SIGMA SINETZ

INGENIEURGESELLSCHAFT MBH | TUTORIAL

4.1

SIGMA Ingenieurgesellschaft mbH
2021

SIGMA Ingenieurgesellschaft mbH
Bertha-von-Suttner-Allee 19
59423 Unna, Germany
www.ROHR2.com +49 (0) 2303 332 33-0

build: February 22, 2021

Contents

User Consulting, Support

If you have any questions regarding the installation and use of the program, please contact the user support. You can contact us by phone, e-mail or via the SIGMA Internet Forum.

Telephone Telephone support is available from Monday to Friday from 9:00–16:00:

GERMAN	ENGLISH
+49 (0) 2303 332 33–33	+49 (0) 2303 332 33–44
`https://www.rohr2.de`	`https://www.rohr2.com`
support@rohr2.de	support@rohr2.de

E-Mail General questions can be directed to support@rohr2.de. If the questions are project-specific, an e-mail can be prepared via the internal program function (Help › SupportRequest). Your default email program will be invoked and the current project will be automatically added as an attachment.

Forum The SIGMA Forum can be reached via Help › Forum or directly via the web address (`https://www.rohr2.de/forum`).

1 SINETZ Program data

1.1 Program descrption

Name:
SINETZ, SINETZfluid
Version:
4.1, release February 2021

1.1.1 Inputs and results

The system has to be divided into pipe segments. A pipe segment consists of a pipe segment with a constant diameter without branches. The pipe sections are connected by coupling points (called nodes).

Results at pipe segmentss:

- Sum of ζ (Zeta values) in a pipe segment
- Volume flow and direction
- Flow velocity
- Friction coefficient λ (Lambda)
- Reynolds number
- Pressure difference
- Temperature difference
- Walltemperature of the insulation (not in SINETZfluid)

Results at nodes:

- Pressure
- Temperature
- Total inflow and outflow·
- Density and viscosity of the medium
- Enthalpy
- NPSH available

The results optionally can be displayed at every segment of the node (cross sections). This can be used for the interpretation of results at nodes with velocity changes where velocity changes have an effect on the static pressure before and after the node.

Results at heat exchangers considering the analyzed mass flow:

- Input pressure considering the pressure loss given by the heat exchanger
- Output pressure considering the pressure loss given by the heat exchanger
- Difference in pressure considering the pressure loss given by the heat exchanger
- Required mass flow
- Input temperature

Results may be graphically or tabular. The graphical representation of the network plan (plot), with the results included therein, is displayed on screen, or produced at a printer or plotter. Extreme flow velocities are highlighted in colors.

1.2 The graphical user interface of SINETZ - Overview

The dispaly of the piping system and the input of the prahics are carried out in the main windows (fig. 1.2.1).

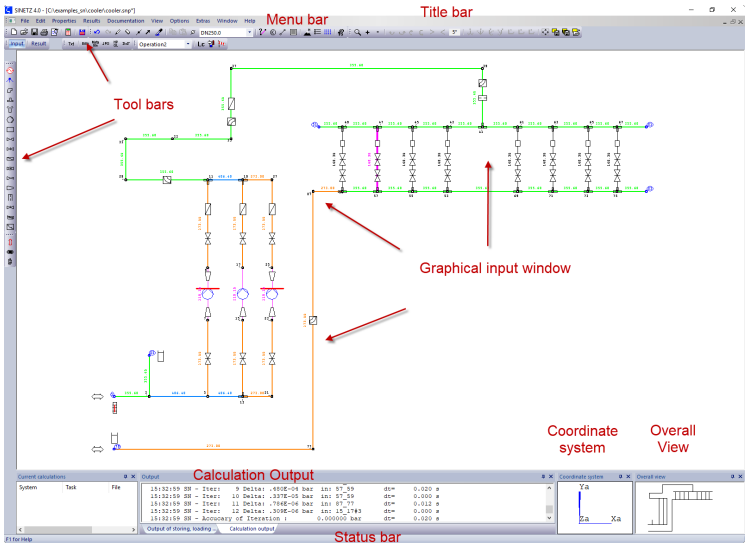

Figure 1.2.1: Main windows of SINETZ

The program functions are also available via the menu commands and the toolbars.

Toolbars Contains icons to execute various functions.

Windows In the main window of SINETZ there are additional windows inserted, which either contain more information about the calculation or provide a better overview.

Title bar Display of the current project including the file path (upper left corner).

Status bar The corresponding program functions or system parameter are displayed (lower left corner).

These elements are freely positionable on the screen.

2 Quick guide

2.1 Using the program

The piping network and all its boundary conditions are entered in the graphical user interface. Based on these information an ASCII input file is generated to start the calculation. This input file may be edited using an ASCII editor.

2.2 SINETZ data formats and interfaces

These data formats can be opened by default:

- *.snp - SINETZ project
- *.nts - Neutral 2D-Interface, to import data from CAD-systems
- *.ntr - Neutral 3D-Interface, to import data from CAD-/CAE-systems using the neutral 3D-interface
- *.dxf - DXF-files
- *.csv - CSV-files

These data formats can be opened if additional programs or interfaces, which are optionally available, are installed.

- *.pcf - Piping component file
- *.n - Intergraph PDS
- *.ntl - PASCE
- *.pxf - AUTOPLANT/AUTOPIPE

For further information about the interfaces, see fig. 2.14. If a SINETZ file is opened, which was created by an older SINETZ version before, a warning message informs about the irreversible conversion of this file. The test license may contain limited amount of interfaces. We recommend to open one of the prepared SINETZ projects or creating new ones under File » New.

2.3 Open or create SINETZ projects

SINETZ is started by doubleclicking the icon in WINDOWS or in the program entry. To create a new SINETZ project or open an existing one, you may proceed as following:

- File > Open
  Opens an existing SINETZ project or imports a possible data format. Alternatively a selection can be made from *List of recently used files*.
- File > New
  Creates a new SINETZ project, which can be defined using three folder levels. The dialog window New Project opens to enter the project relevant information and parameters of the SINETZ project.

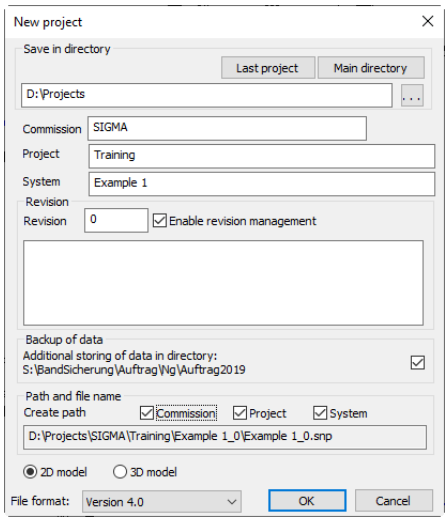

Figure 2.3.1: Creating a new SINETZ-Project

Choose 2D- or 3D-Model The 2D representation is recommended to model a system according to a (P&ID) or layout plan. The 3D representation is recommended to model a system according to isometrics.

2.4 Options

The settings regarding the calculation, which apply to all projects, can be set in Options > General settings. If a new project is created, the dialog window Project settings (fig. 2.4.1) opens automatically, alternatively in Options > Project settings.

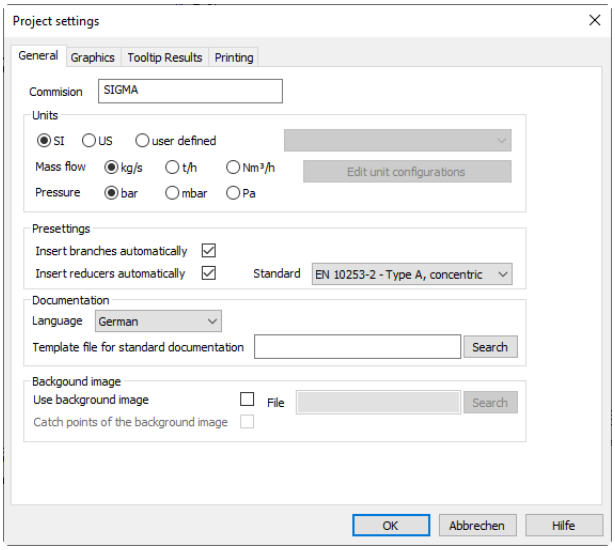

Figure 2.4.1: Project settings

General

- The displayed units of pressure and massflow can be chosen or set individually.
- Branches are inserted automatically, and so for the calcualtion of its zeta values. This option can be switched off as needed.
- Reducers are inserted at nodes automatically, if there are differences regarding the diameters. The calculation of its zeta values are also calculated automatically. To consider the correct angles of the reducers, a norm can be used. This option can be switched off as needed.

Graphics In the register Graphic the settings regarding the graphical display can be set. Size of paper, scale, grid and settings for texts and colors can be customized here.

Tooltip Results The results at nodes and segments can be displayed as tooltips. The tooltips are displayed in the result mode, when the mouse cursor is moved to a node or segment.

Printing In the register Printing the caption of plots and results can be customized.

Info

Each of the mentioned options can be customized at any time, while the project is being edited.

After the project settings are confirmed, the dialog window Dimensions (fig. 2.4.2) opens. In this dialog window the pipe dimensions, which are necessary to the project, can be defined.

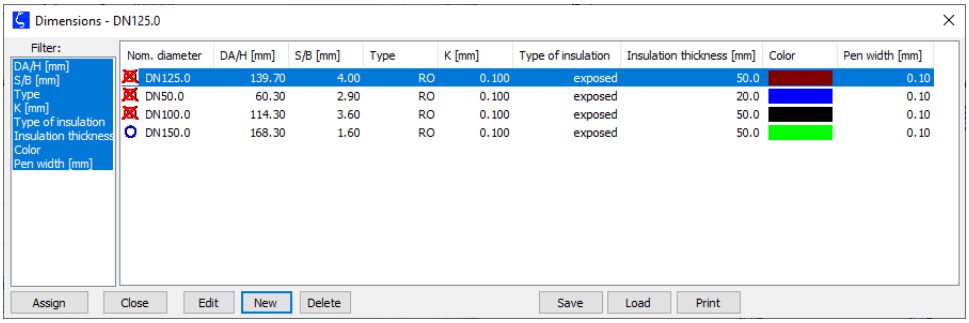

Figure 2.4.2: Pipe dimensions

Pipe dimensions and insulation

Pipe dimensions and insulation can be defined using the dialog window Dimensions (fig. 2.4.2) (also reachable under Edit > Dimensions). The pipe dimension can be selected, with which the input of the system is to be started or continued. If a new project is created the standard pipe dimension DN200 is predefined. Doubleclicking this pipe dimension opens the dialog window Pipe dimensions and insulation (fig. 2.4.3) to change its parameters.

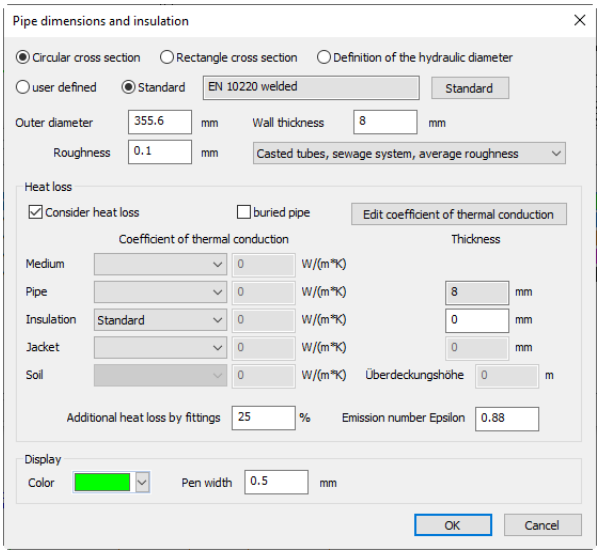

Figure 2.4.3: Definition of a pipe dimension

For a better distinction between the pipe dimensions, color and pen width can also be customized. To define a pipe dimension according to a standard, select the option *Standard* in the dialog window Pipe dimensions and insulation. This opens the dialog window Standard pipe dimensions (fig. 2.4.4).

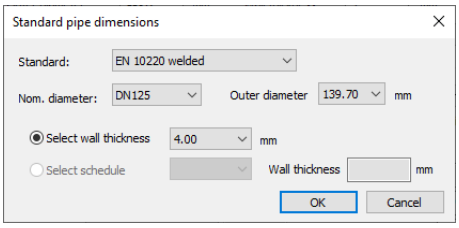

Figure 2.4.4: Selecing a pipe dimension according to a standard

heat loss

If heat losses should be considered, the corresponding function in the dialog window Pipe dimensions and insulation (fig. 2.4.3) is to be selected. If selected, then additional ambient conditions are necessary for each load case.

Additional pipe dimensions can be defined later as needed. Now the dialog windows regarding the pipe dimensions are closed. The next step is to draw the piping system.

The active pipe dimension is depicted in the toolbar (fig. 2.4.5).

The segments, which will be drawn, are assigned with the active pipe dimension. If another pipe dimension is needed, the requested pipe dimension can be selected in the toolbar. If needed, new pipe dimensions can defined in Edit > Dimensions.

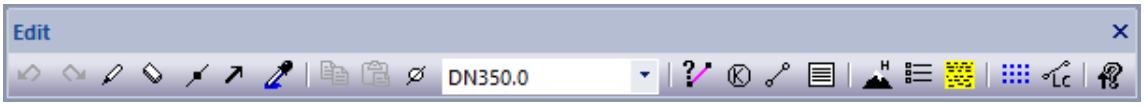

Figure 2.4.5: Toolbar Edit

Alternatively the entire piping system can be drawn, ignoring the pipe dimensions at first. After the system has been drawn, the correct pipe dimensions can be assigned. This method is chosen for this example.

2.5 Geometry input

The system is divided into separate segments. One segment consists of a pipe segment with a constant diameter without branches. By using the function Edit > Draw (or by using the corresponding icon) the system can be drawn. Via the context menu (Right click) *End function* or Esc the drawing function is stopped to, as needed, restart drawing at an intermediate node.

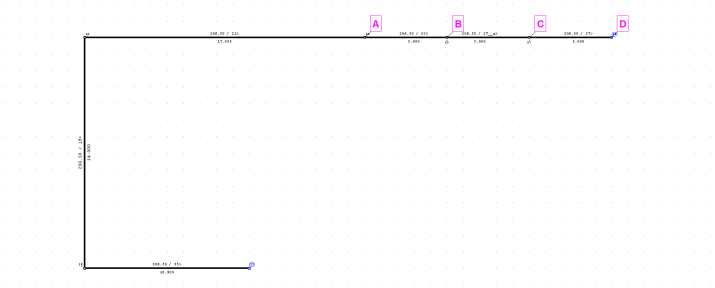

Figure 2.5.1: Creating the main system

2.5.1 Duplicate parts of the system by copy

According to fig. 2.5.1 the subsytems will be inserted at certain nodes, which are highlighted by letters. In this example a subsystem will be created once and duplicated afterwards.

Select Segments can be selected for further editing using the function Edit > Select or  .

Copy The function Edit > Copy or  can be used to extend the piping systems with symmetrical subsystems. Doing so the requested subsystem needs to be selected before, then a base point has to be defined. For more information, see the later instructions to fig. 2.5.2

Move The function Edit > Move or  the position of nodes can be adjusted. If mutiple segments are selected, the positions of each node of the selected segments can be adjusted.

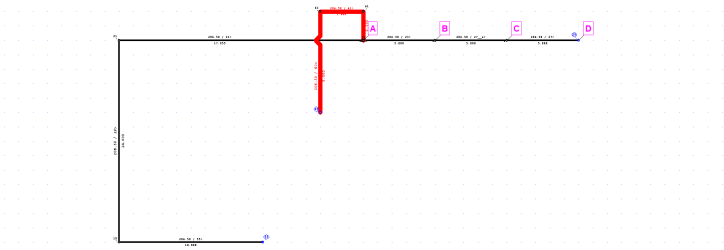

Figure 2.5.2: Creating and selecting the subsystem

At first the subsystem, which will be copied afterwards, is created and selected using Edit > Select or  (see fig. 2.5.2). Then using the following steps the subsystem will be duplicated:

1. Click Edit > Copy or 
2. Click the base point of the selected subsystem (node **A** in fig. 2.5.2)
3. Click Edit > Paste or 
4. Then click the nodes **B**, **C** and **D** one by another (after clicking node **C** a query appears, which should be answered with *Yes*)

Each time something should be pasted multiple times, the query appears. After the above mentioned steps have been followed, the duplicated segments have got the same properties sa the original one (same lengths, dimensions and, if available, the same components) (fig. 2.5.3).

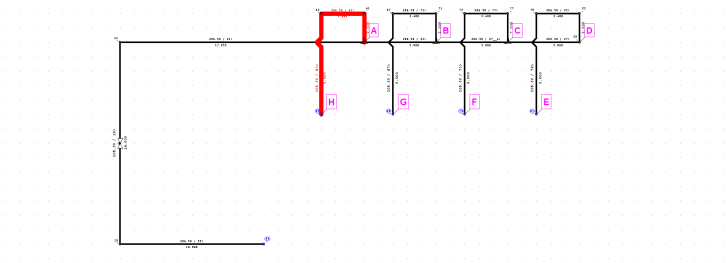

Figure 2.5.3: The subsystem have been copied and pasted

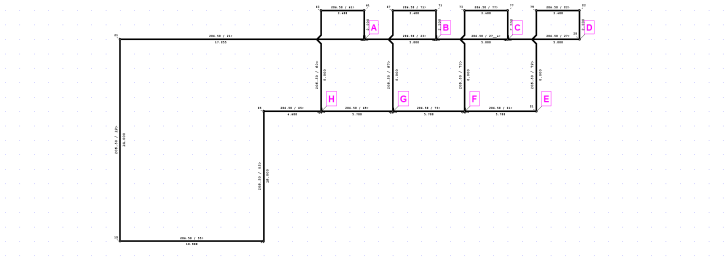

Figure 2.5.4: Completing the system

2.5.2 Changing pipe dimensions

After the system has been created, the correct pipe dimensions can be assigned as following:

- The segments, which shall be assigned with a certain pipe dimension, are selected using Edit > Select or  or the context menu (fig. 2.5.5), which can be opend by rightclicking in the graphical input window.

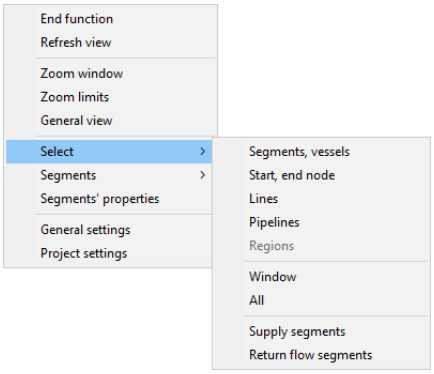

Figure 2.5.5: Context menu - Select

- Now the requested pipe dimension can be selected in the dialog window Dimensions using Edit > Dimensions or 
- Using the function *Assign* in this dialog window, the selected pipe dimensions will be assigned to the selected segments.

2.5.3 Changing pipe segment data

The piping system is created without consideration of a scale. The pipe segments are determined by the scale factor, but can be changed with a double-click. Doing so the dialog window Segment (fig. 2.5.6) opens, displaying its segment data.

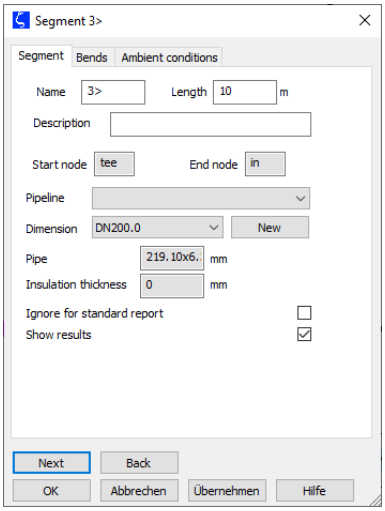

Figure 2.5.6: Dialog window about pipe segments - Register segment

In the register segment the name, the length and the pipe dimension can be changed among others. Using the functions *Next* and *Back* in the dialog window it is possible to easily switch to other segments.

Changing the segment data using *List data*

The function *List data* can be accessed using Properties > List data or  can be used to enter the segment lengths. For further information, see section 2.6.1.

2.6 Checking segment parameters

2.6.1 Segment properties

 Properties > Segments' properties

The properties of the segments can be checked in the dialog window Show segments with properties (fig. 2.6.1)

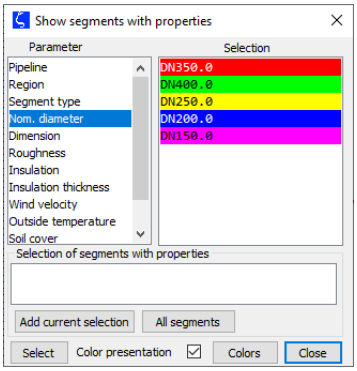

Figure 2.6.1: Display of segment related properties

In this dialog window the requested parameter can be selected. Also this dialog window can remain open while editing the system and all of its functions can still be used. Using this the editing of the system with specific properties can be facilitated. All visible segments are considered when the dialog window Show segments with properties is opened, but the segments, which are hidden at that moment, are ignored.

Parameter, Selection In the list *Parameter* the requested one can be selected. In the list *Selection* all defined properties of the parameter, previously selected in the list *Parameter*, are listed. Selecting a property will display any segment, which also has this property. Multiple properties can be selected by holding Strg and selecting properties one by another. All segments, which don't have the selected properties, are hidden, and the segments, which do have at least one of the selected properties, are displayed as usual.

Selection of segments with properties In this list additionally required properties are displayed. Only segments are displayed that have properties of both lists.

Add current selection The currently selected properties are added to the list *Selection of segments with properties*. Using this the display of segments can be filtered by multiple properties.

Color representation Regarding the selected parameter all properties are displayed in color. All segments are displayed that have the same properties in the list *Selection of segments with properties* if available. The color of the displayed segments are shown in the list *Selection*.

2.6.2 List data

Properties > List data

All data within the system are listed and categorized in the dialog window List data (fig. 2.6.2). The data are accessible via icons and registers in the toolbar of the dialog window List Data .

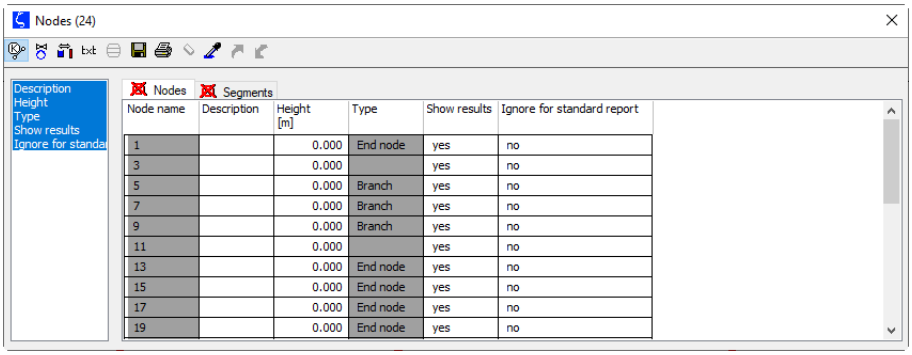

Figure 2.6.2: All data in the system are listed

In each category, accessed via the icons in the toolbar, multiple of its types are listed in the registers. The register itself is marked with if an active element of the type is in the system. If that is not the case, the register is marked with . Clicking the parameter at the top of the list of a register will rearrange the list accordingly.

Editing within the dialog window of an element Double-clicking an entry of a list in the dialog window List data opens the corresponding dialog window, in which the element itself can be edited.

Editing in the list Some data can be edited by clicking the requested parameter of the requested element in the dialog window List data . The new value can be entered in the list directly, but this function does not apply to all parameters. The parameter of the selected entries can be changed in the corresponding dialog windows. In case the lengths of multiple segments shall be changed to a one certain value, the following process is recommended:

1. Open dialog window List data fig. 2.6.2 using Properties > List data or
2. Now the requested segments in the dialog window are selected by holding Ctrl and clicking the segments one by another.
3. Rightclick *Length [m]*, which will open the dialog window Assign length (fig. 2.6.3)
4. Enter the requested length in this dialog window
5. Confirm with *OK*

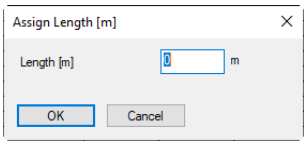

Figure 2.6.3: Assigning the length using *List data*

Info

After entering/ changing data and confirming with *OK* the changes will apply to all selected elements.

2.7 Load case definition

Lc Edit > Load cases

The load cases, which are supposed to be calculated, can be defined in the dialog window Load cases (fig. 2.7.1).

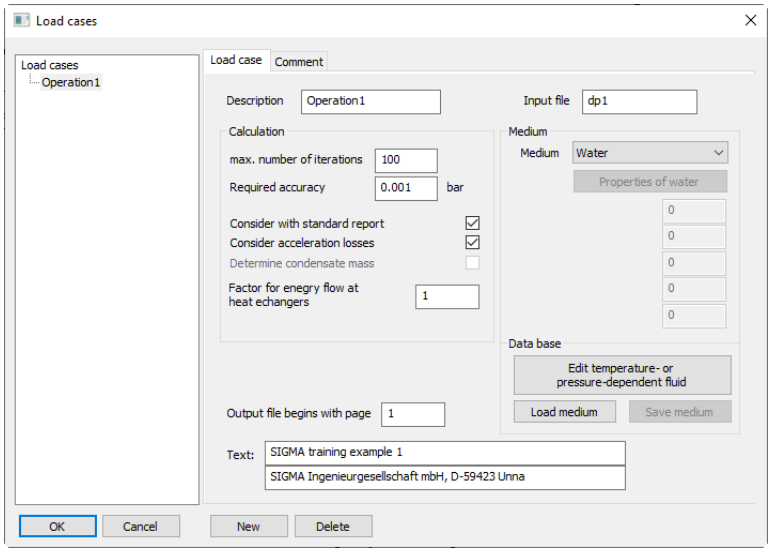

Figure 2.7.1: Dialog window to define laod cases

In this dialog window the following parameters can be defined among others:

- Maximum amount of calculated iterations
- required accuracy of the calculation
- Switching on-/off the acceleration losses due to reducers and branches
- Medium

2.8 Components

2.8.1 Additional resistances

Additional resistances and pressure loss specifications are treated as components and can be accessed via Edit > Components or the toolbar Components (fig. 2.8.1) and inserted by clicking a pipe segment. If multiple load cases are defined, the parameters of most

Figure 2.8.1: Toolbar Components

of the components can be entered dependending on the load case. Doing so load cases can be defined, in which the components can be set to open or closed depending on the operation condition. Reducers can be entered at nodes if its pipe segments have got different dimensions.

2.8.2 Pressure loss e.g. consumers

Edit > Components > Pressure loss

Pressure losses (fig. 2.8.2) are components assigned to the requested segments.

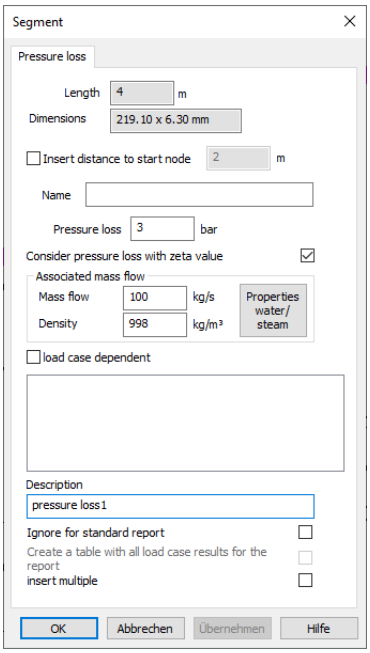

Figure 2.8.2: Dialog window to define pressure loss specifications

More stable calculation using the component *Pressure loss*

To ensure a more stable calculation (imporivng the convergence rate) of the component *Pressure loss* it is recommended to activate the function *Consider pressure loss with zeta value* in the dialog window Pressure loss (fig. 2.8.2). By considering the massflow and the density the zeta value can be calculated.

The function *Properties water/steam* in the dialog window Pressure loss (fig. 2.8.2) opens the corresponding dialog window Fluid properties water/steam (fig. 2.8.3).

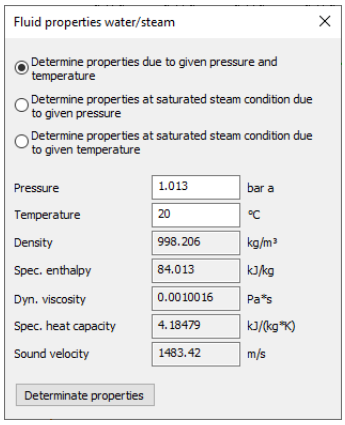

Figure 2.8.3: Calculating the properties of water/steam

The fluid properties of water and steam can be calculated and displayed using a given pressure and a given temperature. By confirming with *OK*, the calculated density will

be applied for the pressure loss specification. If multiple load cases are defined, the parameters for the pressure loss specificaton can be entered depending on the load case.

2.8.3 Pumps

Edit > Components > Pump

After a pump was assigned to a pipe segment, the dialog window Pump (fig. 2.8.4).

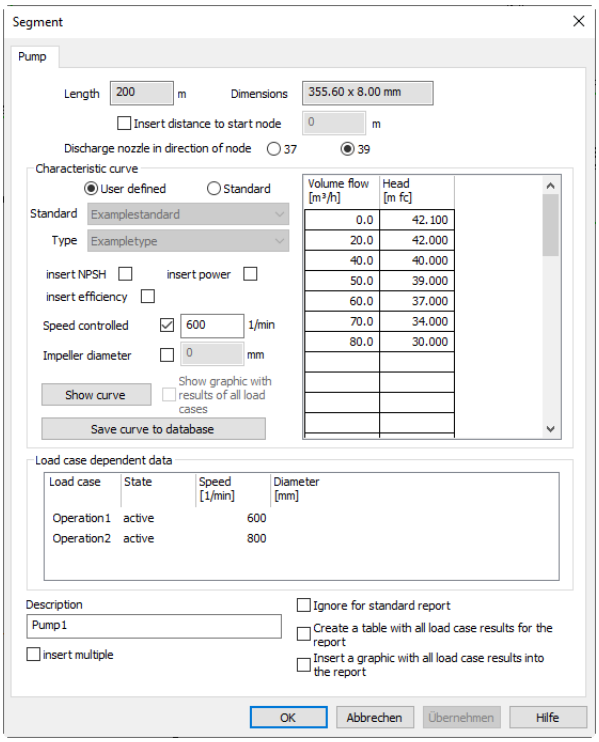

Figure 2.8.4: Pump

Pump curves can be saved to a data base file, if valid values have been added. Pumps can be inserted multiple times. In *Load case dependent data* the pump can among others be switched on or off depending on the load case. By double-clicking a load case in the dialog window Pump (fig. 2.8.4) the dialog window Pump state (fig. 2.8.5) opens.

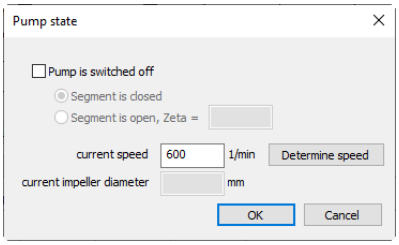

Figure 2.8.5: Dialog window to customize the pump state depending on the load case

2.9 Boundary conditions

 Edit > Insert boundary conditions

In each system it is necessary to assign boundary conditions (pressure, temperature and massflow specifications). At first click the icon  in the toolbar Loads and then click the node, to which the boundary conditions shall be assigned. This will open the dialog window Boundary conditions pressure loss (fig. 2.9.1).

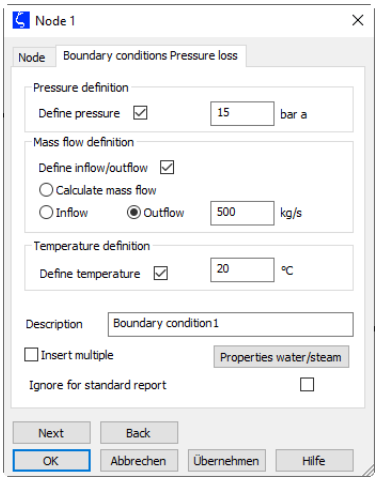

Figure 2.9.1: Dialog window to define boundary conditions

Alternatively the boundary conditions can also be assigned by double-clicking the node, and switch to the register *Boundary conditions and pressure loss* in the dialog window of the node. The following aspects need to be considered when defining boundary conditions:

- For unknown in- or outflows, the option *Calculate massflow* must be enabled. Then the massflow will be calculated by the program.
- At least one pressure specification and one unknown massflow specification must be defined in each load case.
- If pressure and massflow have been defined at at in- or outflow, then the other in- or outflow will have no pressure or massflow specifications.

Necessary boundary conditions for a system to calculate

At least one temperature specification is required. The amount of pressure specifications and unknowm massflow specifications needs to be the same in each load case.
Also for closed systems: In any case at least one massflow specification *Calculate massflow* must be defined in the system.

2.10 Ambient conditions

 Edit > Assign ambient conditions

If heat losses shall be considered in aa certain load case, ambient conditons (Ambient temperature, wind speed and soil temperature) need to be defined. The wind speed is not necessary if the pipes are buried. Ambient conditions can be defined via Edit > Insert ambient conditions or  from the toolbar Loads . This opens the dialog window Ambient conditions (fig. 2.10.1).

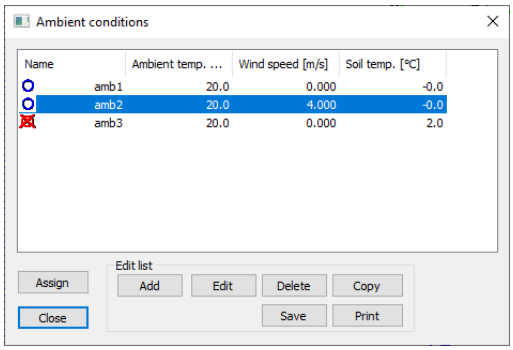

Figure 2.10.1: Listing of all defined ambient data

The function *Add* opens the dialog window Edit ambient data (fig. 2.10.2) to defined ambient conditons.

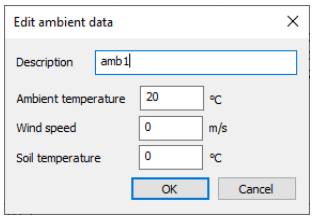

Figure 2.10.2: Dialog window for defining/editing ambient data

The function *Assign* in the dialog window Ambient conditions the currently highlighted ambient data will be assigned to the currently with  selected segments.

2.11 Calculations and results

 File > Calculate

If multiple load cases are defined, it has to be selected in the dialog window Calculation (fig. 2.11.1) which of them shall be calculated.

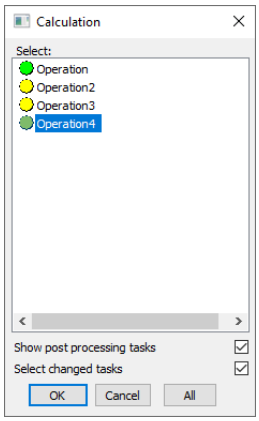

Figure 2.11.1: Selection of the load cases to be calculated

Results

The results can be displayed graphically (fig. 2.12.2 and fig. 2.12.3) or in a list (fig. 2.12.6. To enable the results, it has to be switched in the result mode via the toolbar Mode (fig. 2.11.2)

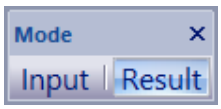

Figure 2.11.2: Toolbar Mode

The output file can be shown in Results > Output file Basically the results of *one* load case is shown. The load case, which results shall be shown, can be determined in the toolbar results (fig. 2.11.3).

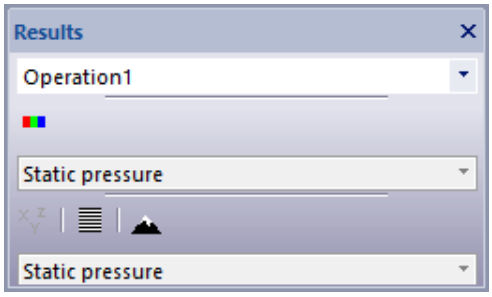

Figure 2.11.3: Toolbar Results

2.12 Results representation in the piping system

2.12.1 Graphicalresultsrepresentation

After the result mode is enabled via the toolbar Mode (fig. 2.11.3), the numerical parameters and results of nodes and segments are displayed at their corresponding locations (fig. 2.12.1).

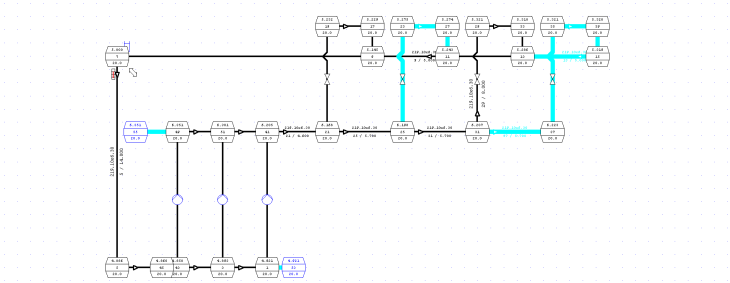

Figure 2.12.1: Simple display of results

The displayed parameters and results can be changed in Options > Project settings > Project settings > Graphics. If thresholds for segments have been defined in Properties > List data > List data > Segments the critical segments will be highlighted.

2.12.2 Height representation of the results output

The results can displayed as heights in Results > Show results distribution or  fromt the toolbar Results (fig. 2.11.3) (fig. 2.12.2).

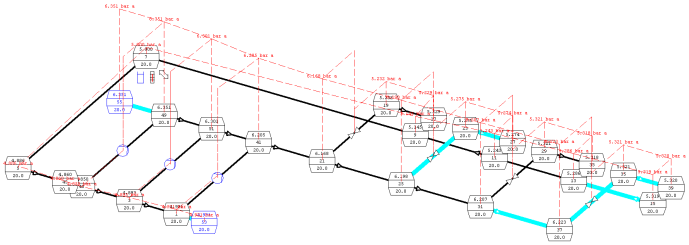

Figure 2.12.2: Display of the static pressure as heights

In the toolbar Results (fig. 2.11.3) the displayed parameter (in this case the static pressure) can be changed.

2.12.3 Colored representation of the results output

The results can displayed as color gradients in Results > Show color presentation or  fromt the toolbar Results (fig. 2.11.3) (fig. 2.12.3).

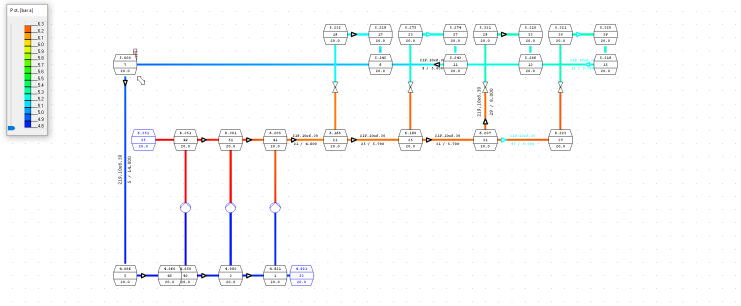

Figure 2.12.3: Display of the static pressure as color gradients

In the toolbar Results (fig. 2.11.3) the displayed parameter (in this case the static pressure) can be changed.

2.12.4 Results at nodes

To display all calculated results of each node in the result mode (fig. 2.11.2), there are multiple options:

- Double-clicking the requested node
- The icon  from the toolbar Edit fig. 2.4.5
- Properties > Node and clicking the node.

The dialog window Results node (fig. 2.12.4) opens, which displays its results.

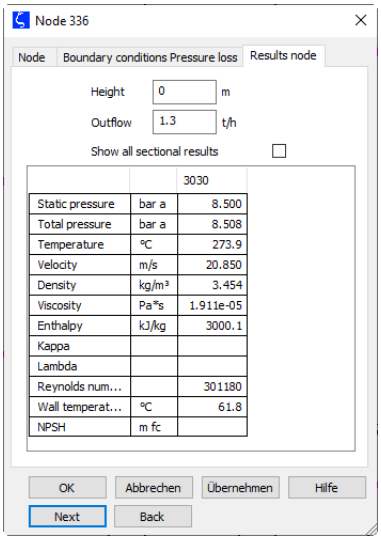

Figure 2.12.4: Results at nodes

At the top of the dialog window the node name, height and calculated in- ou outflow are displayed. Most of the node related results can be shown in Properties > List data, see fig. 2.12.6.

2.12.5 Results at pipe segments

To display all calculated results of each segment in the result mode (fig. 2.11.2), there are multiple options:

- Double-clicking the requested segment
- The icon  from the toolbar Edit fig. 2.4.5
- Properties > Segment and clicking the segment.

The dialog window Results segment (fig. 2.12.6) opens, which displays its results.

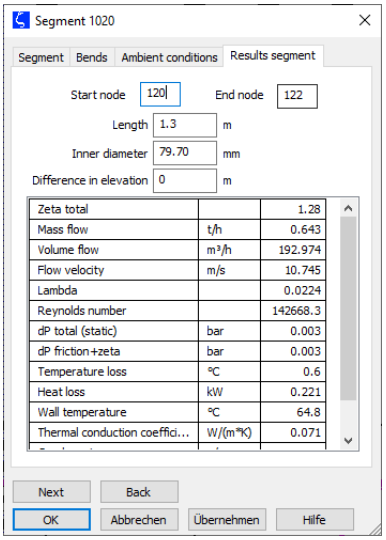

Figure 2.12.5: Results at segments

At the top of the dialog window the segment name, start node and end node are displayed. Most of the segment related results can be shown in Properties > List data, see fig. 2.12.6.

2.12.6 List of results

The results can be listed in the results mode (fig. 2.11.2) via Properties > List data or from the toolbar Edit (fig. 2.4.5) (fig. 2.12.6).

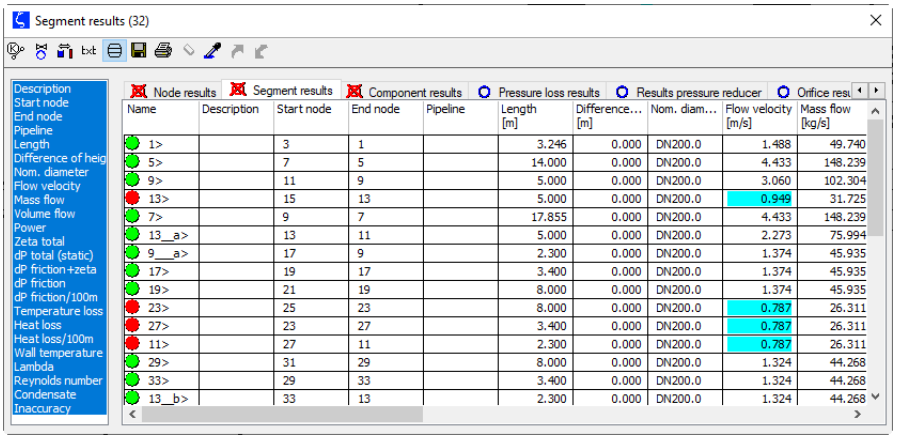

Figure 2.12.6: Listing the results

Critical results are highlighted. The tables can be exported to e.g. Excel.

2.13 Output file

The results are written in an output file in ASCII-format. The Name of the output file refers to the input name and has got the file ending .OUT. The file can be shown in Results > Output file .This file will be printed in the file viewer in File > Print or in the console using the command **PRINT** *Name of the file*. Margins and font size for the print can be customized.

- The headline of each page contains information among others about the used SINETZ version and the date when the output file is created.
- Further headlines contain in Options Project settings Project settings > General entered commission.
- Components in the system are listed in tables.
- Each pipeline will be printed including their pipeline name and the associated node names. Please note that the sequence of the nodes is given here in positive flow direction and may differ from the input data.
- Then the segment data e.g. length and diameter are listed. Please note, that the printed length is corrected if bends are defined in the segments.
- The maximum velocity in a segment will be printed.
- Equivalent inner diameters are printed for rectangle cross sections.
- The zeta values are changed due to the zeta values of bends and branches.
- The calculated pressure increases due to pumps/fans are printed as negative pressure loss.
- The heat losses due thermal insulation including additional losses will be printed as specific heat loss per metre, where the wall temperature is interpreted as the outer temperature of the insulation.

In a summary at the end the minimum and maximum values for pressure and velocity are printed. Also the entire heat loss of the system due to the insulation will be printed.

2.14 Results output

The output files may be exported in rtf or pdf format for documentation purposes. On the basis of factory templates or user defined templates you are able to create a complete Calculation report of the SINETZ analysis by means of the function Standard report. The documents, exported by SINETZ, can be used as single documents or inserted into the users own documentation.

Save output files in RTF format SINETZ output options > Output file in the SINETZ file viewer Output files of the calculation are stored in rtf-format in the project directory. A format file is required for the conversion into the rtf format. An example format file is stored in the subdirectory
🗀 `\SINETZ\R2rtf\(#SINETZ#.rtf)`

Save output files in PDF format SINETZ output options > Output file in the SINETZ file viewer Output files of the calculation are stored in rtf-format in the project directory. A format file is required for the conversion into the rtf format. An example format file is stored in the subdirectory
🗀 `\SINETZ\R2Pdf\(#SINETZ#.pdf)` A format file is required for the conversion into the RTF or PDF-format. An example format file is stored in the subdirectory and

Create standard report This command is used to create a standard report including inputs and load case results in rtf format. On the basis of a standard format file the input and output tables are created directly by the input tables and results of the current project. The standard documentation is created on the basis of standard templates. The language of the documentation depends on the default program language. User defined templates, e.g. with company specific header or footer, can be selected in Options > Project settings Project settings > General.

Report generation SINETZ offers the opportunity to generate calculation reports. The SINETZ report generation enables to create a calculation report including input data and results basing on factory templates or user defined templates. The creation of the report is carried out by the steps:

- Select report template,
- Specify the content and settings
- Create report

The function *Show report* is used to display a prepared report without actualization or modifications. The report consists of multiple text and graphic modules and can be edited in Documentation > Edit report template and selection of a template afterwards (fig. 2.14.1).

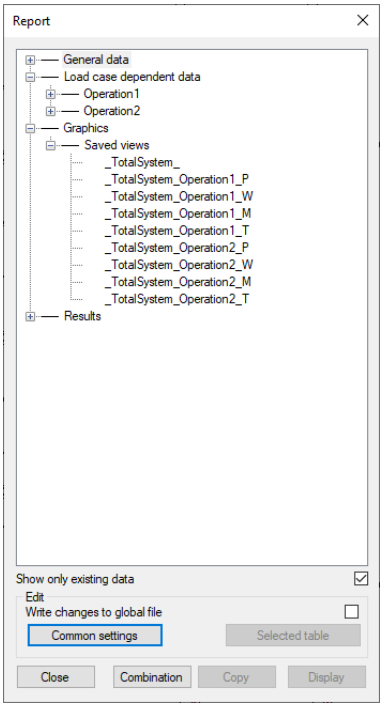

Figure 2.14.1: Listing of the text and graphic modules of the SINETZ report

To insert those modules into the report, selected the requested modul and copy using the function *Copy* in the dialog window Report (fig. 2.14.1). Then the copied module may be inserted at any location in the report (using e.g. ctrl + V). After recalculating the piping system the modifications of the current analysis are take over into the report. After recalculation the report is updated automatically.

3 Checking CAD/CAE import using SINETZ interfaces

SINETZ has got a great variety of interfaces to CAD and CAE programs. The following interfaces are within the scope of SINETZ (default).:

Neutral 2D-interface SINETZ NTS To simplify the data import of data with 2D coordinates (e.g. flow diagrams, P&ID), the format of the neutral interface has been defined, basing on the listing of all elements and their parameters in the system (pipe, bend, instruments, ..) Being based on the listing of all elements in the system it can be created e.g. by a report from a database. The file format is *. nts (Neutral Interface SINETZ). Read the NTS files (*.nts) by File › Open. Choose file type NTS. Several files can be selected in the dialog window. The data of all files is taken over into the calculation system Prerequisite for the correct connection of several NTS-files is that the absolute coordinates of all files refer to the same origin.

Neutral 3D-Interface ROHR2/SINETZ, NTR To simplify the data interchange with 3D-CAD systems, the format of the neutral interface has been defined for the program system ROHR2, basing on the listing of all elements and their parameters in the system (pipe, bend, instruments, ..) The Neutral interface is part of the SINETZ program in release 3.4 or higher as ROHR2 Neutral Interface. The Neutral Interface manual is part of the SINETZ documentation. The file format is *.ntr Reading data by the Neutral 3D-Interface. Read the data by the File › Open command. Using *.ntr interface the formats of the follwing CAD-systems can be imported:

- AVEVA PDMS
- CADISON
- HICADnext
- ROHRCAD

For other CAD programs there are special interfaces, which are available optionally. These interfaces base on neutral interfaces for 3D-files.

DXF interface A SINETZ system (*.dxf) can be generated using P& ID schemes in dxf-format. Based on all lines („LINE“, ”POYLINE“, „LWPOLYLINE“) segments are created in SINETZ. The layers to be considered can be selected. The segment lengths can be calculated using the DXF-coordinates or can be set to a constant value. The function Export › DXF file . . . saves the entire system into a 2D DXF-format. DXF-files can also be entered as graphics in SINETZ using Edit › Insert text, graphic.

CSV interface By using this interface files in text format *.csv (comma separated value) can be imported into SINETZ.

Optional special CAD interfaces

For the following file formats there are interfaces, which are optionally available:

- Isogen (Alias), *.pcf
- Intergraph-PDS, *.n (SMARTPLANT)
- PASCE, *.ntl,
- AUTOPLANT/AUTOPIPE, *.pxf
- PIPENET *.sdf (Export)

These files can be imported as follows:

- The files are converted from the special CAD-format into *.ntr format via File > Open .
- The *.ntr-file is imported into SINETZ via the neutral 3D interface.

Interfaces, which are optionally available, are not part of the SINETZ scope of delivery. The import interfaces are available in the demo version (some of the interfaces are optionally available). By using File > Open it is possible to view file of the corresponding format. Doing so the imported data can be checked by just using the demo version (fig. 3.0.1).

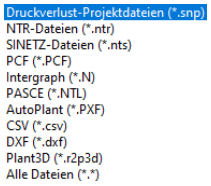

Figure 3.0.1: Available interfaces in SINETZ

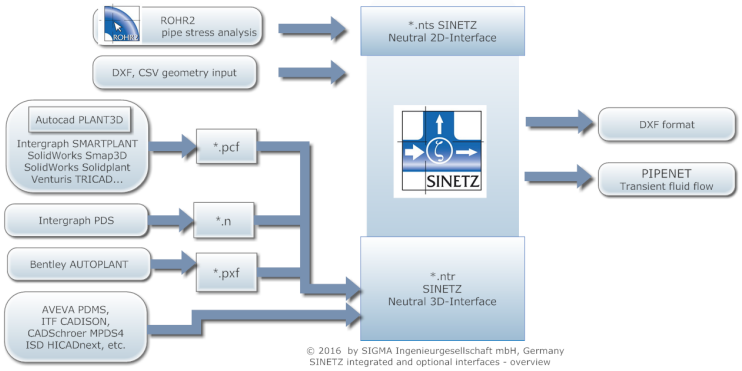

Figure 3.0.2: SINETZ interfaces