report.

Procedure

To break apart one or more grayboxes of the placed isometric drawings on a compilation document, proceed as follows:

1. Select the grayboxes of the isometric reports you want to break apart.
2. Select the "Break apart placed document" command in the context menu of the selection. This command transfers the graphical information of the pipes from the graybox to the overview isometric drawing and then deletes the graybox.

Result

- The selected grayboxes are broken apart and placed in the center of the compilation report.
- If the corresponding COMOS connectors exist, related components are automatically connected to each other on the report. The objects are placed side-by-side in this case.
- If you have continued the pipe run in the broken apart isometric reports on additional reports, the pipe run is displayed without references on other reports.
- Graphical tools, such as lines, compression, etc. can still be used.

Exceptions

The following are not transferred into the compilation report:

- Elements placed multiple times beforehand that are not relevant for the material list.
- Flags
- Dimensions

See also

Editing compilation isometric drawings (Page 88)

Basics of compilation isometries (Page 84)

Changing the bill of material relevance of an object (Page 48)

4.10.5 Editing compilation isometric drawings

Requirement

Grayboxes are broken apart. See also chapter Breaking apart an isometric report in a compilation isometric drawing (Page 87).

Editing options

If you have placed isometric reports in an assembly isometry, the following editing options are available to you:

- Adding dimensions and labeling.
  The "Create dimensioning and labeling" button is enabled in the toolbar.
- Display 3D preview of placed isometric reports.
- Changing the graphical display of the compilation isometric drawing. Changes in a compilation do not have an effect on the individual isometric reports (sources).

Retro-modification of components

To change pipes or components within a pipe run on a compilation report, delete the respective isometric report in the compilation report.

Change the individual isometric report and place it once again in the compilation report.

Identification of individual isometric reports

To identify and open individual isometric reports, select an object of your choice in the compilation report and select the "Navigate > Documents" command in the context menu. Those isometric reports which include the same object are listed for selection.

See also

Creating a compilation isometric drawing (Page 85)

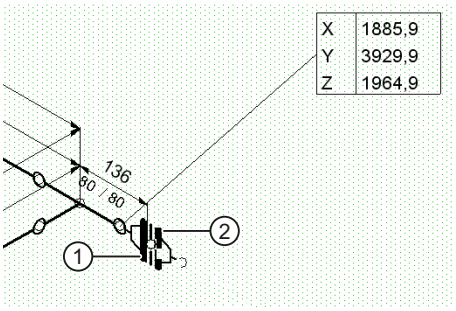
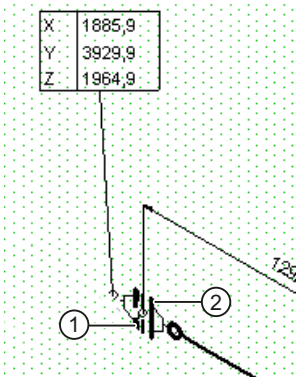
Placing an isometric report in a compilation isometric drawing (Page 86)

4.11 Sheet separation

Sheets are automatically separated if isometric drawings are created from existing 3D data with the "3D -> ISO" command. If there is a sheet separation, the administrator determines in the report template which type is applied for the isometric sheets. See also section Options script of the report templates (Page 95).

Several isometric sheets may be created depending on the number of components on the 3D display. Separation preferably takes place between fixed components, such as a valve and a flange. If a practical separation between the minimum and maximum number of objects specified in the option script is not possible, the separation takes place between pipes.

If sheet separation took place, the respective connection components are displayed with dotted lines and there will be a reference to their counterpart on the other isometric drawing.

Isometric drawing A	Isometric drawing B
	
1: Flange placed on isometric drawing A. 2: Flange placed on isometric drawing B and only depicted on isometric drawing A. Contains a reference to the flange on isometric drawing B.	1: Flange placed on isometric drawing A and only depicted on isometric drawing B. Contains a reference to the flange on isometric drawing A. 2: Flange placed on isometric drawing B.

4.12 Using bills of materials

4.12.1 Basics

The bill of material is a list in an evaluating report which lists all pipe parts and fastening parts included in a pipe. The bill of material sorts the components according to type. If you label components as "not bill of material relevant", these components are not listed in the bill of material. See also chapter Changing the bill of material relevance of an object (Page 48).

The report templates "TB_A30" and "TB_A40" are preconfigured in such a way that a bill of materials is automatically added on the isometry.

The structure of the bill of material may vary in its details depending on the way the administrator has configured the report template. See also chapter Changing standards of isometric reports (Page 101).

4.12.2 Calculating bolts

The administrator may define bolt sets using the COMOS PipeSpec Designer module. COMOS Isometrics accesses the stored data and calculates the required bolt material for the connection. You can find more information on this topic in the "PipeSpec Designer" manual, keyword "Preparing bolt sets".

You can extend the bill of material on an isometric report with information on bolt sets (lengths, dimension and type).

Requirement

An isometric report with bill of material is open.

Procedure

Click the "Options > Bolts > Calculate bolts" command in the context menu of the open isometric report.

Result

COMOS searches for the bolt sets with the corresponding bolt parts. An object is created on the engineering page below one of the connected pipe parts or below the bolt set group for each required bolt set. The bolt parts are linked with the pipe parts to which they are connected. They are displayed on bills of material.

The status display in the Navigator is updated during calculation of bolts or nuts. See also section Configuring the status display for bolt calculation (Page 126).

4.12.3 Updating the bill of material

You can update a bill of material as follows:

- The bill of material is automatically updated when you open a bill of material or an isometric report with an integrated bill of material.
- Click the "Reevaluate document" button on the toolbar of an open isometric report.

Check the entries for correct information for your own sake.

Information on bolts or nuts in a bill of material is not updated by clicking the "Reevaluate document" button and should be recalculated after changes have been made. See also section Calculating bolts (Page 90).

Creating an isometric drawing from existing 3D data 5

You can use existing 3D data on your isometric report.

Requirements

- The "Units" index card is open in the Navigator.
- Components with 3D data are located below a "Pipe" object.

Procedure

- To apply all components found under a "Pipe" object in an empty isometric report, proceed as follows:
  - Right-click the user interface of the isometric report.
  - Select the "Options > 3D > Iso" command in the context menu.
    COMOS searches the Navigator structure below the respective pipe for existing components with 3D data and displays them on the isometric report.
- To apply a specific 3D object from the Navigator to an empty isometric report, proceed as follows:
  - Use drag&drop to move the planned component to the report.
  - Select the object on the isometric report.
  - Select the "Options > 3D > Iso" command in the context menu.
    If you only applied the components connected to a component, the search for "stop components" takes place in all possible connection directions. These are usually vessel nozzles and pumps. All components including the pipe components between starting and stop component are applied to the isometry. If the starting component is itself a "stop component", it is ignored.
- To extend a component on an isometric report with components that are already connected to it in 3D, proceed as follows:
  - Select the respective component on the isometric report that has a free connector.
  - Select the "Options > 3D > Iso" command in the context menu.
    All components still connected in 3D are applied to the isometry.
    If a component is already displayed on another isometric report with connected reports, only another isometric symbol is created on the open report after selecting the "Options > 3D -> Iso" command.

Administration 6

6.1 Isometric reports

6.1.1 General

By default, the following report templates are offered for isometric reports in base projects on the "Documents" tab:

- "A30 > T > TB TB_A10 Isometric drawing DIN A2"
- "A30 > T > TB TB_A10 Isometric drawing DIN A3"
- "A30 > T > TB TB_A10 Isometric drawing DIN A2 with BOM"
- "A30 > T > TB TB_A10 Isometric drawing DIN A3 with BOM"

6.1.2 Options script of the report templates

The following variables are available in the options script:

Variable "Application"

Code	`Application = "ISO"`
Type	String
Function	Sets the basic types of report templates to "ISO". Activates the iso-specific basic functionality.

Variable "AutoMarkAsChanged"

Default Code	`AutoMarkAsChanged = False`
Type	Boolean
Function	Specifies whether revision rectangles are also to be displayed without revision monitoring.

You can find additional information on this topic in the "COMOS Platform Administration" manual, keyword "Revision monitoring".

Variable "BOMCompressionKeys"

Code	See below.
Type	String
Function	Specifies compression keys for the Bill of Material entries.

You can find more information on this topic in the "COMOS Platform Administration" manual, keyword "Revision monitoring for reports".

Components that have identical values in the attributes that are defined here are summarized in the Bill of Material in one line.

- Compression keys pipe spec, nominal diameter, and description
  Code:
  ```
  Dim BOMCompressionKeys(3)
  BOMCompressionKeys(0) = "Y00T00011.Y00A00052|VALUE" ' Pipe class
  BOMCompressionKeys(1) = "Y00T00011.Y00A00003AA01|VALUE" '
  NominalPipeSize1
  BOMCompressionKeys(2) = "Y00T00067.Y00A00047|MEMO"
  PositionNrAutoOn = TRUE
  ```
  All components that match in the compression key get the same position number.
- Compression key position numbers
  Code:
  ```
  Dim BOMCompressionKeys(3)
  BOMCompressionKeys(2) = "Y00T00002.Y00A00081|VALUE"
  PositionNrAutoOn = False
  ```
  Since the position numbers themselves function as compression keys, the position numbers may not be assigned automatically. Thus the position numbers that were set manually for the compression keys are evaluated.

See section "Variable "PositionNrAutoOn".

The option script setting can be overwritten by the settings for the base objects for tag symbols. Components can be excluded from compression via the base objects of the following tag symbols.

- "A10 gasket"
- "A60 position number"
- "A70 weld"

See also chapters Edit with tag symbols (Page 115) and Base objects of the tag symbols (Page 121).

Variable "DimensionDigitsForDistance"

Default Code	`DimensionDigitsForDistance = 4.0`
Type	Long
Function	Specifies the number of decimal places for the dimensions.

The value that was input for length "L" in the toolbar for the dimension can be input with as many decimal places as desired. However, a value that has been rounded off to a specific number of decimal places is actually displayed for the dimension.

See also chapter Creating dimensions (Page 61).

The display of angles at space offsets is rounded off with up to three decimal places. This value is specified by the system.

Variable "DimensionTextHeight"

Default Code	`DimensionTextHeight = 4.0`
Type	Double
Function	Specifies the text height of the dimension values.

Variable "DimIsoHeight"

Default Code	`DimIsoHeight = 5`
Type	Long
Function	Distance of the main dimension from the pipe

This option specifies the factor by which the text height is multiplied in order to determine the height of the main dimension (dimensioning line). It corresponds to the distance of the main dimension from the pipe with automatic dimensioning.

Variable "DimLevels"

Code	`Dim DimLevels(3)`
	`DimLevels(0)="Y00T00002.Y00A00077AA0" & "|" &"1;2"`
	`DimLevels(1)="Y00T00002.Y00A00077AA0" & "|" &"3"`
	`DimLevels(2)="Y00T00002.Y00A00077AA0" & "|" &"4;5"`

Type	String
Function	Mapping of dimension types to dimension levels

Evaluated when dimensions are generated automatically (context menu of the isometry: "Dimensions > Create > ...").

The dimension types that were defined in COMOS are assigned to the dimension levels.

- Main dimension level 1 (`DimLevels(0)`):
  Type 1: Space offset dimension, type 2: Length dimension pipe
- Part dimension level 2 (`DimLevels(1)`):
  Type 3: Position dimensions
- Part dimension level 3 (`DimLevels(2)`):
  Type 4: Component length
  Type 5: Length dimension connections

Attributes are defined at the components in which root points can be entered for the dimension types. The script specifies to attribute to be evaluated when the context menu for the corresponding dimension level is called.

The following attributes are used:

- "Y00T00002.Y00A00077AA01 dimensioning type 1
- "Y00T00002.Y00A00077AA02 dimensioning type 2
- "Y00T00002.Y00A00077AA03 dimensioning type 3
- "Y00T00002.Y00A00077AA04 dimensioning type 4
- "Y00T00002.Y00A00077AA05 dimensioning type 5

Variable "GraficalTexts"

Code	`Dim GraficalTexts(3)`
	
	`GraficalTexts(0) = "Y00T00002.Y00A02620" & "|" & Document.CObject.Spec("Y00T00190.Y00A00062").Displayvalue & "|" & "Type=Position"`
	`GraficalTexts(1) = "Y00T00002.Y00A02620" & "|" & Document.CObject.Spec("Y00T00190.Y00A00062AA02").Displayvalue & "|" & "CON=1;Type=Weld"`
	`GraficalTexts(2) = "Y00T00002.Y00A02620" & "|" & Document.CObject.Spec("Y00T00190.Y00A00062AA03").Displayvalue & "|" & "CON=1;Type=Gasket"`

Type	String
Function	Controls the mouse context menu and the associated functionality concerning the tag symbols.

In detail, the following is specified by the variable:

- Which tag symbols are offered in the mouse context menu of the isometry.
- When one of the mouse menus is called: Which attribute is evaluated at the component to determine the base object of the tag symbol.
- Which identification key the report element of the tag symbol is given internally within the system.

The script passes the following detailed information to COMOS:

- `Document.Spec("Y00T00190.Y00A00062").Displayvalue`:
  The display value that was input at the base object of the isometry in the specified attribute is offered in the mouse context menu. Must be configured accordingly.
- `Y00T00002.Y00A02620`:
  If a context menu is called, COMOS checks for each engineering object placed on the isometry whether there is an attribute with the same name (type: Reference) in the "system information" tab.
  The attribute must include a link to the base object of the desired tag symbol. A DocObject of the tag symbol is created and its symbol script is executed.
- `[identification key]=[value]`:
  System identification that specifies the created DocObject in more detail. For position numbers: DocObject of type "`Position`".
  If separate report templates are to be used for construction isometries and spools then GraphicalTexts should be commented out in the spool template.

Variable "IsoEnabled"

Default Code	`IsoEnabled= TRUE`
Type	Boolean
Function	Switches the crosshair cursor saved at the cursor from 90 degree angles to 30 degree ISO angles.

Variable "MainDimensionAutoOff"

Default Code	`MainDimensionAutoOff = TRUE`
Type	Boolean
Function	Suppresses the automatic creation of dimensions of level 1 when drawing.

If `False`:

The dimension types that were assigned to dimension level 1 are also created automatically when the pipe is drawn

In the database: the length of the pipe, space offset dimensioning, coordinates boxes.

Variable "PositionNrAutoOn"

Default Code	`PositionNrAutoOn = TRUE`
Type	Boolean
Function	Automatically generates position numbers.

Evaluated when the following tag symbols are generated on the construction isometry via the mouse context menu (reads all the position numbers of its component):

- "A10 gasket"
- "A60 position number"
- "A70 weld"

If separate report templates are to be used for construction isometries and spools, PositionNrAutoOn should have the value `FALSE` in the template for spools. Otherwise, under certain circumstances, the position numbers that were generated before by the spool document are overwritten by the construction isometry.

The position numbers are also taken into the Bill of Material as long as there are still such tag symbols on the isometry.

- `PositionNrAutoOn = TRUE`:
  Position numbers are generated automatically for all components that do not have any tag symbols yet. If a component already has a position number, then it is overwritten.
  The numbers are written to the following attribute in the engineering object:
  SYSISO.Y00A00081 position number.
  If the compression key is active, components with the same key get the same position number. See variable "BOMCompressionKeys".
- `PositionNrAutoOn = FALSE`:
  The position numbers are assigned by the user (depending on the nominal diameter or the same number for all nominal diameters).

Variable "PositionDefaultStartNr"

Default Code	`PositionDefaultStartNr = 1`
Type	Long
Function	Start number for the automatic assignment of position numbers

This is the same for all number ranges (tag symbols for position numbers).

Variable "PositionDefaultStep"

Default Code	`PositionDefaultStep = 1`
Type	Long
Function	Step increment for the automatic assignment of position numbers

This is the same for all number ranges (tag symbols for position numbers).

Variable "PositionIsMissingAliasNr"

Default Code	`PositionIsMissingAliasNr = -999`
Type	Long
Function	Numbers that are input in the Bill of Material if a position number does not exist yet for a component. (E.g. -999)

Variable "SymbolRotationByDimension"

Default Code	`SymbolRotationByDimension = TRUE`
Type	Boolean
Function	Changes the symbol alignment synchronously with the dimension alignment.

Starting point:

Mark a dimension by two single left-clicks. Its grab points are then made visible. At the same time the dimensions and components that you have encompassed are marked.
`SymbolRotationByDimension = TRUE`

If you change the alignment of the selected dimension by means of its round green grab points then the alignment of all symbols for which the following apply is changed:

- The symbols are encompassed by the selected dimension (components and dimensions).
- The symbols have the same alignment as that of the selected dimension.

You can also rotate only the dimension that had originally been selected using the round grey grab point.

`SymbolRotationByDimension = FALSE`

The enclosed components do not rotate as well.

Variable "SymbolType"

Code	`SymbolType = "M25_P1"`
Type	String
Function	Specifies the diagram type for the symbols.

Variable "MinCountDevices"

Code	`MinCountDevices = <Number>`
Type	Integer
Function	Determine the minimum number of components on an isometry. Influences sheet separation.

Variable "RecommendedCountDevices"

Code	`MinCountDevices = <Number>`
Type	Integer
Function	Determine the target number of components on an isometry. Influences Sheet separation Example: 10

Variable "MaxCountDevices"

Code	`MinCountDevices = <Number>`
Type	Integer
Function	Determine the target number of components on an isometry. Influences sheet separation.

6.1.3 Stipulating the tolerance for gaps

Set a tolerance for gaps so that connections with gaps are only deemed inconsistent from a certain gap size.

Procedure

1. Open the "SYSISO" tab in the properties of the report template for isometries.
2. Enter a value in the "Gap tolerance" field.
3. Confirm your entries.

6.2 Changing standards of isometric reports

6.2.1 Purpose

Base object: "@30 > M00 > A80 > A10 > T > TB > A10 Isometric drawing"

Report template: ".A30 > T > TB > TB_A10 > TB_A10 Isometric drawing DIN A2""

Changing isometric reports in individual cases

To make a change to a single report, select an isometric report in the "Units" tab that meets your requirements.

Changing an isometric report template graphically

To make only graphical changes to an isometric report template for many cases, copy an existing report template from the "Templates" tab and modify it. With this approach, a north arrow and a bill of material may already be included depending on the original template.

Changing fundamental behavior of isometric reports

To change the standard behavior of an isometric report and, for example, change attributes on the base data page for this purpose, create a copy of the respective base object in the "Base objects" tab and make your changes.

6.2.2 Checking the "Attributes > Spools" tab

Procedure

1. In the "Units" tab of the Navigator, navigate from the required report object to the report template in the "Documents" tab in the base object (SO1 by default).
2. Open the properties of the report template.
   You can view the settings for creating the spool isometric drawing in the "Attributes > Spools" tab and make any necessary changes.

Changing the graphic symbol for the beginning or end of a production section

1. Click the "Navigate, Properties" button next to the "Spool mark" field.
2. Select the "Navigate > Object" command.
3. You can change the graphical symbol in the properties of the "A10 Spool mark" object. See also section Checking the "Symbols" tab (Page 103).

Changing the labeling symbols at components

Enter the desired attributes in the "Tag symbols that are automatically displayed when spool documents creation is marked" field.

Applying dimensions from original drawings

1. Enter the desired level of the original document in the "Dimensioning levels for spool documents" field.
   Permitted entries are "1", "2" or "3".
   Dimensions are not applied if the field is empty.
2. Save your entries.

6.2.3 "Spools" tab

Field	Description
"Spool flag"	Reference to the base object for spool marks. Queried if the command "Spool > Set spool marks" is called in the context menu of the isometric drawing.
"Dimensioning levels for spool documents"	Default: Empty string; only the system level is drawn. Possible values: Numeric; divided by commas.
"Tag symbols that are automatically drawn upon the creation of spool documents"	Defines the flag symbols that are automatically generated upon creating spool documents. Default: Empty string; all tag symbols that are also available in the context menu of the isometry. Possible values: Object names of tag symbols as a string, separated by a comma Example: "A10,A20"

6.2.4 Checking the "Symbols" tab

Procedure

1. If you want to change or view the symbol for the beginning or the end of a production segment, open the properties of the following object in the "Base objects" tab of the Navigator:
   "@10 > A20 > A30 > A20| > A10 > A20 > A10 Spool marks"
2. Open the "Symbols" tab.
3. Change the graphical symbol shown in the "Symbol" column, if necessary.

6.2.5 Checking the "Attributes > ISO" tab

Open the properties of the following object in the "Base objects" tab of the Navigator:

"@10 > A20 > A30 > A20| > A10 > A20 > A10 Spool marks"

In the "Attributes > ISO" tab you can see or change how a spool document is created and how spool marks behave.

Inherited properties

- "Isometric display" option:
  When you select this option, set flags are isometrically distorted. This option is deactivated by default.
- "Consider compression" option
  When you select this option, similar components are compiled in a bill of material position. This option is activated by default.
- "Allow drag & drop" field
  When you select the entry "Yes" from the list, you can drag the object "Spool mark" with drag&drop onto the isometry.

Settings for spool marks

- "Function code area" field:
  You can specify an area in this field in which spool marks may be created. Spool marks may be set on flanges only, for example, as shown in this figure.
- "Range of known connection objects" field:
  The function code area for welds and gaskets is entered in this area by default.

6.2.6 Checking the "SYSISO" tab

Procedure

1. In the "Units" tab of the Navigator, navigate from the required report object to the report template in the "Documents" tab in the base object (SO1 by default).
2. Open the properties of the report template.
   You can view the settings of report properties specific for isometric drawings in the "Attributes > SYSISO" tab and make any necessary changes.
3. Select the type from the "Isometry document type" list:
   - "Normal" for a standard isometric report
   - "Spool" for a spool isometric report
   - "Compilation" for a compilation isometric report
4. Select the angle of the north arrow in the "Angle of rotation for north point" list:
   - "30°" (default setting, north arrow points to top right in isometric report)
   - "150°"
   - "210°"
   - "330°"

   Do not change the rotation angle in running projects! Isometric reports with different rotation angles cannot be placed in one compilation isometric report.

5. Select the graphic version of the north arrow in the "North arrow" list.
6. Select the presentation type for the pipe slope from the "Gradient input type" list.

6.2.7 Calling and editing the options script of a report template

You can set variables, control attributes and extend your own script lines in the script of the report template.

Procedure

To edit the script of the report template, proceed as follows:

1. Double-click the required report template in the "Documents" tab of the Navigator.
   The report user interface opens.
2. You have the following options to open the "Options" window with the shown script:
   - Right-click the report user interface and select the "Options" command in the context menu.
   - You can also double-click the report user interface.

Main contents of the script

```
UIProgID = "ComosWspRUI.WspRUI"
Application = "ISO"
'XXDocProgID = "ComosXXDocIso.XXDocIso"
```

A user-specific DLL file is loaded upon activation; this file must be available.

```
KeepScriptRunning = TRUE
IsoEnabled= True
SymbolType = "ISO"
```

The behavior is specified as isometric report from the script start until this point.

See also

Options script of the report templates (Page 95)

6.2.8 Changing symbol bar for specific isometric drawings

The symbol bar is structured according to pipe part types. Under the following node, the base object icon bar of the isometric drawing is managed. The base object icon bar is used to determine the preferred piping components and to place piping components on the isometric drawing.

"@30 > M23 > A10 > Y30 > A20 Pipe part types"

6.2.8.1 Adding a component type to the document-specific symbol bar

Procedure

You can find more information on this topic in the "PipeSpec Designer" manual, section "Create a pipe part type".

Result

A new symbol for the new pipe part type is displayed in the symbol bar on the isometric drawing.

This provides access to all components which are included in the selected pipe spec and whose function code corresponds with the name of the new pipe part type.

Example

Function code of the pipe part type for elbows:

- 21XXX

Function code for concrete elbows:

- 21300 Angle elbow
- 21400 Segment elbow
- 21500 Return elbow...

6.3 Managing standard tables

6.3.1 Basic information on creating standard tables

Overview

The standard tables have the following functions in Isometrics, among others:

- Contain dimensions of components and pipes
- Contain specifications for positioning in the 2D and 3D representation
- Manage special features and their characteristics

You can find additional information on this topic in the "COMOS Platform Administration" manual, keyword "Standard tables" or in the "PipeSpec Designer" manual, keyword "Managing parameters".

Note

Data inconsistencies if parameters are not managed correctly

Not managing parameters correctly can lead to data inconsistencies in the database and take up significant additional time and effort.

Seek the assistance of or instructions from your account manager in particular if you wish to amend the following parameters:

- Nominal diameters
- Nominal pressure
- Types of gasket
- Connection types
- Function codes

Administration of standard tables

To create new standard tables or edit existing ones, select the command "Administrator > Base data > Standard tables" in the COMOS menu.

You can find additional information on this topic in the "COMOS Platform Administration" manual, keyword "Administration of selection lists".

6.4 Managing the 3D data

All components that are placed on an isometry must be 3D-capable.

The pipe part catalog meets this requirement. Therefore, all components placed on an isometry in the database derive from the pipe parts catalog.

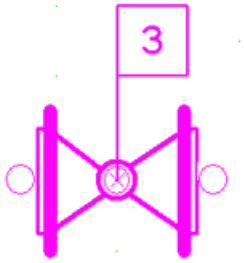
You can expand the pipe part catalog or create a separate catalog. You can find more information on this topic in the "PipeSpec Designer" manual, keyword "Expanding pipe parts catalog".

6.5 Creating symbols

6.5.1 Symbols and subsymbols

The graphic symbol that you see when you place a component on the isometric drawing is composed of several components.

Example: Valve symbol:



First of all, the component symbol itself. Subsymbols are added to this symbol:

- Symbol for contact faces and connection types
  You use the component properties to define which symbols are displayed: Using the "Symbols" tab as well as attributes in the "3D geometry" tab and "Connection <1-n>" that are provided with standard tables in which scripts for graphic symbols are defined.
  Where these sub symbols will be added to the component symbol is controlled by the component base object – by defining connectors for the sub symbols.
- Optional: tag symbols, e.g. position numbers
- Optional: other text variables, e.g. `*V*P E:Z` (insertion point for the valve drive).

6.5.2 Anchor points for tag symbols

A *V*P text with the label SX must be placed in the symbol for the component.

Example: *V*P SX:Y00T00002.FR001*

The *V*P text determines the following:

- from which attribute the reference to the base object of the tag symbol is read (for position numbers, e.g. "System information" tab, "Position number" attribute. A reference must be set there to the tag symbol for the position number).
- where the tag symbol will be added to the component symbol (the insertion point is identical to the point of origin of the text variable).

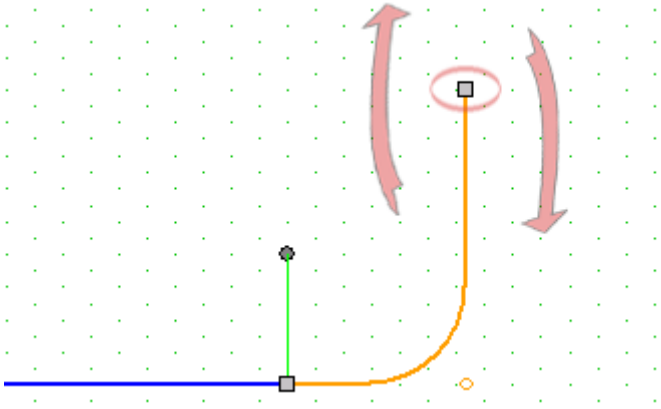
If a `*V*P` text with corresponding key is missing, the tag symbol is automatically placed in the center of the symbol.

6.5.3 Connectors

Connectors are defined in the component symbol. The connectors determine where other symbols will be added – for example, the symbols of connection types and the contact faces, but also of dimensions.

The following types of connectors are available:

- Logical connectors: `CX[Number]`
  Purpose: Correspond to the connectors in the engineering objects. Each logical connection of a symbol on a report thus has a counterpart in the engineering object. Logical connectors are required when data is to be transferred from one object to another object.
  As a rule, the components that are placed on the isometric drawing do not themselves define logical connectors. Instead, they define placeholder connectors. The symbols inserted at these placeholder connectors then define logical connectors. See also section Example configuration (Page 110).
  Exception: All components (except pipes) must have a logical connector for the point of origin (`CX0`). `CXO` has no direct equivalent in the Navigator and is used when setting graphic symbols, e.g. when setting a dimension.
- Physical connectors: `CP[Number]`
  Purpose: To define connectors for graphic report elements (e.g. dimensions). They exist only on the report, not in the Navigator.
  Please note: Pipes cannot be connected to CP connectors.
  They are addressed on the isometric drawing with <Ctrl> or <Shift> key.
- Placeholder connectors: `C#[Number]`
  These connectors are needed to add additional symbols to the symbol of the component, e.g. symbols for connection types and contact forms.
  If a `C#` connector on the isometric drawing is not connected, the user can move it with its grab point. Through that, pipes and elbows can be extended from their unconnected end:



- Dynamic connectors: `CY[Number]`
  Only for pipes. If a branch is implemented on a pipe, for example, the connectors are created dynamically. They are not predefined in the symbol of the base object.
  Dynamic connectors can be created as often as possible. The dynamic connectors are created in the database as logical connectors.
- Connectors for fastening parts: `CM[number]`:
  Connectors via which the fastening parts are connected with each other, independently of the pipe connections.

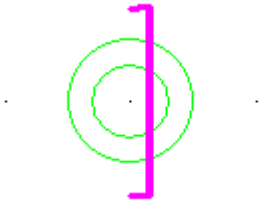
6.5.4 Example configuration

General

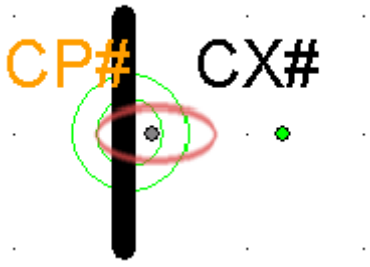
All symbols (component, contact face, connection type) may be drawn in non-rotated display (0°).

Procedure

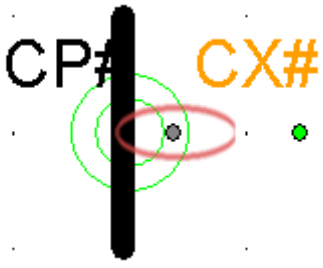
1. Configure the base object of the tag symbol.
2. Configure the standard table for contact faces.
   For DIN/EN: "Standard tables > Y10 > M20 > Y30 > A10 > 1 > A60 Contact faces"
   Scripts for symbols of contact faces are stored in the standard table. The contact face contains purely graphical information.
   If necessary, select the row of the required contact face and customize its symbol for the "M25_P1" drawing type.
   Example: Contact face `Form C`:



3. Configuring the standard table for connection types
   "Standard tables > Y10 > M23 > A10 > Y10M23N00004 Connection types"
   Scripts for symbols of connection types are stored in the standard table.
   If necessary, select the row of the required connection type and customize its symbol for the "M25_P1" drawing type.
   Example: Flanged end connection type:
   Point of origin of `CP#`



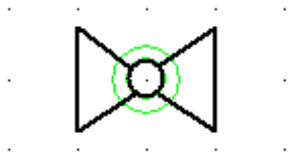
Point of origin of `CX#`



The flanged end has two connectors:

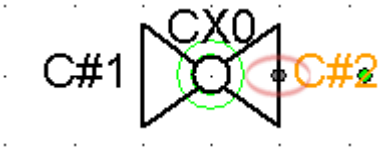
- `CP#`: Connector point for dimensions
- `CX#`: Connector point for the other component, which is defined through the connector table and is created when placing the primary component.

4. Base object of the shut-off valve: Configuring symbol:
   Base object of the shut-off valve, "Symbols" tab: Open the Symbol Designer and design the component symbol for drawing type "M25_P1":

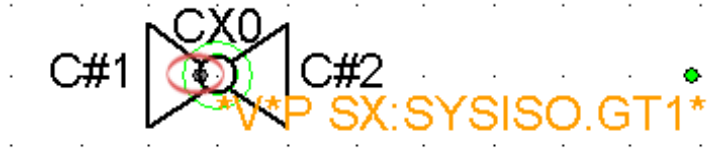


   Add logical connector for the point of origin:

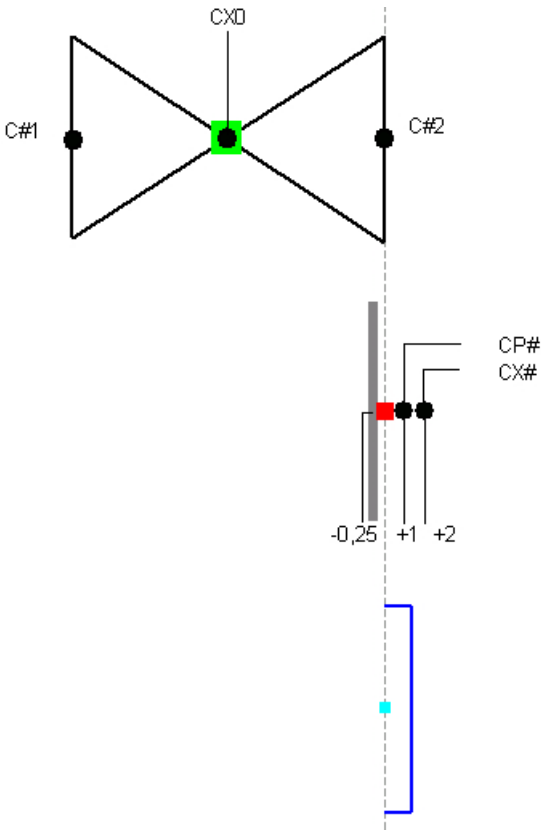
   Add connectors at which the connection types and contact faces will be inserted:



   Highlighted in figure: The insertion point (point of origin) of the symbol added via placeholder `C#2`.
   Insert the `*V*P` text for tag symbol:

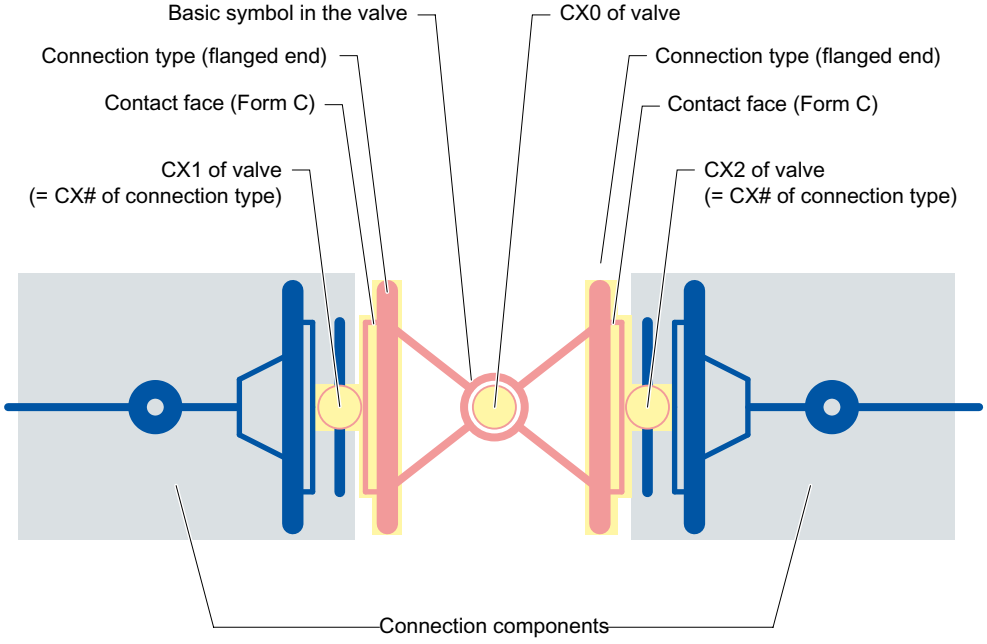


   Highlighted in figure: The insertion point (point of origin) of the tag symbol specified using the text variable.
   Here, a schematic overview of the connectors and subsymbols that will be inserted through these points:

The figures show: 1. Valve 2. Connection type: Flanged end 3. Contact face: Form C	

5. Base object of the shut-off valve: Configuring properties:
   - "SYSISO" tab (Y00T00002):
     In the attribute specified by means of the `*V*P` text, set the reference to the base object of the tag symbol. A symbol for the "M25_P1" drawing type must be prepared on its "Symbols" tab. See also chapter General information on base objects of the tag symbols (Page 121).
   - "Y00T00002 3D Geometry" tab:
     "Y00A00089 Contact face" attribute
     Must be stored with the standard table for contact faces:
     For DIN/EN: "Standard tables > Y10 > M20 > Y30 > A10 > 1 > A60 Contact faces"
     Select the desired contact faces.
     "Y00A005781 Connection type" attribute
     Must be stored with the standard table for connection types:
     "Standard tables > Y10 > M23 > A10 > Y10M23N00004 Connection types"
     Select the desired connection type.
     (Number of attributes = Number of placeholder connectors)

6. Place the blocking valve on the isometry.
   Following takes place when the valve is placed on the isometry:
   - A 3D object is created in the background. By means of the 3D connector of the underlying 3D object, the isometry determines which connection direction the individual connectors have.
   - Contact faces:
     Contact faces are assigned to the placeholder connectors.
     For each contact face: The contact face symbol is tilted in the respective connection direction and is placed with its point of origin on the `C#` connector of the component.
   - Connection types:
     Connection types are assigned to the placeholder connectors.
     For each connection type: The connection type symbol is tilted in the respective connection direction and is placed with its point of origin on the C# connector.
   - Connected components:
   - By means of the Connector table of the PipeSpec management, it is determined if connector components are to be created and if yes, which ones.
   - On the isometry (graphic):
     The logical connectors (`CX#`) defined at the symbol of the connection types are connected with the logical connectors of the connected components.
   - In the Navigator (database-side):
     The connectors (CX1, CX2) prepared at the base object of the valve are joined with the connectors of the connected component.
   - Fastening parts:
     By means of the Fastening table of the PipeSpec management, it is determined if fastening components are to be created and if yes, which ones.
7. Result on the isometry:



6.6 Edit with tag symbols

6.6.1 Managing tag symbols for position numbers

The tag symbols with the following base objects all read the position number of their component:

- "Gasket"
- "Position number"
- "Welding point"

This is done via the symbol of the tag object, text variable:

The position number of the component must be saved in the "'Y00T00002.'Y00A00081'" attribute.

As soon as the value of this attribute changes, the value that is in the tag is updated as well.

These tags are created via the mouse context menu. For this, the following conditions must be met:

- the options script of the report template, the base objects of the components, and the properties of the isometric drawing must be configured accordingly. See also section Options script of the report templates (Page 95), keyword "Graphical/Texts".
- The "Allow Drag & Drop" attribute must be deselected on the base object of the tag symbol, "ISO" tab:

6.6.2 "ISO" tab:

Basics

Prepared tag symbols: See chapter Base objects of the tag symbols (Page 121).

- You can reconfigure the prepared flags, if necessary, and change their graphic appearance.
- If required flags are missing, create them new.

You can find more information on this topic in the "COMOS Platform Administration" manual, keyword "Configuring symbols".

There are also many isometry-specific configuration options available in the "ISO" tab at tag symbols:

Attributes of the tab

Control element	Description
"Isometric display" option	Isometric display of tag symbols • Selected: The symbol is turned in the area. The alignment of the label flag cannot be changed anymore using the grab points. – Available for the following tag symbols when creating via the context menu: all – Available for the following tag symbols if dragged to a component with the geo type "Pipe" using Drag&Drop: "A20 Equipment" - "A50 Pipe spec / Nominal diameter" • Disabled: The symbol is always displayed in front view
"Consider compression" option	Set compression key for base object Only relevant for the following tag symbols: • "Gasket" • "Position number" • "Weld"
"Adjust to flow direction" list	Aligning the tag symbol in flow direction Value "Yes": The tag symbol automatically turns to the flow direction after being placed.
"Must be assigned to a component" list	Enables the docking of tag symbols to components Value "Yes": Tag symbols can be docked to a component by dragging them onto the component.
"Allow drag & drop" list	Controls whether the tag symbol can be dragged onto the plan from the base data Value "Yes": The tag symbol can be dragged onto the plan from the Navigator.
"Allow graphical mirroring" list	Allows the mirrored display of the tag symbol. Important for spool marks, since they always have to point in the direction of the spool.
"Allow automatic rotation" list	• Value "No": Default setting for tag symbols with text. • Value "Yes": The tag symbol is aligned on the symbol of the pipe. If the pipe is horizontal, the symbol also sets itself horizontal. The texts within the symbol are rotated automatically in the appropriate direction. The option to manually rotate is turned off. The rotation grab point does still exist, but the rotation is not put into effect, the flag always jumps back to the direction of the pipe

Control element	Description
"Flag is selected with the component" list	• Value "Yes": All flag symbols assigned to a component are selected as soon as the component is selected • Value "No": Only the component is selected, but not the flag symbol assigned to it.
"Grouped attributes" field	The stated components are grouped. The tag symbol is displayed for the entire group. Example: "Y00T00059.Y00A00031; Y00T00011.Y00A00003AA01;"

See also

Using flags (Page 58)

6.6.3 Allocate position numbers via the Navigator

All components below a pipe that meet certain criteria can be assigned a position number. It can also be deleted.

Implementation

A context menu entry is added in the Navigator using a script at a pipe object:

- "Viper"
  - "Set position numbers"
  - "Delete position numbers"

Numbering never deletes or overwrites existing details in the "Y00A00081 position number" attribute.

```
Function OnMenuCreate(Popup,Context)
If me.SystemType = 8 Then
        Popup.Add "Set position numbers","SetIPos"
        Popup.Add "Delete position numbers","ClearIPos"
End If
End Function

Function OnMenuExecute(ID,Context)
  If ID = "SetIPos" or ID = "ClearIPos" Then
         Set IsoLib = CreateObject("COMOSIsoLib.IsoLib")
  End If
  If ID = "ClearIPos" Then
       Set libe = IsoLib.GetIsoLib
       libe.ClearIPos me
       Set IsoLib = Nothing
  End If

  If ID = "SetIPos" Then
        Set libe = IsoLib.GetIsoLib
```

```
        libe.SetIPos me
        Set IsoLib = Nothing
  End If

End Function
```

Requirement for displaying position numbers

- The "position number" attribute is defined on the "System information" tab at the pipe part.
- The "tag symbol for position numbers" attribute is defined on the "System information" tab at the pipe part.
- The pipe part is Bill of Material relevant

Grouping

If the "Grouped attributes" attribute does not exist on the tag symbol or the value of this is empty or invalid or the component is not Bill of Material relevant, there is no grouping completed for the components.

Each component gets its own number.

The existing position numbers of a group are analyzed. This may result in the following states:

- The entire group has no position number: The position number is assigned by the system. See below for assignment of position number by system.
- The entire group has a position number. From CDevice or previous numbering: The position number is adopted.
- The entire group has several position numbers: The larger number is selected and applied by default.

Assignment of position number by system

If a group does not have a position number, the system tries to generate one. The first step is a search for the "Starting position" attribute under the flag and if the value exists it is taken as the start value. You can also set the "Increments" attribute for the increment. The default setting for both attributes is one if no value is set or if the attribute does not exist. If the position number created this way collides with the existing ones, it is set in the current group.

Non-grouped numbering

The component can have two states:

1. "Position number" has a value (old numbering or from CDevice): The value is retained.
2. "Position number" has no value: The value is assigned by the system.

Configuration option in options script

If you delete or create labeling texts from a report user interface via the context menu, the menu entries are defined in the options script of the corresponding report template. See also chapter Options script of the report templates (Page 95), section "Variable "GraphicalTexts"".

See also

Allocated components (Page 119)

6.6.4 Allocated components

When the tag symbols are generated via the mouse context menu, they are created for the following components:

- "Gasket" tag symbol
  Only generated for components that are placed on the isometric drawing and whose function code is in the range `42000` to `42999`. Whether the component has a separate `DocObj` and is Bill of Material relevant does not matter.
- "Position number" tag symbol
  Generated for all components that are placed on the isometric drawing, have a `DocObj` in the Navigator and are Bill of Material relevant. Following components therefore get no tag:
  - Components that were set to not Bill of Material relevant.
  - Some components that, on placing another component, were automatically created through the connector table.
    Example: A flange was placed on a pipe on the isometric drawing, both components have a welded end as connection type. The PipeSpec is configured in such a way that the pipe and the flange are linked with one another through a welding.
    Result: The welding is created in the Navigator under the flange, but gets no own `DocObj`, and hangs on to the `DocObj` of the flange instead. (It is selected automatically with the flange.) The welding gets no tag symbol with the base object "Position number". If the welding was dragged onto the isometric drawing from the symbol bar, it has a `DocObj` of its own and gets tag symbol.
- "Welding point" tag symbol
  Generated only for components that are placed on the isometric drawing and whose function code is in the range `45100` to `49999`. Whether the component has a separate `DocObj` and is Bill of Material relevant plays no role.

6.6.5 Compression key

You can use the options script of the report template to assign a position number to each component or to use a compression key.

In regard to the tag symbols, the compression key is only important if the position numbers are generated automatically (report template: `PositionNrAutoOn = TRUE`).

If `PositionNrAutoOn` is set to`FALSE`, the compression key is only important for the bill of material.

Activating the compression key

Components that have the same values in predefined attributes get the same number during the automatic generation of the position numbers. In addition, they are summed up in the Bill of Material in one row.

- Requirement:
  Options script of the report template, BOMCompressionKey: is set first. See also section Options script of the report templates (Page 95), variable "BOMCompressionKeys".
- Exception:
  The compression key can be disabled for single tag symbols. See below.

Deactivating the compression key

On automatic generation of the position numbers, all components get a separate number (and will be listed in the Bill of Material separately).

Requirement:

Report template, options script: BOMCompressionKey: commented out.

Deactivating compression for single base objects

If the compression is activated via the report template, you can still exclude some tag symbols from the compression:

Base object of the desired tag symbol,

Properties, "ISO ISO" tab, "`BOMKEYENABLED Consider compression`" attribute:

Deactivated (default setting: activated).

Effect:

All tag symbols based on this base object will be excluded from the compression.

In this way, you can, for example, deactivate the compression for welding points and gaskets, but compress all other components.

If the compression is deactivated in the options script, it is not possible to activate it for individual tag symbols using the checkbox.

6.6.6 "Position number isometric drawing" options group

Component-related rules for the assignment of position numbers:

"System information" tab

Control elements	Description
"Tag symbol for position numbering" field	Reference to a base object whose tag symbol is visible if tag symbols are supposed to be generated A *V*P text determines where the tag symbol is inserted on the symbol of the component. See also section Anchor points for tag symbols (Page 108).
"Position number" field	Saves the position number of the component. On the one hand the position number is read out in the bill of materials. On the other some of the tag symbols prepared in the database read out the value entered here. See also section Edit with tag symbols (Page 115).

6.7 Base objects of the tag symbols

6.7.1 General information on base objects of the tag symbols

Tag symbols are added to a component on the isometric drawing, and display additional information on this component. Each tag symbol has a different display type on the isometry.

Tag symbols can be placed on a report and saved there as report elements. However, the objects are not available on the unit page in the Navigator.

Your base objects have the following system settings:

- Class: "Data set"
- Subclass: "None"

Base objects: "@10 > A20 > A30 > A20 > A10 > A10 Isometries tag symbols"

"M25_P1" drawing type

Graphical symbols of the tag symbols for isometries have the drawing type "M25_P1". Text variables can be used to define which attributes are supposed to be evaluated.

6.7.2 Gasket

- Database: "A10 gasket"
- %N text: `%N ComosDevSpec('Y00T00002', 'Y00A00081', 'displayvalue')%`
  Function:
  - Tag symbol for gaskets
  - On the component via the %N text reads which value was entered in the "Y00T00002.Y00A00081 Position number' attribute.

6.7.3 Equipment

- Database: "A20 Equipment"
- %N text: Not defined

6.7.4 Flow direction & text

- Database: "A30 Flow direction & text"
- %N text: Not defined
- Function: Symbol arrow which points in the flow direction.

Has to be created manually in the database. Can only be placed on pipes via drag&drop.

6.7.5 Hangers / support

- Database: "A40 Hangers / support"
- %N text: `%N ComosDevSpec('Y00T00002', 'Y00A00081', 'displayvalue')%`
- Function:
  - Flag symbol with position number
  - Reads the position number: %N ComosDevSpec('Y00T00002', 'Y00A00081', 'displayvalue')%

6.7.6 Nominal diameter / PipeSpec

- Database: "A50 Nominal diameter / PipeSpec"
- %N text: `DN %N ComosDevSpec('Y00T00011', 'Y00A00003AA01', 'displayvalue')% / RK %N ComosDevSpec('Y00T00011', 'Y00A00052', 'displayvalue')%`
- Function:
  - Flag symbol with nominal diameter and PipeSpec
  - Has to be created manually in the database. Can only be placed on pipes via drag&drop.
  - Reads the nominal diameter and PipeSpec currently set on the component:

6.7.7 Position number

- Database: "A60 position number"
- %N text: `%N ComosDevSpec('Y00T00002', 'Y00A00081', 'displayvalue')%`
- Function:
  - Tag symbol for position numbers on components, except gaskets and welding points
  - Reads the position number: %N ComosDevSpec('Y00T00002', 'Y00A00081', 'displayvalue')%

6.7.8 Weld

- Database: "A70 weld
- "%N text: %N ComosDevSpec('Y00T00002', 'Y00A00081', 'displayvalue')%
- Function:
  - Tag symbol for position numbers on welding points
  - On the component via the %N text reads which value was entered in the "Y00T00002.Y00A00081 Position number' attribute.

6.8 Status display on the report permanently assigned by the system

Status displays permanently defined by the system are predefined, but can be altered. They are scripts that are called based on specific events.

6.8.1 Standard table "Connection symbol (Spec brake)"

Database: "Y10 > M00 > A10 > Y10M00N00020 Connection symbol (spec break)"

A script is activated with the "Options > Display connection info > Yes" context menu, which is called as soon as the component is updated or moved on the drawing and it is connected to another component using the report connectors.

This script checks in the database whether the pipe parts connected to each other are derived from different pipe specs.

If this is the case, a flag is inserted at the connection point which states the "Y00T00011.Y00A00052 PipeSpec" attribute in the direction of the respective pipe spec.

See also

Connection objects (Page 81)

6.8.2 Standard table "Pipe: Sheet reference symbols".

Database: "Y10 > M20 > A10 > Y10M20N00004 Pipe: Sheet reference symbols".

If the same component is placed on two different isometric reports, the corresponding component is not duplicated; a reference to the respective component is created instead on the new isometric report. The component is not part list relevant and is displayed as a "phantom component".

6.9 Managing dimensions

6.9.1 "Isometric drawing dimensioning rules" options group

Dimensioning rules are managed at pipe parts on the following tab:

"Attribute > System information", "Isometry dimensioning rules" control group

Dimensioning types

Different dimensioning types are defined for isometrics. If dimensionings are created automatically via the context menu, then this control group is evaluated. The attributes in this define whether a dimensioning is created for this component, and if yes, which root points it will own.

Name	Description	Function
Y00A00077AA01 to Y00A00077AA05	"Dimensioning type 1" to "Dimensioning type 5"	Defining root points for dimensionings

The attribute evaluated for a given context menu command can be configured with the `DimLevels` variable of the options script from the report template. See also chapter Options script of the report templates (Page 95).

See also

Defining root points for dimensions (Page 124)

6.9.2 Defining root points for dimensions

The root points correspond to logical connectors (`CX1`, `CX2` ...) or physical connectors (`CP1`, `CP2`, ...) of components. See also section Creating symbols (Page 107).

Set the dimensions at these root points. The root points are defined in the base object of a component in the "System information" tab: "Dimensioning type 1" to "Dimensioning type 5" attribute.

If these attributes are missing or are blank, the connection types of the component determine the root points.

Notation

Use the following notation:

- "-": continuous dimensioning: no root point
- "0" to "4": the root point is set at a logical connector
- "P1" to "P4": the root point is set at a physical connector.

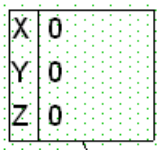
Example

Example for root points of level 2 (attribute Y00T00002.Y00A00077AA02):

- Flange: "P1-" or "-P2"
- Pipe: "-"
- Pipe elbow "-0-"

6.9.3 Changing the name of position coordinates

You can replace the standard names X, Y and Z of the position coordinates with individual names. See also section Dimensioning types (Page 59).



Procedure

1. In the properties of the object "@30 > M00 > A80 > A10 > T > TB > A10 Isometric drawing" select the "Attribute > SYSISO" tab.
2. Enter the required names in the text fields "Name of the x axis", "Name of the y axis", and "Name of the z axis".
3. Save your changes by clicking "OK".

Result

Your individual names of the position coordinates are visible in the coordinate boxes on all new and updated reports.

6.10 Configuring the status display for bolt calculation

Procedure

To enable or disable the status display in the Navigator, proceed as follows:

1. Open the properties of the "@99 > A40 > Y10 > 14 Bolt status" object on the base data page.
2. Open the "Attributes > System data" tab.
3. Activate or deactivate the status display in the "Calculated bolts" list.

6.11 Spools

6.11.1 Detecting the base object for the spool mark

The base object upon which the spool marks are based ascertain the properties of the isometry in the "Spool flag" field.

See also section "Spools" tab (Page 103).

6.11.2 Properties of the base object "Spool marks"

Spool marks are automatically generated when spools are created. They are created on the isometry, but not as objects in the Navigator.

The graphical symbol of the spool mark is always inserted on the physical connectors (CD connections) of the stop components.

Properties of the spool marks

General attributes: See also section "ISO" tab: (Page 115).

Spool-specific attributes:

Control element	Description
"Function code area" field:	Definition of stop components. See also section Adapt stop components definitions for spool marks (Page 127).
"Automatically create a production drawing" list	See also section Fabrication documents (Page 128).
"Construction drawing template" field	References to the template for the production drawing. Only those documents whose report templates are located below the document group folder referenced here are considered during automatic creation or deletion of construction drawings.

Control element	Description
"Spool document template" field	Reference to the report template that is used for spool documents. Same template as for isometries.
"Range of known connection objects" field:	When a spool is being defined, there are connecting components that can only be added once to the component list. The connection object can be defined through the function code. Value: 41000-45999 (Welding points and gaskets). The here placed function codes can be found in the following standard table: "Y10 > M23 > A10 > Y10M23N00001 Function code"

6.11.3 Adapt stop components definitions for spool marks

If the "Spool > Set spool marks" command is called, COMOS determines the limits for the piping construction. Starting from the selected component COMOS searches for the next stop components.

The borders of the piping construction are defined via an attribute on the base object:

"Attribute > ISO" tab, "Function code area" attribute.

Procedure

1. Open the "ISO" tab in the properties of the base object of the spool flag.
2. In the "Function code areas" field enter the number range of the components which you wish to designate as stop components.
   Example: "30000-31999".
3. Optional: Separate different number ranges or individual numbers with a semicolon (";")
   Example: "30000-31999;41100".

Result

The value entered is reconciled with the "Value 1" column of the standard table for function codes in order to ascertain the stop components in the piping construction: "Y10 > M23 > A10 > Y10M23N00001 Function code"

6.11.4 Controlling the spool mark

In the mouse menu of a spool mark, there are also the following commands:

"Spool >"

- "Delete"
  The spool pair is deleted. If a spool document exists for the marks it is deleted as well. Further, the construction drawings of the components placed on the spool document are deleted.
- "Creating documents"
  A spool document is created below the isometry.
  Properties of the document: See also section Properties of the spool document (Page 128).
- "Rotate mark"
  Usually only called after the mark was manually moved and does not point in the right direction.
  The "Graphical mirroring permitted" attribute must be selected on the base object of the spool flag.

6.11.5 Properties of the spool document

Base object:

"@30 > M00 > A80 > A10 > T > TB > A10 Isometric drawing"

See also section Auto-Hotspot.

- Report template:
  Referenced on the base object of the spool flag in the "Spool document template" attribute.
- "Name":
  Same name as the marks
- "Description":
  Same as the description for the report template.
- "Description 2":
  Same description as the marks.

The spool document is blocked. I.e. no construction relevant changes can be conducted. But via the tools of the menu bar you can place free texts and graphics in order to add comments.

6.11.6 Fabrication documents

Requirement

- The "Create construction drawing automatically" attribute is activated at the base object of the spool mark.
- On the base object of the spool mark a reference to a document group is input:
  "Construction drawing template" attribute
  All documents below the here referenced document group are considered as construction drawings.

Create automatically

When opening the spool document for the first time, a check is performed for all components on the spool isometry to see whether a construction drawing is located below their base object.

If such a document is found, it is automatically copied below the component on the engineering page.

See also section Auto-Hotspot.

Create manually

In the Navigator via the context menu of the component.

Delete automatically

If you delete the spool isometry or the spool mark, all construction drawings of the components placed on the spool are deleted as well.

6.12 Retrieving the 3D coordinates of a component using a script

You retrieve the 3D coordinates using the `Get3DDeviceInfo` method for the `Comos.IsometricSpools.SpoolsManager` component.

Example procedure

1. Navigate to the base object of an engineering object whose 3D coordinates you want to retrieve.
2. Open the properties of the object.
3. Open the "Script" tab.
4. Enter the code for calling up the 3D coordinates in a function.
   Example:

```
Dim oSpoolManager
Set oSpoolManager =
CreateObject( "Comos.IsometricSpools.SpoolsManager" )
If Not oSpoolManager Is Nothing Then
     Set xmlFile = CreateObject("Microsoft.XMLDOM")
     Set nodeExport = xmlFile.createElement( "xml3D" )
     xmlFile.appendChild nodeExport
     oSpoolManager.Get3DDeviceInfo <Engineering object whose 3D
     coordinates you want to output>, nodeExport
End If
```

Result

The 3D coordinates of the specified engineering object are stored in the `nodeExport` variable.

6.13 Script functions in options script for isometrics

6.13.1 D3ToIsoAutoOn

If this value is `TRUE`, the corresponding ISO is automatically generated in 3D when an ISO plan is opened.

6.13.2 DimensionTextHeight (Double)

Example / Syntax : `DimensionTextHeight = <parameter>`

Height of the dimension text in millimeters. The default value is `4`. If a different unit has been entered in `DimensionUnit`, the unit entered here in `DimensionTextHeight` still applies in mm and is subsequently converted into the other unit. The parameter can be set to any decimal number in the Double variable value range. The decimal delimiter is set in accordance with the country setting.

6.13.3 DimIsoHeight (Long)

See also chapter Options script of the report templates (Page 95).

6.13.4 DimLevels (String)

See also chapter Options script of the report templates (Page 95).

6.13.5 GraficalTexts (String)

See also chapter Options script of the report templates (Page 95).

6.13.6 IsoEnabled (Boolean)

See also chapter Options script of the report templates (Page 95).

6.13.7 MainDimensionAutoOff (Boolean)

See also chapter Options script of the report templates (Page 95).

6.13.8 PositionDefaultStartNr (Long)

See also chapter Options script of the report templates (Page 95).

6.13.9 PositionDefaultStep (Long)

See also chapter Options script of the report templates (Page 95).

6.13.10 PositionIsMissingAliasNr (Long)

See also chapter Options script of the report templates (Page 95).

6.13.11 SymbolRotationByDimension (Boolean)

See also section Options script of the report templates (Page 95).

User interface reference

7

7.1 Component-specific toolbar

The following two tables show the most important control elements for configuration of component properties and component dimensions.

These component-specific control elements are displayed dynamically, before you place a new component in an isometric report or after you have selected an existing component in the report. If you point the mouse to a control element of the list or option type, the names are shown as tooltip.

Configuring component properties

Control element	Description	Availability
"Specify insertion point" list	Defines the connector at which the component is inserted. Select the entry "Automatic" when mounting components in pipes. The available entries depend on the selected component. The following entries are always included: • "Automatically": Places the symbol in the automatic mounting direction predefined by the system. You can reverse the mounting direction by pressing the <Ctrl> key. • "Point of origin": The component is placed so that the point of origin of the component is located on the insertion position. Available for all objects (if component is to be placed without additional connector)	When creating a new component.
	• "1 -Welded end": The connection is made to the first connector, the welded end. • "2 -Welded end": The connection is made to the second connector, the welded end. • "Automatically": If placed on another pipe, the connection is made as a branch. If pipe is placed at another connector, connector 1 is connected to 2 and vice versa. • "Branch 1 connection": The connection is made to the first connector as a branch. • "Branch 2 connection": The connection is made to the second connector as a branch.	Inserting a welded pipe
	• "1 -Welded end": The connection is made to the first connector, the welded end. • "Automatically": Connection is made with matching end, if possible. The means flange at flange end or weld at welded end. • "2 -Flanged end": The connection is made to the second connector, the flanged end. • "Branch 1 connection": The connection is made to the first connector as a branch.	Inserting a flange
	• "1 -Nozzle weld end": The connection is made to the first connector as welded end. • "3 -Welded end": The connection is made to the third connector, the welded end. • "Branch -1 connection": Corresponds to the "Automatically" insertion entry. The connection is made to the first connector as branch.	Inserting a weldelot
"Fix nominal diameter" option	Specifies that the nominal diameter of a pipe or component is not changed because of components with different nominal diameters. See also section Inserting the component on the pipe run (Page 41).	Only for creating a new pipe or a new component.
"Pipe Spec" list	Specifies the pipe spec.	• When creating a new component. • Connection tool • Selected component
"Nominal diameter 1" list	Specifies the nominal diameter. Before you place a component, you can display all nominal diameters in a list with a right-click on "Nominal diameter 1".	

Control element	Description	Availability
"Nominal diameter 2" list	Specifies the additional nominal diameter. This nominal diameter must be less than or equal to the "Nominal diameter 1" value. One of the nominal diameters must match the nominal diameter of the pipe.	For components with different nominal diameters (e.g., "Reduction").
"Accept" button	Applies the changed properties for the currently selected component.	Only with selected component.

Depending on the component, additional component properties may also be available, for example, the angle of rotation of a non-return valve or a component number. The component properties shown here can be extended as desired by the administrator.

Configuring component dimensions

Control element	Description	Availability
"Dimension level" list	Specifies the level (1-3) for the dimension.	For new dimensions.
"G%" field	Specifies the slope of the pipe. Maximum value: 17.632 %; alternatively you can make entries in "degree" (9.999 °) or ratio (17.632/100).	For dimensions with slope.
"XY" field	Lets you change the currently configured length in the XY level.	
"X" field	Makes it possible to specify the dimension of a component in X, Y, Z direction. The entered dimension is displayed after pressing the <Enter> key in the report. To delete the entered dimension, empty the respective field and confirm with the <Enter> key.	For dimensions of space offsets.
"Y" field		
"Z" field		
"W1/W2" field	Specifies the angle of the space offset. Only "W1" is available with a 2D space offset. In case of two angles, the active angle is highlighted in yellow in the report.	
"L" field	Lets you specify the length of a component.	Only with selected dimensions.
"!" button	Applies default values.	
"X" button	Cancels all default values.	
">" button	Selects the next directly adjacent dimension line of a level.	

Control elements with input field

The values in input fields are highlighted in color according to their status:

- green
  The default value is permitted.
- red
  The entered value is not permitted.
- White
  The displayed value is the actual dimension.
- Gray
  The displayed value cannot be edited.

7.2 "Calculate documents" window

Control element	Description
"Protocol" area	Calculated documents are listed one after another during the calculation. If an error is found in a document, the entry is labeled with "Error".
"Open reports as viewable" option	Selected: Each report is opened as soon as it is calculated. Deactivated: Only the reports in which an error was detected are opened.
"Clipboard" button	Saves the contents of the "Protocol" area to the clipboard.
"Ignore" button	Ignores a detected error and continues with the calculation of the remaining documents.
"Pause" button	Interrupts the calculation and saves the calculation results up to that point.
"Continue" button	Resumes the calculation.
"Cancel" button	Cancels the calculation and saves the calculation results up to that point.
"Finished" button	Closes the window.