

# Wentyle version 5.0 ENG

AutoCAD application supporting the drawing  
and calculations of ventilation installation

## User's Manual 1

### General

This program is used for drawing ventilation installations. The program contains following databases:

- Circular and rectangular ducts and fittings made by **ALNOR**
- Ducts and fittings made of Climaver board supplied by **BH-Res**
- **DOSPEL** fans, plastic ducts systems, grills and ventilation termination elements
- **FLAKT Bovent** wide selection of air diffusers, fire dampers, air volume control

The program enables to conveniently construct an installation and easy introduction of changes and modifications to the designs. The elements are automatically described and upon completion an **automatic specification** of elements used in the project is created.

It is also possible to insert flexible hoses and perform **automatic calculation of pressure drops** for the created installation or a part of it.

Additional information about the program can be found at [www.tomicad.pl](http://www.tomicad.pl), among others the FAQ.

Copyright by TomiCAD Oprogramowanie Projektowe Sp. z o.o.

Should you have any questions concerning the overlay please contact its author:

Tomasz Bieńkowski

[tomicad@tomicad.pl](mailto:tomicad@tomicad.pl)

We appreciate any feedback concerning the operation of the overlay from the users, especially information concerning what do they expect from the future versions.

### Contents:

|                                    |   |
|------------------------------------|---|
| General                            | 1 |
| <u>Compatible AutoCad versions</u> | 2 |
| Installation and startup           | 2 |
| Removing the overlay from the menu | 2 |

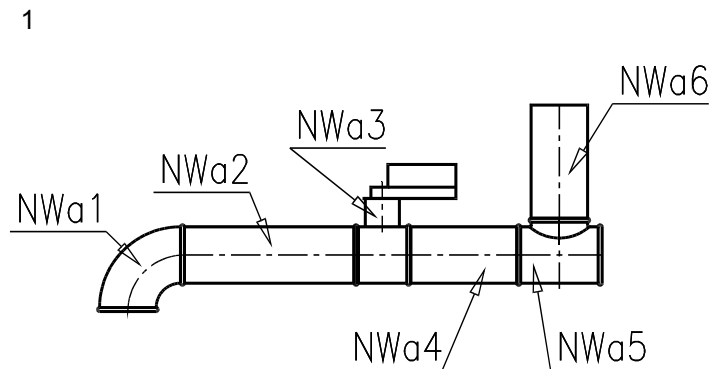
### Using the overlay application

|   |   |
|---|---|
| Designing along the line                | 3 |
| Inserting fittings                      | 4 |
| Viewing the PDF catalogue               | 5 |
| Database of user's sizes                | 5 |
| Moving/changing height of description   | 6 |
| References with arrow and without arrow | 6 |

|                                 |    |
|---------------------------------|----|
| Inserting ventilation ducts     | 6  |
| Elastic hose                    | 7  |
| Editing fitting                 | 8  |
| Adjusting duct                  | 9  |
| <b>"Edit" repair properties</b> | 9  |
| Editing group                   | 9  |
| User's element                  | 10 |
| Renumber                        | 11 |
| Freeze/Thaw                     | 12 |
| Specification of elements       | 12 |
| Inputting into Excel            | 13 |
| Setup – settings                | 13 |

### Calculations

Described in the second part of the manual



| ZESTAWIENIE                                  |  | produkcja SPIRO@system: ALNOR SYSTEMY WENTYLACJI SP Z O.O. |                  |
|--|--|--|------------------|
| ELEMENTÓW INSTALACJI WENTYLACJI MECHANICZNEJ |  | ul. Zwierzyniecka 8b, 00-719 Warszawa                      |                  |
|  |  | tel. +48 22 8511500, fax. +48 22 8511514                   |                  |
| Oznaczenie                                   | Opis elementu                              | Szt.   | Uwagi            |
| Instalacja nawiewna Nwa                      |  |  |                  |
| Nwa 1  | Kolano SPIROsystem BL 100 90               | 1  | producent: ALNOR |
| Nwa 2  | Kanał wentylacyjny SPIROsystem SR 100 300  | 2  | producent: ALNOR |
| Nwa 3  | Przepustnica SPIROsystem DTFL 100 HM024SR  | 3  | producent: ALNOR |
| Nwa 4  | Kanał wentylacyjny SPIROsystem SR 100 188  | 3  | producent: ALNOR |
| Nwa 5  | Trójnik SPIROsystem TCPL 100 100           | 1  | producent: ALNOR |
| Nwa 6  | Kanał wentylacyjny SPIROsystem SR 100 200  | 1  | producent: ALNOR |
| Instalacja wyciągowa Wya                     |  |  |                  |
| Wya 1  | Redukcja SPIROsystem RCPL 160 80           | 2  | producent: ALNOR |
| Wya 2  | Kanał wentylacyjny SPIROsystem SR 160 2000 | 1  | producent: ALNOR |

## Compatibility with individual versions of AutoCAD

Below please find the list of AutoCAD versions, with which the WENTYLE program is compatible. The list also specifies under what conditions:

- Full versions of AutoCAD 14, 2000 - 2009
- AutoCAD LT version 2000LT - 2009LT, providing that the LT-Extender was installed LT-Extender [www.lt-extender.com](http://www.lt-extender.com) (LT-Extender – only in the new version 1.9 ).

Due to the fact that the LT-Extender ([www.lt-extender.com](http://www.lt-extender.com)) software distribution is blocked by court order, soon it will be impossible to purchase the LT-Extender for AutoCAD LT (text written in March 2009) therefore it will be also impossible to launch Wentyle in such configuration.

Of course, machines already running AutoCAD LT with LT-Extender will still be able to use and install Wentyle.

There is also a shortened version for Wentyle for IntelliCAD.

## Installation and startup

The WENTYLE program installation is initiated by executing the Setup.exe file

In case of any problems with the installation manual installation must be performed, according to the instructions from the file Manual installation.txt.

Note: After next reinstallation of the overlay the icon bars, despite the fact they are loaded, might not be displayed. In such case right click on any icon bar and use the Customize option to activate appropriate groups.

After successful installation you can use the overlay at any moment of your work with the AutoCAD.

In case of most icons after clicking and holding the cursor on them an additional vertical icon bar is displayed. If you have any doubt regarding the element to which a particular icon refers, just hold the cursor on that icon for a moment and the description of that icon - element symbol or icon bar description - will be displayed.

## Removing the overlay from the menu

The overlay must be loaded – i.e. any overlay command was executed in the current AutoCAD session. Then in order to uninstall the toolbar enter UNM using the keyboard and press the <ENTER> key.

In order to reinstall the overlay icon bar:

Enter \_appload using the keyboard and press the <ENTER> key, then search the SUPPORT catalogue for Wentyle.lsp file and load it. Now enter MNU using the keyboard and press the <ENTER> key, exit AutoCAD and restart it.

## Using the overlay application

### **Designing along a path**

This method of drawing is used for quickly constructing branches of installations.

After activating the “Designing along SR line”, “Designing along PR line” or “Designing along line” command of the Climaver system, you need to click successive points between which the program will draw the installation, automatically selecting appropriate lengths of ducts and angles of bends.

If, during the drawing of installation branch, you right click on the successive point, you will open a dialog box that enables to:

- Change the section size of the drawn branch – the program, depending on the point that you are in, inserts a reducer or a reducer with a female coupling into the sequence of elements. After selecting the section size and reducer type click “Next”.
- Inserting a T-piece – from the “Insert” panel select “T-Piece” and click “Next”. The program will insert the T-piece and wait for the rotation angle to be specified, and then it shall draw the appropriate missing duct and will wait for consecutive points. Reducer T-pieces can be inserted in a system of rectangular ducts and Climaver ducts, this is why next to the “T-piece branching” list there is another list that specifies the “T-piece outlet”.
- Undoing the last step – “Undo” button
- Completing the drawn branch with a T-piece extended into another branch – select the current point in such a way that the constructed branch will be directed towards an element of another branch (the duct that you want to extend into), right-click to open the dialog box and select “Duct+ending T-piece”.

Constructing the branch can be completed either using the “Cancel” button or the “Ending duct” selected from the dialog box, whichever needed.

Notes:

If you start the drawing by selecting a point on the outlet from a previously inserted element the drawing “along the path” procedure will assume the section size of this point as the initial size.

Even after clicking the first point of the “path”, you can right-click to open a dialog box and change the branch diameter, for example.

If the angle between consecutive segment of the broken line that determines the “path” will be a non-standard one, the program will inform about this and insert the closest normal bend from the database of round bends, slightly crossing the round ducts that contact this bend, or if you are drawing a system with rectangular ducts, it will round the angle to the nearest full degree.

Normally, the procedure of drawing “along the path” does not show the descriptions of inserted elements, they can be made visible using the “Thaw” command-icon or by changing the setting with the “Setup” command-icon.

The descriptions of elements can be moved with the „Move description” command that now deletes the old reference and enables adding a new one.

Bends, reducers and T-pieces can be also inserted on at a time, according to the principles described in following sections. This also concerns all other elements from the program database.

## Inserting a fittings

You can select the command to insert an element by clicking on the element icon. In most cases if you keep the icon "pressed" an entire icon bar of elements of the given type, e.g. "Bend", will be displayed. Holding down the mouse button move the cursor to the desired icon on the bar and release the mouse button.

A dialog window for that element is displayed, in this window you can select the "Size" and "Insertion point" (sometimes also such accessories as: „Damper closing level", "Engine type" etc.). Also enter the installation type (if it was not specified earlier). An element defined in such a way and attached to the cursor can be inserted into the project just like a typical AutoCAD block. After defining the insertion point and rotation angle you still have to define the description position (Drawing G) and insert the mark of reference.

Prompt in the command line:

„Select the description – closer to the end of the future reference" means that you need to click the text closer to the side (under the text) where the "through" line of the reference, which connects the description to the fitting, starts.

You complete the insertion by indicating the end of reference arrow.

Note: If the description (drawing, item C) shall be placed over another, already inserted, description then it will automatically locate itself over the existing description creating another level of the description ladder. For more details on this please see the "Moving the description" section.

**Note:** A "check" mark „>," at the element size means that the given element, and more specifically its size, is provided with data for automatic calculation of pressure drop.

The concept of „Insertion point" (to be selected in the dialog window) is explained by the drawing (item A and B). This is simply the drawing view of the fitting. Red dot on the insertion point selection icons selects the point in which the given view is attached to the cursor at the time of insertion.

Once it is entered in the dialog window the installation symbol is displayed as default when inserting following elements; a consecutive number is suggested for the next element (after deleting inserted elements perform the renumbering in order to keep numbers correct.).

The button „More" displays a window that among others allows entering user's descriptions, symbols and notes into the specification.

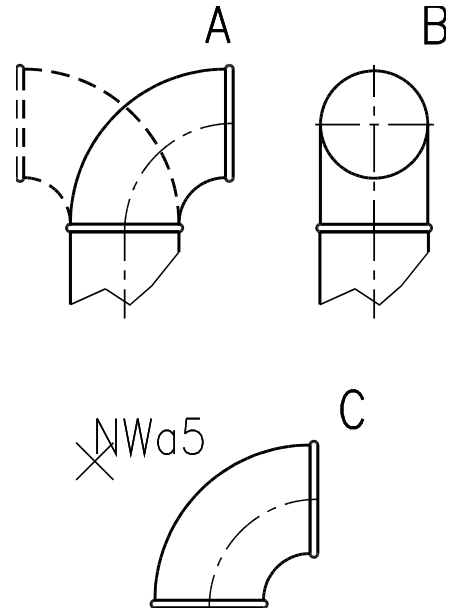
Use axes of ducts as bases when inserting.

The latest inserted diameter is suggested for the next element.

If "Count" selection option in the dialog window is unchecked the inserted element shall not be taken into consideration in the specification of elements contained in the project (this is in case a drawing contains **two views of the same** branch of ventilation ducts - **in this case uncheck the "Count" option for the elements of one of the views of this branch so that the elements of this branch will not be listed twice** - this can be performed with the global editing option.

The position of the description can be changed with the „Move text" button (see below).

The size of the inserted fitting can be changed with "Edit fitting" command, described below.



## Viewing the PDF catalog

From the element insertion dialog window you can view the catalogue page of the given element by pressing the „PDF” button, providing that the **PDF database for WENTYLE overlay**, which is available from the manufacturer on a CD-Rom or can be downloaded from [www.tomicad.pl](http://www.tomicad.pl), was previously installed.

## Changing the color of broken line

The color of broken line, used in case of insulated fittings for example, can be changed by entering KolorPrzerywanej (Broken Line Color) command from the keyboard and entering the color number code (from 0 to 255).

Note: In order for the command to work, at least one element insertion command must be previously executed in the drawing session.

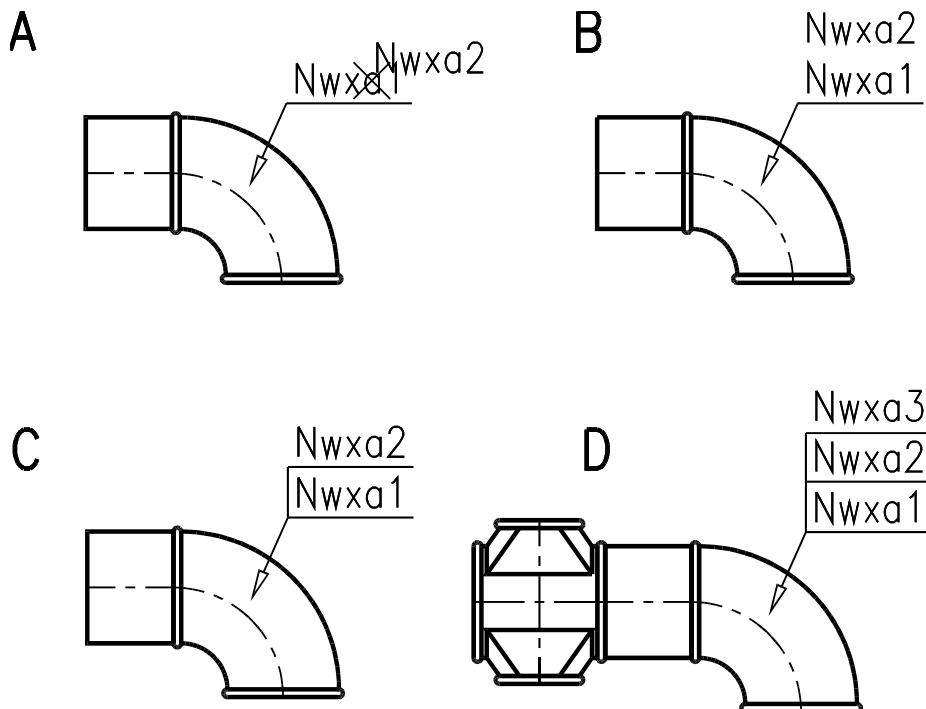
## Database of user's dimensions

In the dialog window of inserted elements, under the “More...” button, there is window with buttons enabling adding a bend size with specified dimensions to the database of user's sizes – the ‘Add’ button.

After pressing the “Add” button the current fitting size will be placed at the **end of list of typical dimensions**. A large number of “user's” dimensions can be added.

Creating a database of user's dimensions is possible for some – parameterized – systems, (e.g. rectangular ducts).

Deleting a size from the database can be performed in a similar way: after selecting the sizes to be deleted on the list of sizes, open the “More...” menu and press “Delete”.



## Moving the description

You want to change the position of selected description using the "Move text" command.

After issuing this command you must first point at the selected element description and then to the new location. If the description was underlined with a reference, you must delete the reference and insert in into the new location using the "Reference with arrow" command or "Reference without arrow" command.

If you want to group the descriptions (which constitute an integral part of element blocks) over one reference, creating something of a ladder, you must simply drop the description being moved onto another existing description, over which you want to locate the description being moved.

Then the moved description will automatically be located over the text onto which it was dropped. This process is shown on the drawing. Add a section of duct to the NWax.1 bend. Drop the duct description onto the existing description of the bend (drawing, item A).

It will place itself as on the (drawing, item B). nNw all you have to do is attach a reference to the moved description using „Reference without arrow" command (drawing, item C).

Item D on the drawing shows the result of adding another level to the ladder – description of the next element, the Nwax3 four-way piece.

## Changing the height of the descriptions

In order to globally change the height of descriptions in the project you must first set the desired height of descriptions in the overlay's "Setup". Then from the "Descriptions" bar select the "Description height" icon/command and select the objects to be changed. If you enter \_all - all elements of the installation drawing will be selected and the height of their descriptions will be changed accordingly.

## Reference with arrow and Reference without arrow

If you moved the element description, you must delete the old reference and insert a new one.

„Reference without arrow" is useful when building cascading (multi-level) descriptions.

Once the command is issued the "Select the description – closer to the end of the future reference" message is displayed, this means that you need to click the text closer to the side (under the text) where the "through" line of the reference, which connects the description to the fitting, starts. You complete the reference insertion by indicating the end of arrow/line of the reference.

## Ventilation duct insertion

Ventilation duct is an object of special class. It has a different meaning in the "Insertion point" dialog window than with the attachments. It determines what data concerning the duct length you will have to provide. This can be a channel of a fixed, standardized length of 3 or 6 meters, the length can also be provided as a arbitrary parameter (second icon in the row), or the program will prompt you to select two points on the drawing, between which the ventilation duct section will "stretch" (last column of icons).

The remaining duct parameters, i.e. size, installation symbol, etc. are entered similarly as for the fitting.

Longer circular ducts will be automatically divided into 3 meter sections, and once the element specification is complete the program will add the number of male couplings necessary to assemble the entire duct.

Also, if a designer inserts ducts with a standard 3 or 6 meter length, the program will automatically add one male coupling for each duct, this is listed at the end of specification in the added male coupling sum.

The inserted channels can be adjusted to the previously modified fittings at their ends ("Adjust duct" command). **A channel subject to this command shall change its diameter and length.**

### ***Selection of the cross-section according to the assumed velocity in the duct***

Enter the throughput [m<sup>3</sup>/h] and maximum permissible velocity for the duct [m/s] and press the "Choose" button to highlight the appropriate diameter on the list of round ducts.

When performing the same selection for rectangular ducts you must also add one of the sides A or B [mm], the second side will be calculated by the procedure, which is tantamount to the selection of cross-section according to the velocity.

Whereas, if we need to calculate the cross-section downstream of a T-connection (designing along a path, rectangular section), change the throughput [m<sup>3</sup>/h] to the throughput downstream of the T-connection, leave box A2[mm] empty (boxes A,B[mm] are filled in) and press the "Choose" button, the program will calculate A2 value, i.e. select the cross-section downstream of the T-connection.

Additionally, by pressing "+" you can add "to memory" the currently set throughput [m<sup>3</sup>/h] to the total of previously stored values, "-" subtracts this value and "C" deletes the contents of throughput total box.

### ***Duct cross-section description***

If the "Visibility descriptions of section" option is enabled in the "Setup", the program will describe each duct providing its diameter or cross-section, for example "fi300" or "400x300".

It is also possible to move the section descriptions using the "Move section description" command and control of visibility by "Show-Hide section description".

Change the height of descriptions using the "Description height" command, according to the coefficient entered in the "Setup".

## **Vent stack**

Vertical ducts (invisible in the project) are inserted in the form of symbolic outline with the characteristic skew axis line. Enter the length of vertical duct in the dialog window.

The skew axis line, which enables to locate the vent stack insertion point also has an important function, it determines the two duct contact points significant for hydraulic calculations.

It allows connecting the vent stack with two installation levels without the necessity to draw this vent stack on additional section.

This connection is made with the „Connect at a distance" command, in order to do so you must connect one end of the skew line with appropriate view of selected fitting and by issuing the "Connect at a distance" command again you connect the other end of skew line with appropriate view of another fitting.

Pressure drop in the vent stack (vertical duct) is calculated according to the base provided in the duct length dialog window.

## **Look for element**

Searches through the system for an element of the specified number, the element is then indicated by a line drawn momentarily between the element and the cursor.

## Elastic hose

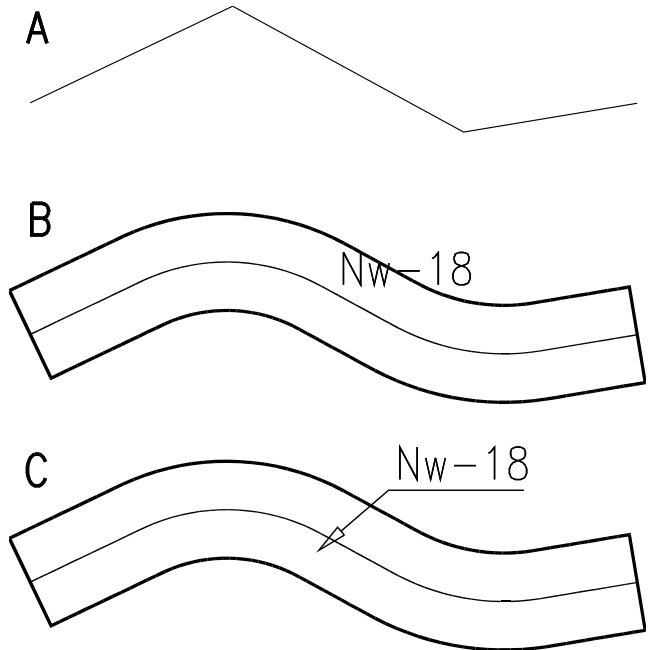
It is a special class object, the command to create this object changes a (previously drawn) poly-line into an elastic hose - see drawing, item A.

The poly-line must not be approximated; the procedure will approximate it automatically. The limits of hose end and beginning height can be entered - the height will be added to the calculated length.

After using the dialog window to select: hose diameter, presentation type selection („Insertion point”) and optionally providing the installation symbol, the hose is drawn as on figure, item B.

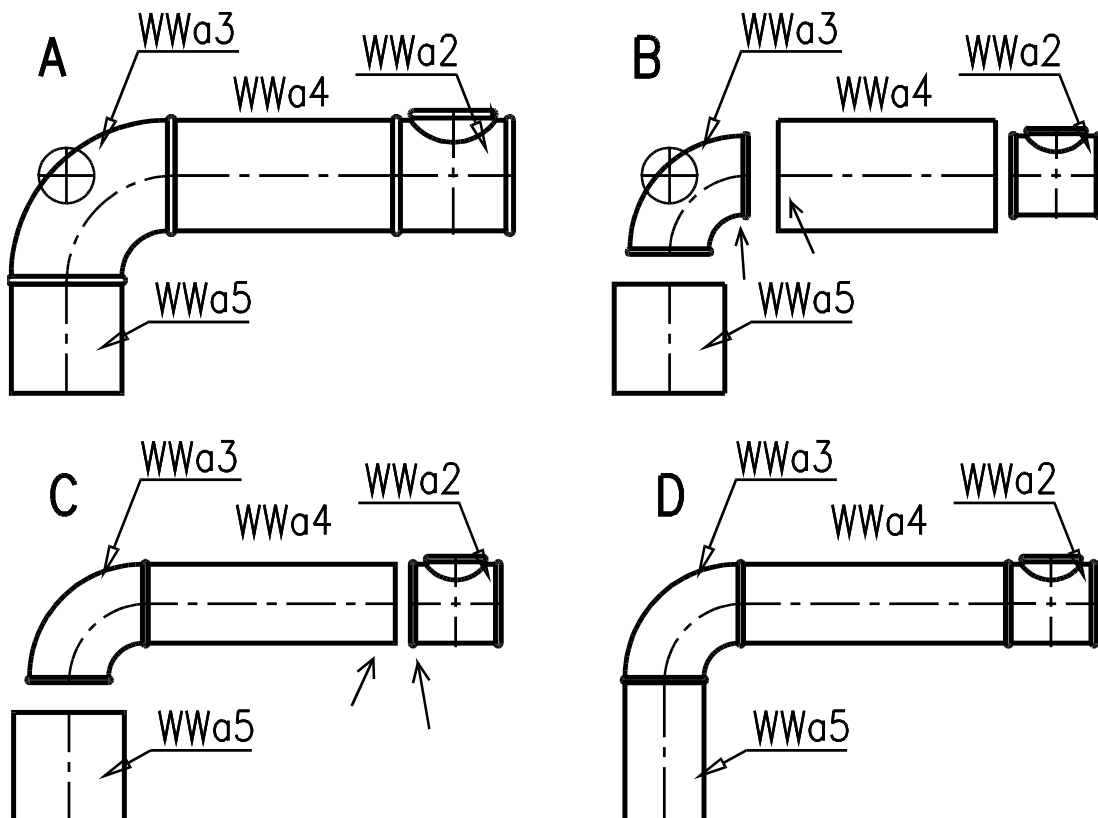
Item C shows the hose after moving the description with the “Move description” command and underlining with a reference with arrow.

Hose length will be calculated and listed in the specification.



A flexible duct is by default drawn using a "Zigzag" line, the drawing options can be changed to a straight line in the overlay's Setup. Note: the "Zigzag" will be often visible only after changing the `_LTSCALE` variable setting.

The program will compute pressure drops [Pa] also on flexible hoses.





## Edit fittings

The "Edit" command (icon) is used to change the size of a previously inserted fitting. On the drawing, item A this command was used to change the diameter of a 90 degree bend from 280 to 200. The result is visible on the drawing, item B.

If you change the fitting size, it will be re-scaled relatively to intersection point of axes, which are going through it, so that the layout of entire installation in the project will be maintained. The normally invisible axes intersection point is marked with a cross in a circle.

On drawing, item B you can also see that the T-fitting size was also changed with the "Edit fitting" command.

Further changes of the installation branch visible on the drawing were made with the "Adjust duct" command described below.

The "Edit" command is used for changing the "Count" attribute, which decides whether the given element shall be listed in the specification or not. If the drawing incorporates a number of views of the same element, then the "Count" attribute should be checked for only one of these views, so that this element will not be listed in the specification several times.

Apart from changing the "Count" parameter – in practice this command is also used to assign and change numbers in the "Installation" and "Numbers" assembly sequence, change specific parameters, for example "Damper closing level", etc.

Please remember that you can also use the edit menu to change the drawing view of an already inserted fitting, e.g. T-fitting. Thanks to this, by copying an entire branch and changing the views of T-fittings in the copied branch with the edit menu we can quickly receive another view of the branch. (Do not forget to uncheck the "Count" attribute with for the second view of the branch using global editing.)

## Adjust duct

This command is used to adjust the previously inserted duct to a new size of adjacent fitting, the size of which was changed with the "Edit" command described above. Adjusting means changing the diameter/section and length of the duct accordingly.

The command prompts you to select the adjusted duct and the fitting to which it must be adjusted. On the drawing on previous page, item B, you could see a duct and an adjacent fitting, as shown by the two arrows. The result is seen on the drawing, item C - the duct is adjusted to the fitting.

The command was repeated for the other end of the duct, adjusting it to the T-fitting, the result is shown on drawing, item D.

Similarly the short vertical duct was adjusted to the bend (second bend outlet).

## Editing groups

If after selecting the "Edit" command you select not one but multiple elements to be edited, or select the "Edit group" command, a dialog window will be displayed; this window allows to change common parameters of elements, e.g: installation designation, installation type, attribute - is the element to be listed in the specification or not, etc.

The "Edit group" command also allows easily selecting an entire installation for editing.

### ***Repair properties of "Edit" menu***

It sometimes happens that the elements in the created installation lose their points of contact. Then the calculation of pressure drops becomes impossible.

If, for example, you **move the grill fixed to the side of the duct**, relatively to this duct, these elements will lose their points of contact, the same problem may occur if we delete such a grill.

In order to repair the connection of the duct with elements fixed to it (grills, saddle piece, etc.) you must use the "Edit" or "Edit group" command.

In such case you must select the "damaged" duct and all elements (grills) fixed to it from the side, then the program will know that you probably want to repair connections and after the standard "Edit group" dialog menu an additional window will be displayed, where you can select the "Repair connections" option.

Once the aforesaid procedure is completed the calculating the described elements will be possible.

This method also allows connecting the copied grills with other ducts.

In some situations, after complex AutoCAD operations simple elements such as bend, T-fitting or duct **lose the points of contact with adjacent elements** – a message informs about this when installation calculation is attempted.

This can be fixed by "Editing" a single element using the "OK" - select the defective element to be edited and click on "OK" in the dialog window. The element will be redrawn and the points of contact will be restored.

Sometimes you must use the "Editing" of a single element to edit the elements that are in contact with the selected one.

### **User's element**

There are two commands allowing to add an own (user's) element to the project.

The first one - „User's element" - takes points of contact with adjacent elements into consideration.

The second one - „Description only=>User's element" described below only requires entering the project description data.

The „User's element" command (icon) allows adding an own (user's) object to the project. This object will be taken into consideration in the created specifications and calculations of hydraulic resistances (with the option to provide own (user's) pressure drop).

After selecting the drawn elements, which are to be included in the "user's" element, on the drawing, you must fill in the boxes in the dialog window:

First of all, specify the name of an AutoCAD block "Block name", which the block of our element will receive in the current drawing.

Then enter the descriptions for the specification: Object name, Object Symbol, Specification remarks and Manufacturer.

The calculation procedure will check the installation, testing the points of contact of adjacent elements, so you must declare such points for your element - as a part of the installation – (there can be a large number of such points).

In order to do this click on the „Show following points", the window disappears and the command line shows a prompt: „>>Specify insertion point" – this is the insertion point of future block. After this the program will prompt you to enter following points of contact, named P1, P2... etc. Once all the points are entered press <Enter>. The dialog window in which we must enter two parameters for each selected point is displayed again.

These parameters are: Size[mm] and Section-Area [m2]

First, highlight the selected point on the list of points, then enter the parameters in the boxes below and press „^” – repeat this for every point.

The provided parameters have secondary importance (description), size[mm] can be found as description in the calculation table, and the section[m2] shall be used for calculating the linear velocity [m/s] in that point, but only to show that velocity in the table, it will not be used for calculating pressure drops on the element. Pressure drop for this element (calculated manually) can be entered with the "dPA in Element" command.

### ***Library of "User's elements"***

"User's elements" defined in one project can be stored on the hard drive and used for another drawing. First of all, you must of course create a "User's element" with an overlay command. Then, from this element you must create a next block (superior) and name it. This block can be saved to the hard drive with "\_WBLOCK" command, using a name that will make it easy for you to identify an own (user's) element, however, this name must differ by at least one character from the name of the block used for defining own (user's) element at the beginning.

The blocks created in such a way, and stored anywhere on the hard drive, can be inserted into drawings when needed using the "\_INSERT" command. After inserting remember to "\_EXPLODE" the inserted block, because it is an auxiliary block, superior to the own (user's) element.

We plan to simplify the aforesaid operations in the following version of the program.

***How to create two views of own (user's) element, so that in the specification they will be listed as two pieces of the same element, e.g. if the project incorporates two Wen-ACm fans, each in a different view.***

Each view must be defined as an own (user's) element. This will create two (ore more) own (user's) elements.

When defining each of them you must enter IDENTICAL description data, except for block name. To differentiate block names you can assign them with different extensions, e.g.: "\_11" "\_12" "\_13" etc. The block names will then look like this: "Wen-ACM\_11" "Wen-ACM\_12" "Wen-ACM\_13" etc.

Overlay elements (blocks) with such extensions are treated as views of the same element and in the specification they will be listed as one element with appropriate number of pieces.

### ***Connecting at a distance using "User's element"***

Some elements, e.g. Bend 90deg. Have their view available in the library; this view can be used for connecting the installation at a distance.

If such a view is missing for a 30 degrees Bend, for example, you can work around this by creating an own (user's) element, i.e. the required view of the bend, and take advantage of the fact that in case of a user's element the points of contact can be defined anywhere, also at a distance from the view of the bend itself.

Using this method you can connect the installation at a distance using an "User's element"

### **Only description=>User's Element**

A simplified command indicating user's own element „Only description=>User's Element"

Does not actually create a User's element itself, but the description of it.

So it can specify an **element, which is not visible on the drawing** e.g. damper in a vent stack, without drawing a section.

Such an approach does not of course require specifying a drawing element, from which the user's element will be created, or moreover, specifying the points of contact.

Using the dialog window you simply specify the name of an AutoCAD block ("Block name"), which will be assigned to the block of our element in the current drawing, and descriptions for the specification: Object Name, Object Symbol, Specification Remarks and Manufacturer. And also the installation number and symbol and description representing an user's element ready for insertion.

## **„Note” command**

This command is used for inserting any remark-type description with a reference.

## **Renumbering**

Is used for renumbering a specified installation or selected group of objects.

Renumbering should be used if, for example, you removed some elements, it will eliminate the numbering gaps.

After selecting the installation or objects to be renumbered a window will be displayed prompting you to enter a number defining the "New number of the lowest selected", if you change that number the renumbering will give you free space in the numbering for the numbers of newly inserted additional fittings, for example.

Simply selecting the renumbering command and pressing OK will only result in setting the number of currently build installation to first available one.

The „Deleted repeating oneself numbers” renumbering option allows to correct frequent installation numbering errors that make it impossible to prepare the system specification (list of components).

## **Renumbering in order**

Numbers the elements in the same order in which they were “clicked” (the number of the first one “clicked” must be specified).

In case of AutoCAD 2004 "clicking the following" must be performed with depressed SHIFT key.

## **Installations are located on different layers**

This enables new commands:

**Freeze** – freezes the specified installation

**Freeze description**- freezes all descriptions

**Thaw** – thaws all description and installations

Color of installation layer (generated quasi-randomly) can be changed.

## **Specification of elements**

Specification of elements, used in the project, is created automatically with the “Specification command (icon) or “Specification according to manufacturers” icon. “Specification” creates a normal list of elements and the “Specification according to manufacturers” creates a list according to manufacturers, In order to simplify the division and sending orders to manufacturers.

In the dialog window you must define whether you want the specification on the drawing - in the form of a table, or as a text file.

The installation description can be expanded, in the installation heading, in the specification, in order to do this you must highlight the installation name on the „Installation description list”, modify the name in „Edit marked description” box and click on “Change”.

You can also set the coefficient of specification size in the form of drawing table “Re-scaling coefficient”.

The specification in the form of drawing table is placed in the middle of the screen, it can, or even must be moved to the place on the drawing provided for the specification. Specification file is always placed in the folder of the current drawing; the user is informed about this.

After all and any changes of ventilation installation in the project you must update the specification (create it again). Then the specification on the drawing will be redefined.

Note: If the specification block was „Exploded” then after the specification update another (updated) specification will be displayed in the middle of the screen and the exploded block will not be updated.

**Note:** if you will need to manually arrange the pages of the specification table on the drawing sheet, or to manually change the text in the specification, you must explode the specification block with the \_EXPLODE command, and then move and edit individual texts and lines. (unfortunately an exploded specification will not be updated after introducing the changes and repeating the specification creation command.

Measurement of the area of main sheet metal fittings and ducts, provided in column „m2”, is carried out according to DIN 18379 guidelines.

## Specification – details

Places the dimensions and other parameters of the fittings in separate columns. The entire specification list can be found in the generated text file the name of which ends with „\_sz”.

This file is located in the drawing folder.

## Specification with ranges of sizes

Divides the fittings and ducts into diameter ranges (up to 100mm, up to 200mm, up to 300mm etc) and circumference ranges (up to 400mm, up to 600mm, up to 1000mm etc.).

Elements not contained in this summary (throttles, ventilators, etc) are revealed at the end of the specification list.

## ALNOR technological export

This type of specification list will be used by the fittings and ducts manufacturer to directly control the machines cutting sheet metal plates.

This will save the manufacturer the necessity to prepare the technological design,

Each of the described types of specification lists contains all elements selected in the project to be included in the specification, therefore it is sufficient to attach one of the described specification lists to the purchase order.

## Pasting the specification into Excel

Create a text file specification.

Then from Excel menu select "Open", set the file type to “Text files” or “All files” and open your file. Then in the displayed window select TAB as the separator, this will allow you to maintain the column arrangement.

## Setup – program configuration

„Setup” command/icon enables configuring some significant program parameters.

The following parameters are configurable:

- Unit of measure, select [mm] [cm] or [m] adjusting the scale of created installations to the drawing>sub-drawing.
- Height of descriptions – conventional description font size
- Or “Reference ended with an arrow”
- „Show descriptions of elements?” – determines whether the description of an inserted element consisting of the installation symbol and consecutive number in the installation is to be shown or not, change of setting will result in changing the status of all previously inserted descriptions.
- or „Description of grill/damper adjustment shall be added to the installation number”
- or „Symbol with element size shall be added to the installation number”
- or whether all letters in installation number are to be capital letters or the first one a capital letter and the rest small letters.