

# **FINE NGAS**

## **Quick Start Guide**

1. Installation – Launching
2. CAD Component
3. Calculation Environment



# Preface

This Quick Start Guide provides a fast and friendly introduction on FINE NGAS' main features and functionalities. In conjunction with this guide, you can find helpful step-by-step videos in [www.4msa.com](http://www.4msa.com). All the features and functions of the program are presented and explained in detail within the complete User's Guide, along with informative examples.

**FINE NGAS**, the **F**ully **I**ntegrated **E**nvironment for **N**atural **G**as Installations combines both designing and calculations in a uniform, integrated environment, consisting of two main components, CAD and Calculations:

- Concerning the **CAD component**, it is based on an autonomous CAD embedding 4MCAD engine adopting the common cad functionality and open dwg drawing file format. The CAD component helps the user design and then calculate and produce completely automatically the entire calculations issue for every Gas installation project, as well as all the drawings in their final form.
- Concerning the **Calculations component** (called also as ADAPT/FCALC), it has been designed according to the latest technological standards and stands out for its unique user - friendliness, its methodological thoroughness of calculations and its in-depth presentation of the results. The GAS Calculation module acquires data directly from the drawings (automatically), thus resulting in significant time saving and maximum reliability of the project results. It can also be used independently, by typing data within the calculation module spreadsheets.

Despite its numerous capabilities, FINE NGAS has been designed in order to be easy to learn. Indeed, the simplicity in the operation philosophy is realised very soon and all that the user has to do is to familiarise himself with the package.

This Guide is divided into 3 short parts:

- Part 1 describes the installation procedure and the main menu structure.
- Part 2 deals with the CAD component of FINE NGAS, showing its philosophy and main features.
- Part 3 describes the calculation environment of FINE NGAS.





---

# 1. Installation - Launching

---

## 1.1 Installing FINE NGAS

1. Insert the CD in your computer CD-ROM drive (e.g. D:, E:) or, if you received your software via Internet, run the installation application you downloaded.
2. When the Welcome page appears (as shown below), click **Next**.



3. When the License Agreement appears, read it carefully. If you agree with the terms, check the respective "radio button" and then click **Next** (you must agree with the terms to proceed with the installation).
4. In the next screen enter your username and organization information and check if you want to create a desktop icon. Then click **Next** to see if the information is correct (see the following window) and finally click **Install** for the installation procedure to begin.
5. Upon completion of the installation procedure, the following last window appears on screen and all needed is to click **Finish**. In case that the **Run FINE 14NG** checkbox is selected, the program will start running.



6. After installation, the program is located within the programs list.

---

## 2. CAD Component

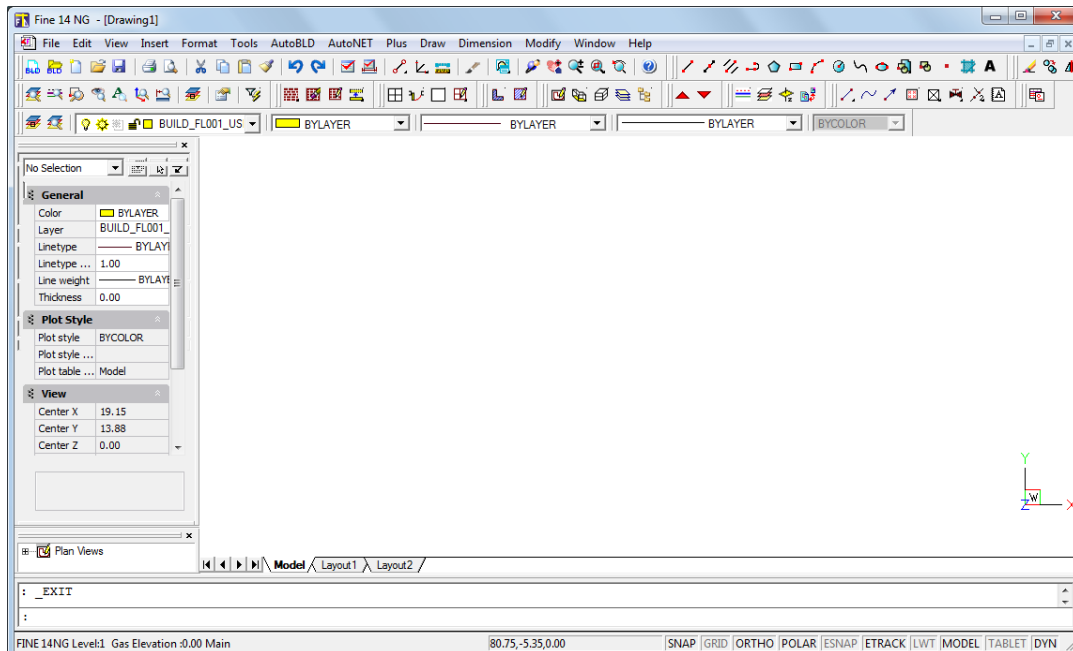
---

### 2.1 Overview

Fine NGAS is a powerful Workstation for Natural Gas installations which automatically performs the necessary calculations directly from the drawings, producing all the case study results (calculation issue, technical descriptions, full-scale drawings, bills of materials etc.). This first Part (Part I) of the user's guide describes the operation of the CAD component of Fine NGAS. As mentioned in the preface, the CAD component is based on 4MCAD technology. Furthermore this CAD component is considering the building and natural gas installation as being composed of intelligent entities with their own attributes and properly related each one to each other. Fine NGAS CAD Component includes 2 main modules, which co-operate closely and give the Designer the impression he virtually works on the building: It is about a) AutoBLD that is used to load-identify the building and b) AutoNET that is used to design and identify the network installations. Those two subsystems are supported by a third one, with the name PLUS, which includes many useful designing facilities.

## 2.2 Main menu

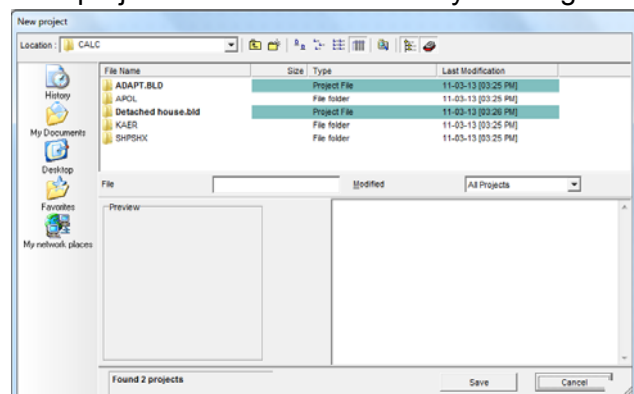
As soon as the program is loaded, the main menu screen appears for the first time:



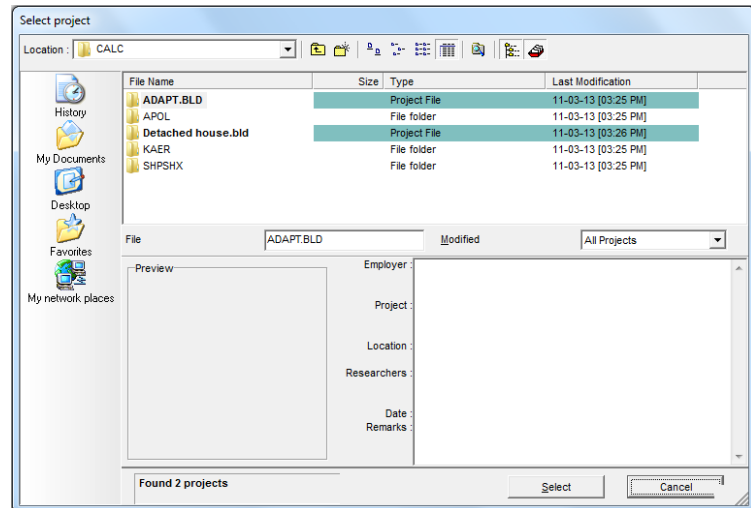
Among the commands of the designing environment, we notice the following main options of the package:

1. Project files management options (New Project, Open Project and Project Information) which are located into the options group FILE.
2. Option Group with the name **AutoBLD**, which includes all the commands required for the Architectural designing.
3. Option group with the name **AutoNET**, which includes all the commands required for the designing and calculation of the application (Single-pipe system, Twin-pipe System, Electrical Wiring etc).
4. Auxiliary option group with the name **PLUS**, which contains a series of designing facilities for the user.

To start creating a project with FINE, a new project should be defined by utilizing the corresponding option in the project FILE management menu mentioned above. In case that "NEW PROJECT" is selected, a window appears on the screen where the name of the Project should be typed.



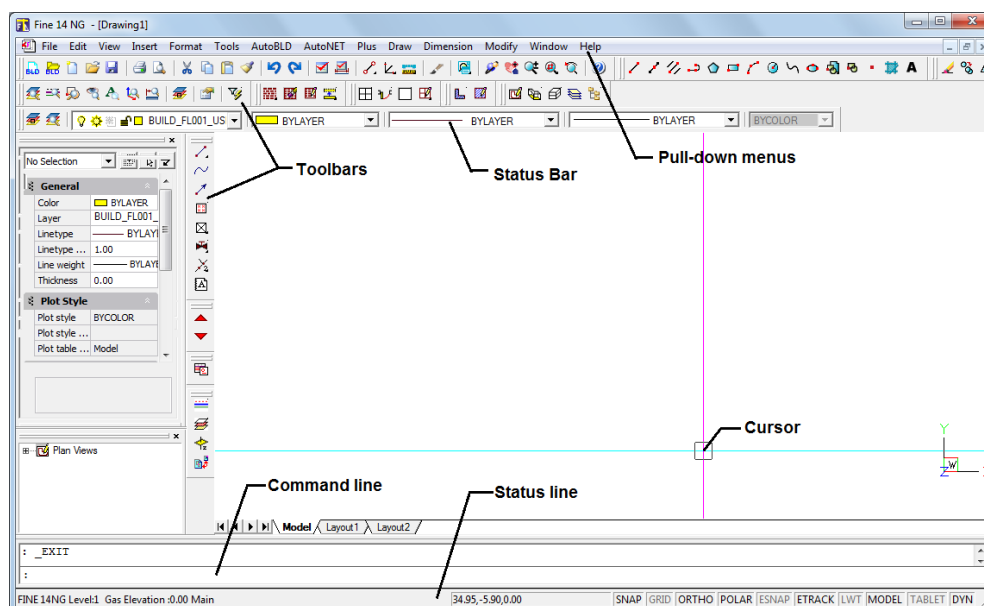
In order to "load" an existing project, which has been created with the program and you want to further edit it or just view it, you choose "Select Project", and a list with the existing projects in the hard drive will be displayed on the screen. At first, the list displays all the projects that exist in the FINE directory and with the use of the mouse or the keyboard and acting correspondingly, you can transfer to any other directory, viewing at the same time the existing projects. It is noted that the projects are included into directories with the extension BLD. If an existing project is selected, it is loaded and displayed on the screen.



Either if a new project is created or a saved one is loaded, you can start working with the use of the subsystem commands described above. A detailed description of these commands is available in the following chapters. Before this detailed description, a short reference of the basic designing principles featured in the designing environment of the package is recommended, in chapter 2.3 that follows next. If you are familiar with the use of 4MCAD or AutoCAD, you may page through or even skip this chapter, while if you are not, you should read it carefully.

## 2.3 Drawing Principles & Basic Commands

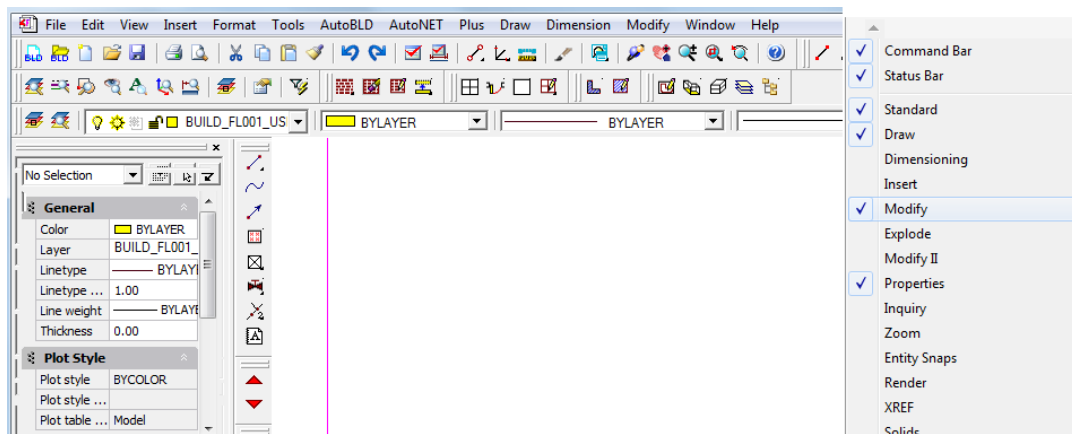
A great advantage of the package is that the structure and the features of the drawing environment follow the standards of the CAD industry adopted by AutoCAD, 4MCAD etc. In particular, the available working space is as follows:



As shown in the above figure, the screen is divided into the following "areas":

- **Command line:** The command line is the area where commands are entered and the command messages appear.
- **Graphics area:** The largest area of the screen, where drawings are created and edited.
- **Cursor:** The cursor is used for drawing, selecting objects and running commands from the menus or the dialog boxes. Depending on the current command or action, the cursor may appear as a graphics cursor (crosshairs), a selection box, a graphics cursor with a selection box etc.
- **Pull-down menus:** Each time you select one of these commands (AutoBLD, AutoNET etc.) a pull-down menu is shown.
- **Status Line:** It is the line on the bottom of the screen where the current level, the drawing status and the current cursor coordinates are displayed. From the status line you can enable or disable tools such as SNAP, GRID, ORTHO etc., which are explained in the following chapter.

**Toolbars:** You can arrange which toolbars you want to be shown in the screen in each project. To enable or disable a toolbar, in the upper part of the screen (where the existing toolbars are shown) right click with the mouse and select the desired toolbar from the list (as it is shown below).



Apart from that, each time you select an application from the AutoNET menu, a toolbar with the name of the application is shown and you can either work from there or from the AutoNET commands.

### 2.3.1 Drawing aids

This section describes the most important drawing aids. These are the commands:

**SNAP:** The graphics cursor position coordinates appear in the middle of the upper part of the graphics area. If "Snap" is selected, the graphics cursor movement may not be continuous but follow a specific increment (minimum movement distance). To change the increment, right click with the mouse on "SNAP" and choose "Settings". To activate or deactivate it, double click on the "SNAP" icon.

**GRID:** The screen grid is a pattern of vertical and horizontal dots, which are placed at the axes intersection points of an imaginary grid. The grid can be activated or deactivated by clicking the corresponding icon or by pressing **F7**.

**ORTHO:** The "Ortho" feature restricts the cursor to horizontal or vertical movement. The status bar shows whether the "Ortho" command is activated by displaying "ORTHO" in black characters. The command is activated or deactivated by clicking the corresponding icon or by pressing **F8**.

---

**ESNAP:** The "Esnap" command forces the cursor to select a snap point of an object, which is within the Pick box outline. The esnap points are characteristic geometric points of an object (i.e. endpoint of a line). If you have specified a snap point and move the cursor close to it, the program will identify it with a frame. The "Esnap" command can be activated either by holding down the "SHIFT" key and right clicking the mouse or through the additional toolbar.

### 2.3.2 Drawing Coordinates

When you need to determine a point, you can either use the mouse (by seeing the coordinates in the status bar or using the snap utilities), or enter the coordinates directly in the command line. Moreover, you can use either Cartesian or polar coordinates and absolute or relative values, in each method (relative coordinates are usually more convenient).

**Relative coordinates:** Enter the @ symbol (which indicates relative coordinates) and then the x, y, z coordinates (Cartesian system) or the  $r<\theta<\phi$  coordinates (polar system) in the command line. The system used (Cartesian or polar) is defined by the "," or "<" symbol respectively. If you do not insert a value for z or  $\phi$ , it will be automatically taken as zero. For example, if you are prompted to locate the second (right) endpoint of a 2m horizontal line, you enter:

@2,0 if you use the Cartesian coordinates (which means that the distance of the second point from the first is 2 m on the x axis and 0 m on the y axis), or

@2<0 if you use the polar coordinates [which means that the second point is at a distance of 2m ( $r=2$ ) and an angle of 0 degrees ( $\theta=0$ ) from the first].

**Absolute coordinates:** They are specified in the same way as the relative coordinates but without using the @ symbol. The absolute coordinates are specified in relation to the 0,0 point of the drawing.

The measurement system can be activated, deactivated or changed with the **F6** key.

### 2.3.3 Drawing Basic Entities

In the "Draw" menu you will find the basic drawing entities:

**Line:** "Line" option is used for drawing segments. When you select "Line" from the menu or type "Line" in the command line, you will be prompted to specify a start point (by left clicking or by entering the point coordinates – relative or absolute – in the command line) and an endpoint (determined in the same way).

**Arc:** The "Arc" command is used for drawing arcs. An arc can be drawn in different ways: the default method is to specify three points of the arc ("3-Points"). Alternatively, you can specify the start point and endpoint of the arc as well as the center of the circle where it belongs (St, C, End). You will not find it difficult to understand and become familiar with the various methods of drawing an arc.

**Polyline:** This command allows you to draw polylines, which are connected sequences of line or arc segments created as single objects. The command is executed by either using the menu or typing "pline" in the command line. You will be prompted to specify a start point and an endpoint (by right clicking the mouse or by entering the point coordinates – relative or absolute – in the command line). Then, the command options will appear (Arc, Close, Length etc). Select **A** to switch to Arc mode, **L** to return to Line mode and **C** to close the polyline.

---

## 2.3.4 Useful Commands

This section includes brief descriptions of the basic program commands, which will be very useful. These are the commands "Zoom", "Pan", "Select", "Move", "Copy" and "Erase" (you will find them in "View" and "Modify" menus). In particular:

**Zoom:** "Zoom" increases or decreases the apparent size of the image displayed, allowing you to have a "closer" or "further" view of the drawing. There are different zooming methods, the most functional of which is the real-time zooming ("lens /  $\pm$ " button). You can use the mouse to zoom in real time – that is to zoom in and out by moving the cursor. There are a number of zoom options as shown by typing "Zoom" in the command line: All/Center/Dynamic/Extents/Left/Previous/Vmax/window/<Scale(X/XP)>.

**Pan:** "Pan" ("hand" icon) moves the position of the visible part of the drawing, so that you can view a new (previously not visible) part. The visible part of the screen moves towards the desired area and to the desired extent.

**Select:** This command selects one or more objects (or the whole drawing), in order to execute a specific task (erase, copy etc.). Select is also used by other CAD commands (for example, if you use the "Erase" command, "Select" will be automatically activated in order to select the area that will be erased).

**Move:** This command allows moving of objects from one location to another. When the "Move" command is activated, the "Select" command is also activated so that the object(s) you want to move (in the way described in the previous paragraph) can be selected.

After you have selected the desired object(s), you are prompted to specify the base point (using the snap options), which is a fixed point of the drawing. When you are prompted to specify the position where the base point will be moved, use either the mouse or the snap options. After you have completed this procedure, the selected object(s) will move to the new position. Please note that the base and the new location points can be also specified with the use of coordinates (absolute or relative, see related paragraph).

**Copy:** The "Copy" option allows the copying of objects from one location to another. The "Copy" procedure is similar to the "Move" procedure and the only difference is that the copied object remains at its original location in the drawing.

**Erase:** Choose this option to delete objects. The procedure is simple: Select the objects you wish to erase (as described above), type "E" in the command line and press <Enter>. Alternatively, you may first type "E" in the command line, then select the object(s) by left clicking and finally right click to erase the object(s).

**DDInsert (Insert Drawing):** This command allows you to insert another drawing (DWG file) or block in the drawing. When this command is selected, a window appears in which you select block or file and then select the corresponding block or file from disk. Then you are prompted to specify the insertion point, the scale factor etc., so that the selected drawing is properly inserted.

**Wblock:** The "Wblock" command allows us to save part of a drawing or the entire drawing in a file, as a block. When this command is selected, you are prompted to enter the file name and then you select the drawing or the part of the drawing you wish to save. The use of this command is similar to the "Screen Drawing" command in the AutoBLD menu, which will be described in a following section. In order to insert a block in a drawing, you use the "ddinsert" command described above.

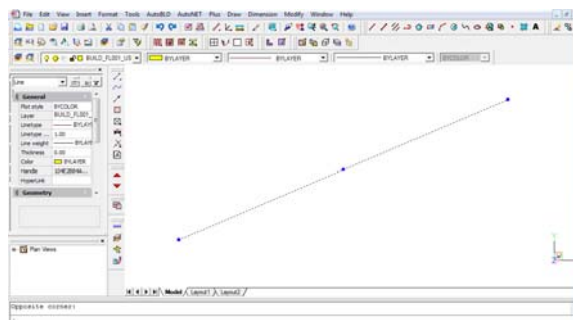
**Explode:** The "Explode" command converts a block in a number of simple lines so that you can edit it in that form. If it is selected, the program will prompt you to select the block ("Select object") you wish to explode.



---

## 2.3.5 Grips

Grips are some characteristic points of an object, which appear after it is selected (by moving the cursor on the object and left clicking). The object is displayed with grips (small squares in blue colour), which mark control locations and are powerful editing tools (by selecting one grip you can for example, move or change the length of the line). When you click a grip, the following prompt appears in the command line: **\*\*STRETCH\*\*** <stretch to point> /Base point /copy/ undo/ exit. If you press <Enter> (or right click), the first characters of the corresponding word are entered, e.g. "sc and enter" for the "Scale" command).



When a command is executed, grips disappear and the objects are deselected. If the command is an editing command (correction or copy), which can be preselected, the objects take part in the execution of the command automatically. In this case, the command overrides the "Select objects" prompt and proceeds. To deselect grips and objects you should press <Esc> twice: once to deselect the objects and twice to deactivate the grips.

In each object the positions of the grips are different. Namely, for a point the grip is the point itself, for a segment the grips are the midpoint and the two endpoints, for an arc the midpoint and the two endpoints, for a circle the center and the quadrants, for a polyline the endpoints of the line and arc segments and the midpoints points of the arc segments, for a spline the spline points, for a block the insertion point, for text the insertion point etc.


## 2.3.6 Print

This section may be read after you have created a drawing and you want to print it. Any drawing can be printed using a printer or plotter or to a file. Printing is performed using "PLOT" command, selected either from the "FILE" menu or typing it in the command line, provided there is a drawing already loaded.

Viewing a drawing before printing gives you a preview of what your drawing will look like when it is printed. This helps you see if there are any changes you want to make before actually printing the drawing.

If you are using print style tables, the preview shows how your drawing will print with the assigned print styles. For example, the preview may display different colours or line weights than those used in the drawing because of assigned print styles.

### To preview a drawing before printing

1. If necessary, click the desired Layout tab or the Model tab.
2. Do one of the following:
  - Choose File > Plot Preview.
  - On the Standard toolbar, click the Plot Preview tool (.
  - Type ppreview and then press Enter.
3. After checking the preview image, do one of the following:
  - To print the drawing, click Plot to display the Print dialog box.

- To return to the drawing, click Close.

The Plot dialog box is organized in several areas as it is shown in the picture below. For help defining print settings before you print, see Customizing print options.


In the plot window, you can select the printer, the paper size and the number of copies and several plot options such as the style (pen assignments), the orientation etc.

Moreover, you can select the plot scale and the plot area. Before you proceed to printing, you select “Apply to layout” and then “Preview” so as to make any modifications you might want.

### To print a drawing

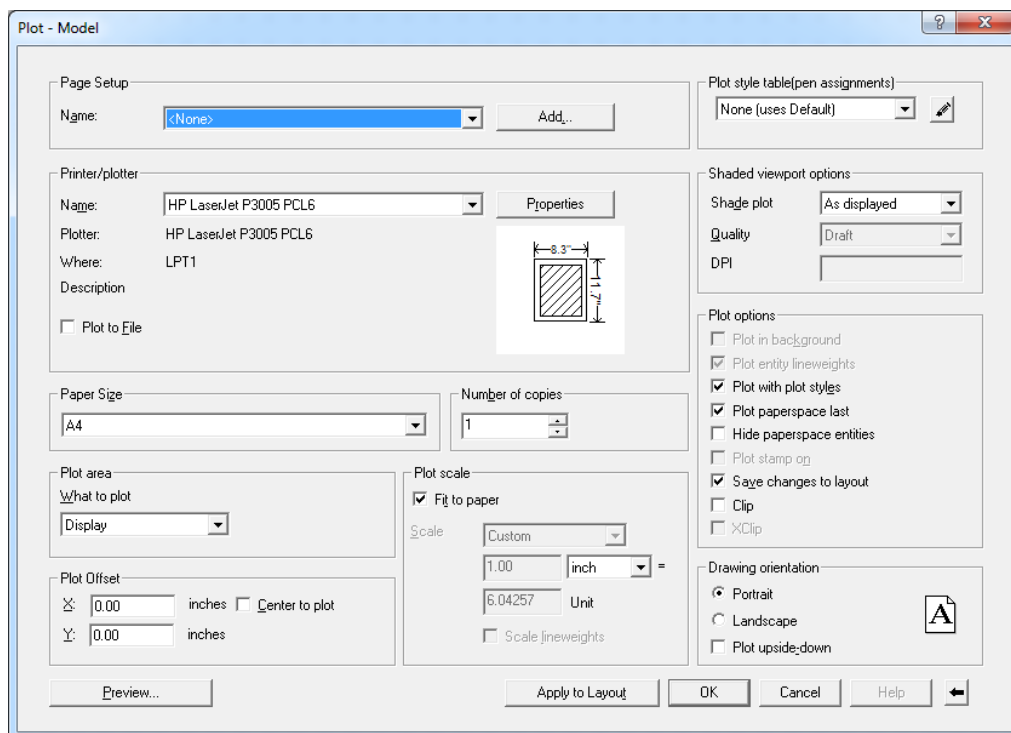
1. If necessary, click the desired Layout tab or the Model tab.

2. Do one of the following:

- Choose File > Plot.
- On the Standard toolbar, click the Print tool (  ). If you click the Print tool, the Print dialog box does not display. Your drawing will be sent directly to the selected printer.
- Type print and then press Enter.

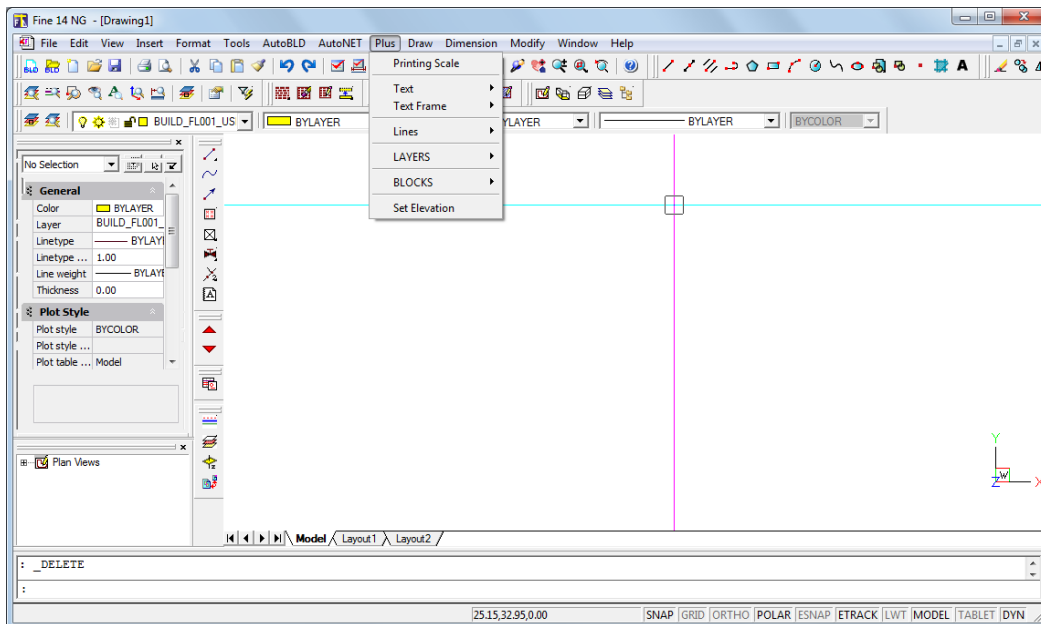
3. From the Plot dialog box, make any adjustments to the settings.

4. Click OK.



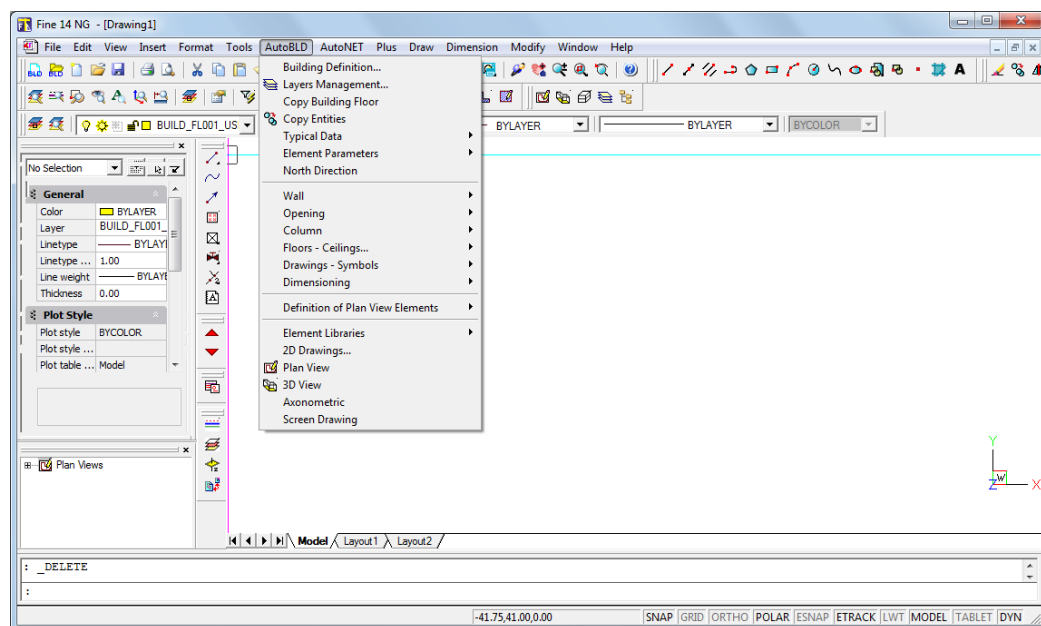
### 2.3.7 Plus Drawing Tools

Those tools belong to the large group of options under the general menu PLUS. These are a series of additional drawing tools, which have been embodied in the package in order to help the user during drawing, and are described within the Full User's Guide.



## 2.4 AutoBUILD: Architectural Drawing

The AutoBUILD option group, as we will see in detail below, includes all the facilities required to insert a building and create an architectural drawing. As it is shown in the corresponding AutoBLD menu, the various options are divided into several sub-groups.



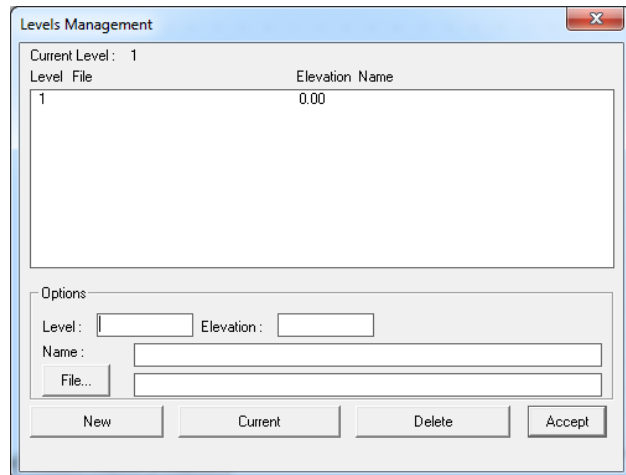
In general, the first sub-group includes commands for the definition of the project parameters, the second sub-group includes drawing commands, the third sub-group includes commands for linking to the calculations and the fourth includes management options for the AutoBLD libraries and commands for the building supervision. In the following sections, the options reported above are described one by one, beginning with the "Building Definition" option.

## 2.4.1 Building Definition

If you select the "Building definition" option, the floor management menu appears.

On this screen the levels of the building are defined, which means that you determine the level and the corresponding architectural drawing (a dwg file ground plan) of each building floor (only in case you use a drawing that was created by another architectural designing program). More specifically:

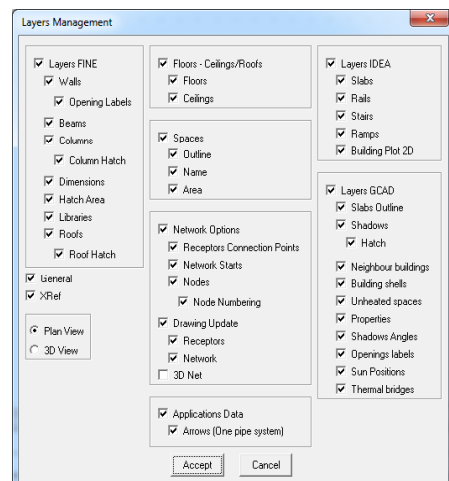
- In the "Level" field, define the Level (floor) number.
- In the "Elevation" field, define the height of the floor level. You can define manually a benchmark for level measurement (e.g. the pavement). You may also define negative levels (e.g. -3 m for the basement).
- In the "Name" field, define the name of the level (e.g. ground floor).
- In the "File" field, define the path and the name of the relevant DWG drawing file, only if you refer to an already existing drawing (which means that you do not intend to draw the ground plan from the start). If there is no DWG architectural drawing available, leave the filename blank.



The insertion and the management of ground plans are performed with use of the xref command. At the bottom of the dialog box there are four functions available which are actually used to manage the floor files. More specifically:

- Press the "New" button to save a new floor or the changes in the data of a floor (e.g. level, dwg drawing).
- Use the "Current" option to select the ground plan/file you want to work on each time.
- Select the "Delete" option to delete the floor you want (after you have selected it). The "Delete" command removes the ground plan of the relevant floor in the project.
- The "Accept" command closes the dialog box (it does not save the floor data. This can be managed with the "New" command). FINE NGAS enables also the use of a "scanned" ground plan, which is a ground plan in a bitmap file created by a scanner. In this particular situation the steps to follow are described in detail within the User's Guide.

The "**Layers Management**" option gives you the choice to enable or disable in a quick and practical way (during working) several layers. If you want, you can disable any element group by simply clicking inside the indicator-box of the corresponding group. When the box is checked, the corresponding group is enabled.



The “OK” command closes the dialog box (does not save the floor data). This can be managed with the “New” command). FineNGAS enables also the use of a “scanned” ground plan, which is a ground plan in a bitmap file created by a scanner. In this particular situation the steps to follow are described in detail within the User’s Guide.

The “Layers Management” option enables the user to define in a quick and very practical way (during working) the logical minor drawings of the ground plans (layers). If the user wishes, he may disable any element group, by simply clicking inside the indicator-box of the corresponding group. When the box is checked, the corresponding group is enabled.

## 2.4.2 Drawing Walls

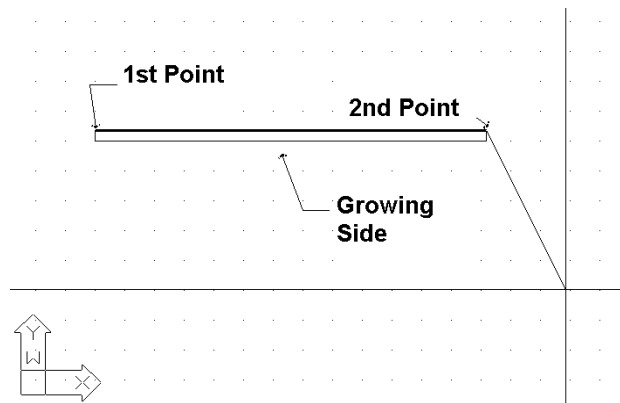
AutoBLD menu contains all the commands required for drawing and editing walls, such as parallel moving of walls, trimming, extending, joining and breaking walls as well as placing openings of any kind on them (windows, sliding doors, openings, arches). During the initial drawing, as well as during any modification at any stage, the drawing is automatically updated (e.g. placing an opening on a wall does not break the wall in two parts, the opening moves easily from side to side whether you are working on the ground-plan or on a 3D view and the wall is restored without leaving undesirable lines after deleting an opening etc.).

The **Wall** option, located at the second subgroup of the AutoBLD group of commands, includes the Outer, Inner, Outer wall from polyline, Inner wall from polyline and Outline options as well as the option subgroup Modify, Delete, Extend, Break, Join, Trim and Move. The first subgroup concerns the wall drawing, while the second their further processing after being drawn. By selecting Outer Wall, first of all its attribute dialog appears with a series of parameters (type, dimensions, colours etc.), which are described in detail within the User’s Guide.

In order to start drawing a wall, you click OK and then follow the instructions shown below:

**Outer wall (straight / arc):** After activating the command (by pressing <Enter> in the menu), you are required to successively provide:

- i) The starting point of the wall (the application message in the command prompt is: *“Wall start \ Relative to wall \ Toggle shape <Linear>”*)
- ii) The ending point of the wall (the application message in the command prompt is *“Wall end \ Relative to wall \ Toggle shape <Linear>”*)
- iii) The direction towards which the wall shall grow, by providing any point on one of the two half-planes defined by the wall line (the application message in the command prompt is *“Enter Side Point”*).



After the above actions, you can see that the wall has been drawn and that you can continue to draw another wall starting from the ending point you defined earlier, unless you right click, which means that you want to stop (or press <ENTER>). You can change the wall drawing from linear into circular, by typing **T** in the following program prompts and pressing <Enter>. During drawing, one can come to the conclusion that the ability of drawing consecutive walls is very convenient since it prevents you from making many movements. As mentioned further below, in the “Element Parameters” section, the thickness of the wall, its height and its level in relation to the floor level (when the level is 0, the wall starts from the floor), are stored within the “Element Parameters” for the wall. By providing proper values for the wall height and level, any possible case of walls of unequal height can be dealt with. The techniques and tools for creating walls are described in detail within the User’s Guide.

Further to the drawing functions, the program also provides powerful editing tools, such as erase, modify (through the wall dialog box), multiple change etc.. Within the User’s Guide there are complete instructions regarding the above commands plus the applicable commands Copy, Stretch, Extend, Trim, Break, Unify, Mirror, Rotate, Scale, Base point. Two other commands that are widely used while drawing the walls are a) the Undo command, which enables you to reverse the previous command executed and b) the Properties command, which enables you to view (and change) the attributes of the selected wall.

### 2.4.3 Drawing Openings

Once the command "Opening" is activated, a second option menu is displayed, including a variety of opening types (window, sliding door, door etc) to draw, plus also a set of editing functions such as "Erase", "Modify" or "Move", applied to existing openings.

**Window:** The option "Window" asks you to select the wall on which the opening will be placed and then define the beginning and the end of the opening (all these actions are carried out using the mouse and pressing <Enter> each time). The window will automatically obtain the data that are predefined in the “Element parameters”, namely the corresponding values for the height, the rise, the coefficient k etc.). Of course, you can draw the window from the ground plan as well as in the three-dimensional (3D) view. During drawing a window, it is very helpful the fact that, after the wall where the window will be automatically placed is selected, the distance from the wall edge is displayed in the co-ordinate’s position on the top of the screen, while the crosshair is transferred parallel to the wall for supervision reasons. The measurement starting point (distance 0) as well as the side (internal or external) is defined by which one of the two edges is closer and which side was "grabbed" during the wall selection. Similar functionality exists for other types of openings, such as Sliding Doors, Doors, Openings etc.. All the details are included within the User’s Guide.

---

#### 2.4.4 Other Entities

AutoBLD provides tools for designing columns and other elements, as well as drawing libraries including drawings and symbols to place within the drawing (i.e. general symbols, furniture, plants etc.). Details are shown within the User Guide of FINE NGAS.

Finally, the Building model of a FINE NGAS project can be viewed through the commands:

- Plan View (2D): The two-dimensional plan view of the respective building level is shown.
- 3D View: A three-dimensional supervision of the ground plan of the current floor (with given viewing angles) is shown.
- Axonometric: Provides three-dimensional supervision of the whole building (for all floors), with the given viewing angles as they have been selected in "Viewing Features".

### 2.5 AutoNET: Network Drawing Principles

The option group AutoNET includes all those tools the designer needs in order to draw (and then calculate) the Gas installation. More specifically, the main AutoNET instructions are described below:

**Drawing Definition:** Layers for each installation are organized properly and the information is shown on the respective dialog. The command "Color" is used to assign the desired color to each network while the command "Linetype" is used to select the desired line type.

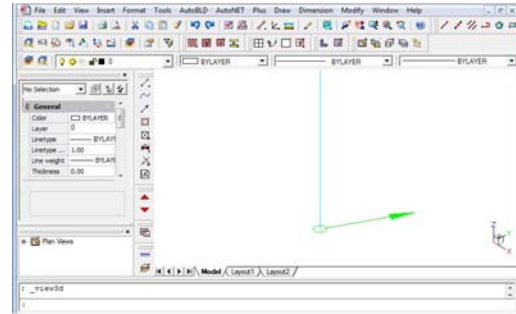
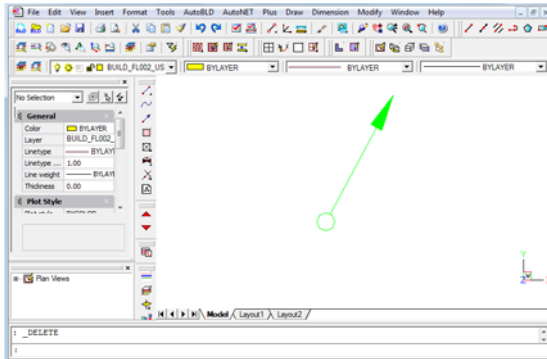
**Copy network of Level:** AutoNET enables copying of typical (installation) plan views and pasting them on other floors through this command, which functions similarly to the "copy level" AutoBLD option.

The basic principles and rules for drawing a network are described below:

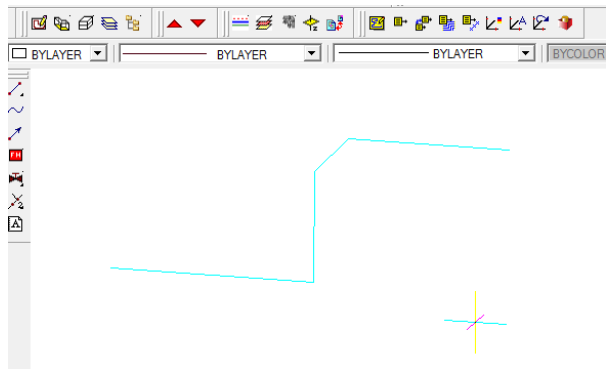
**Network Drawing:** The installation network drawing is carried out with a single line, by drawing lines and connecting them to each other, exactly as the network is connected in fact. The user should keep in mind some general principles regarding drawing and connecting between straight or curved, horizontal or vertical network branches.

**Horizontal & Vertical Piping:** In any case, the piping drawing is carried out exactly as the line drawing (in Autocad or 4MCAD). The user is able to draw horizontal or vertical network branches. Note that vertical branches are different from columns, which will be described below, as they are within the borders of the active floor and do not "cross" floors like columns. The pipe installation elevation is the current elevation. Modification of the pipe installation elevation is possible through the command "elev". If you type "elev" (in the command line), you are prompted to determine the new current elevation. Press <Enter> if it is 0 or type 0 if there is another value but 0. At this point it should be emphasized that, if a horizontal piping which is found on a specific level is drawn and it is connected to another piping or a contact point (receptor), the program automatically "elevates" or "lowers" the pipe so that connecting to the other pipe or receptor, respectively, is possible. In this way, the program facilitates the drawing of piping in three dimensions while the designer is actually working in a two-dimension environment. In any case of a network design, all facilities provided by AutoCAD can be utilized through relative co-ordinates.

**Column Drawing:** Drawing vertical branches which cross floors (one or more) is possible through the option "Main vertical pipes (Building)". When the respective option is selected from the menu, the program asks for the column position ("Enter xy Location") and then for the height of the starting point ("Enter Height for First Point") as well as the height of the ending point ("Enter Height for Second Point"). For example, if you want to draw a vertical branch (column) from 0 to 3, by inserting the location point (XY) and then the numbers 0 and 3 successively, the symbol for direction change appears on the ground plan and in 3D View.



**Vertical sections within the same floor:** If you want to elevate or lower a pipe within the same floor, you can use the relative coordinates. For example, if you have drawn a horizontal pipe (in elevation of 0 m) and you want to elevate it to 2 m, when in the command line asks for "Enter next point", you will type @0,0,2 and continue drawing the pipe (see the adjacent photo). In the same way, if you want to lower the pipe by 2 m, you will type @0,0,-2.



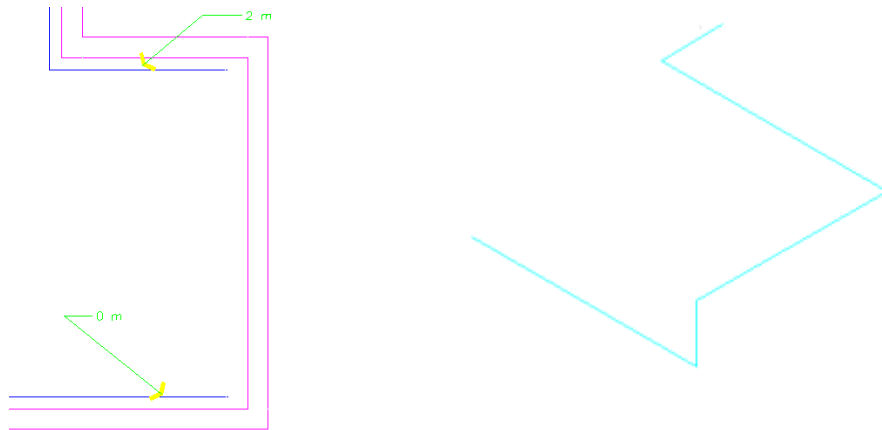
**Drawing of Curved Pipes:** You can draw curved pipes by selecting the command and inserting the points from which the curved pipe is to pass (give at least 3 points). The respective command prompts for the following:

- First point: Insert the starting point of the pipe.
- Next point: Insert next point, the one after that and so on (successively), defining the pipe routing in this way and to stop press <ENTER> or right click of the mouse.

You can easily modify curved pipes using "grips". As soon as the pipe is selected, grips appear which you can move, altering in this way the pipe routing. In the Bill of Materials and the Calculations phase, the program will measure the pipe length precisely.

**Connecting network sections:** Connections between network sections (horizontal, vertical or both) as well as between network parts and appliances can be easily executed by using the "Snap" commands. For example, suppose that the two horizontal parts of the ground plan below, which are placed in different heights, have to be connected. If you start by "grabbing" the "upper" pipe end and then end up at the "lower" pipe end, the result in the three-dimension representation will be as on the right.





**Special Commands for Pipe Construction:** This is actually a set of commands aiming at the facilitated drawing of the installation piping. More specifically, there are two basic commands:

- **Multiple supply pipes:** Multiple supply pipes can be drawn, when the in between distance is known, by simply defining the routing.
- **Pipe parallel to Wall:** A pipe parallel to the wall (walls) that you mark is drawn, with a given distance from the wall, in printing mm (which depends on the printing scale as well). The program asks for the first point and afterwards the wall or the walls (successively) parallel to which (in a certain fixed distance) the pipe is to be drawn.
- **Pipe parallel to Points:** A pipe is drawn parallel to the points you defined (supported by automatic snap), with a given distance from the crooked line defined by these points. The program asks for the first point and then for the other points (successively) parallel to which it is desired to have the pipe drawn. When all points are inserted (and you right click), the distance will be requested.
- **Pipe parallel to Wall (or Points) and Receptor Connection:** This is a particularly useful command similar to the two commands above "Pipe parallel to wall" and "Pipe parallel to points", which, however, enables selecting the receptors to be connected on the routing which will be drawn parallel to the walls or the points. Therefore, it is possible to connect a whole set of appliances to the nearest vertical or horizontal pipe, with 2-3 moves.

For better understanding of the command function, assume that in a given room with its appliances it is desired to install a pipe parallel to the wall and connect the appliances to it. The steps are the following:

- Select the "Pipe parallel to points and receptor connection" command and the following options will appear:
- Select receptors: Select the appliances to be connected to the pipe applied in a parallel arrangement against the wall by defining certain points on the wall.
- Enter the first point & Enter the next point: Provide the points parallel to which you want to install the pipe. The points are shown on the drawing with an X.
- Distance from a point <1.00>: Provide the distance in printing mm where the pipe is going to be drawn starting from the inserted points.

The program draws the pipe and connects it to the appliances.

**Modify an existing network:** You can edit an existing network by using any CAD command (i.e. copy, move or erase etc. of a network section) or utility (i.e. grips) during the design process. The only rules to apply are the following: Pipes supplying the appliances should be connected to the touch points of these receptors. Obviously only one pipe can be connected to a touch point. The connection with the touch points which appear as red "stars" in the ground plan can be executed with the "esnap" function. Piping can be branched to one another and extend in any way as long as they do not form loops, something which does not apply to reality anyway. If however a mistake occurs, the program (during the recognition procedure) will perform all checks and indicate the mistake and its location. A necessary step before the "Network recognition" is defining the point (1) where the network starts, that is the "supply point". In FINE NGAS application, the menu includes the specific options, so that you can be easily guided when drawing any installation.

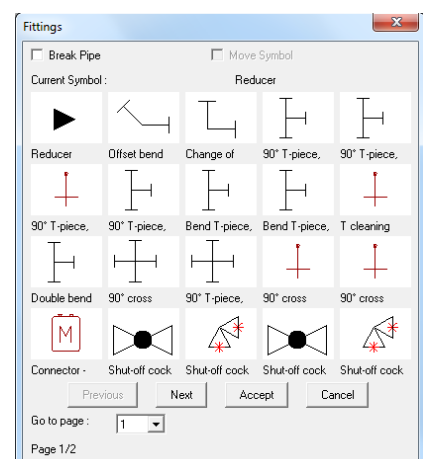
**Placing the receptors:** Placing a receptor can be done simply through the following steps:

- Select a receptor (from AutoNET->Receptors) and press "OK" (or alternatively double click). It can be observed that the receptor moves on the ground plan with the graphic cursor.
- If you move the mouse properly, the receptor can be carried in such a way that its base point (which coincides with the cross of the graphic cursor) can be placed in the desired point. Right click or <ENTER> to confirm your selection.
- If you move the mouse again, the receptor will rotate around the base point. Thus, if you confirm the angle in which you want to place the receptor (again by right clicking), the receptor "freezes" in its final position.

You can also insert and place either the whole receptor or only its touch points in the ground plan by checking one of the two options in the upper part of "Receptors" window. This is important when an existing ground plan includes drawn receptors and there is no need to redraw them, but just move the touch points so that the information for the respective supplies will be available.

Regarding the installation height of a receptor, it should be pointed out that receptors are always installed in the current height. The current height can be changed with the "Set elevation" command (from the PLUS menu).

**Fittings:** The "Fittings" command selects the accessories to be also inserted in the drawings, which applies exactly the same to the receptors. Fittings have "touch points" upon which the piping will be connected so that the network can be identified. A symbol may also have more than one touch points (e.g. a shut-off cock), in which case the accessory will be numbered as a junction point in the "Network Recognition". The program provides the capability of cutting off the line automatically when a symbol is inserted on the line, exactly where the accessory interjects. This capability is defined by the indication of the accessories box "Break Pipe". If this option is activated, then the program will automatically "Break" the pipe when the accessory is placed. Moreover, the "Move Symbol" indication is in the same box, which defines whether the accessory will be moved in relation to the position it was initially placed (so that it will be placed parallel and on top of the pipe) or the pipe will be moved (so that the accessory can be attached).



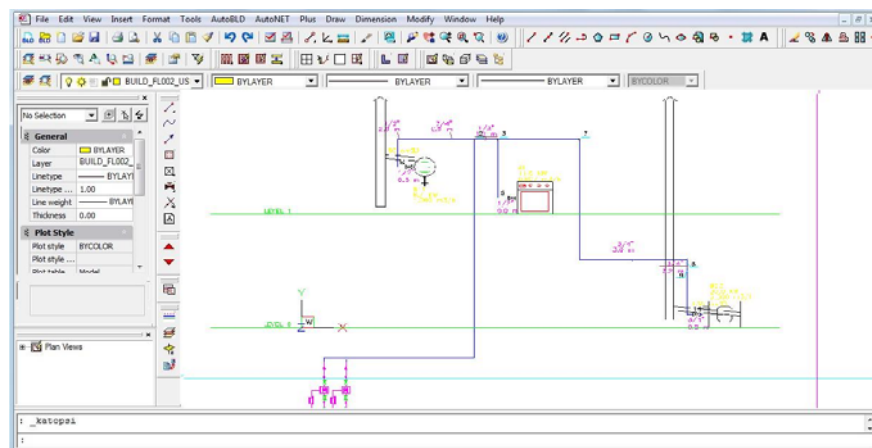
**Symbols:** "Symbols" include various general symbols and other drawings that can be used in the corresponding installation.

**Network Recognition and Numbering:** Since the network has been drawn according to the current rules and the supply point has been determined, the "Network Recognition" option converts the network in the required standard pattern and updates appropriately the calculation sheets. During updating, junction points and receptors are numbered on the ground plan. Note that if a receptor is not numbered, means that the receptor is not connected to the network. Besides, if a network section has a different color it cannot be connected to the network. Connect it or select "Break at selected point" at the connection point with the previous pipe.

**Calculations:** The "Calculations" option leads you in the corresponding calculating environment (ADAPT/FCALC), which means that the window of the current application is "opening", while FINE NGAS remains "open". In order to transfer the data from the drawings, you should select "Update from Drawing" in the "Files" menu of the corresponding calculating application (in order to carry out the corresponding calculations, answer "Yes" to the question "Calculate" that appears). From now on, apply all the capabilities mentioned in the ADAPT/FCALC User's Guide for the respective application. It has to be noticed that the numbering of the sections, the lengths of the network sections, the receptors with their supplies and the accessories (from the piping routing) are transferred in the calculation sheet. Of course, if the user wants to, he can intervene in the calculations in order to make any modifications.

**Legend:** The "Legend" option creates a legend with all the symbols that have been used in this specific project. By selecting it, the program asks for the location where the legend is going to be inserted. Use the mouse to define the location and the legend will appear automatically on your screen, exactly under the location point.

**Vertical Diagram:** This option is used for the automatic creation of the vertical diagram of the installation and in its appearance on the screen, within a few seconds. In case there is already a vertical diagram, the program asks if you want to update it. It is obvious that, in order to create a vertical diagram, you should draw and identify a network and enter the calculation sheets, so that the program knows all the data needed for the vertical diagram creation (pipe dimensions, junction points numbering, etc).



By the "creation" command the window of the vertical diagram's manager appears on screen. This window is composed of two parts, the part with the network tree and the part with the vertical diagram. Through appropriate commands, the user can intervene in several ways on the output of the diagram:

- Enable or disable various branches of the network
- Change the order of the columns of sub-networks in the vertical diagram

- Change the sub-networks direction connection on the vertical columns (right or left)
- Read the information of each node
- Describe the sub-networks

The changes done in the vertical diagram with the help of the above icons are displayed in real time, in the second part of the window. On the upper side of this window there are also icons for processing the diagram (real time zoom and pan, zoom extends etc). In addition, in the upper-left side there are some other icons having to do with the appearance of the screen, such as the hiding of the left part of the window, the appearance of the level names and heights on the left to be edited, the appearance of the numbers of the receptors, the layers and others.

Finally there are some options for the initialization of the vertical diagram, its recreation and the definition of the drawing parameters. In particular, these parameters include the following options:

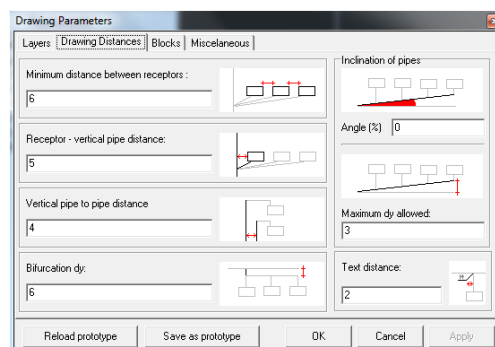
**Layers:** Through a supervisory window table, the user can define the drawing scale, the colors corresponding to the various layers and the height of the texts (in mm drawn on paper) placed on the vertical diagram.

**Drawing dimensions:** The drawing dimensions that will be considered on the creation of the diagram are also defined on mm drawn on paper.

**Blocks:** There can be defined on each application different network starting points and type of tables. The user can choose from a set of dwg drawings.

**Others:** A set of attributes concerning the form of the vertical diagram is defined, such as the condensation of the columns, the number of branches over whom the node is considered as collector, whether the z height information will be considered in the diagram creation and whether the sub-networks pipes on the vertical diagram will be placed over or under the receptors. Finally, it should be mentioned that during the editing procedure concerning the vertical diagram manager, if there are mistakes the program displays the proper messages and warnings.

**Library Management:** The Library Manager leads to a submenu including the options "Numerical data" and "Drawing data". The first option leads to the libraries with all the numerical data of the materials. The "Drawings" option leads to a dialog box where the following data can be seen, regarding each application.



## 2.6 AutoNET: FINE NGAS Installation

The previous chapter described the drawing principles, while this one describes those commands in relation to the special features of FINE NGAS.

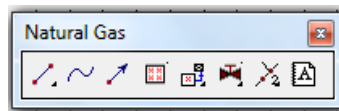
Regardless of the fact if there is an AutoBLD building model, an external reference, a digital image or even no architectural drawings, a gas installation can be drawn and then calculated.

Although there are no limitations regarding the order of actions followed in drawing an installation, the following order is suggested:

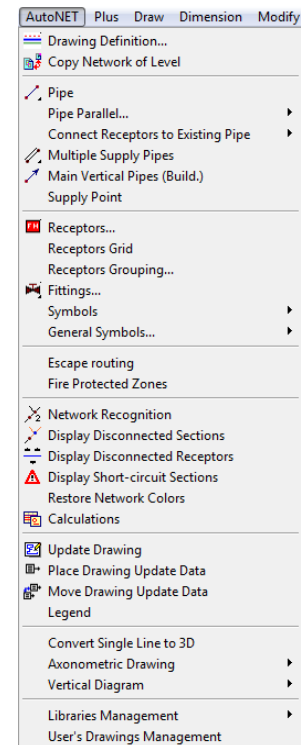
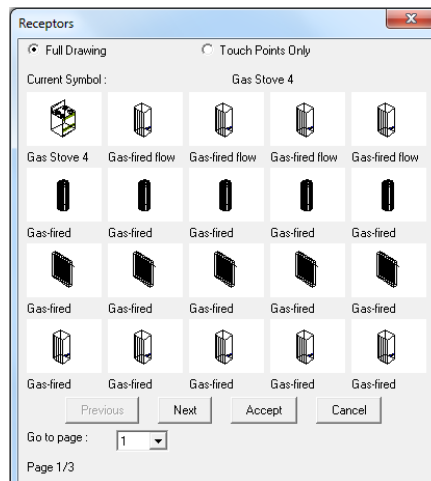
- Place the receptors (Gas stove, gas-fired water heater etc.)

- Draw the horizontal pipes
- Connect the receptors to the pipes
- Draw the vertical pipes (if needed)
- Connect the horizontal to the vertical pipes
- Define the Supply point(s)
- Place the chimney(s)
- Assign the appliances to the chimneys
- Run “Network Recognition”
- If there are no mistake messages, proceed to the Calculations

**Placement of the Receptors to the drawing:** We select “Receptors” either from the AutoNET menu or from the “Natural Gas” toolbar.



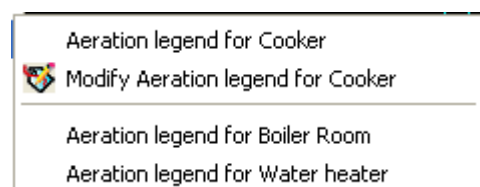
When all receptors’ window appears, select the type of the receptor, which will be placed at the specific point.



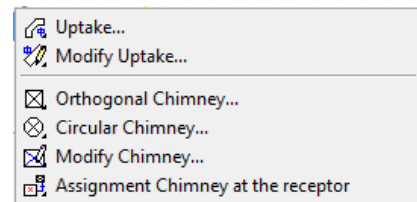
The gas receptors are placed as referred earlier. In case they already exist on the architectural plan view, then we just click at the “Touch points only” field.

**Receptors Data:** This is a useful command that gives you the ability to place easily a legend with information for each receptor. When you select the command, the program asks you to choose the receptor and automatically places its data on the drawing.

**Aeration legend:** With this command you can place an aeration legend for the boiler room, the gas stove and the water heater. In order to place it, first you have to define the space by giving two points, one internal and one external, select the openings of the space (for the boiler room and the water heater) and finally select the location of the legend. Automatically a legend with detailed information of the space is shown and the program checks if the airing requirements are met.

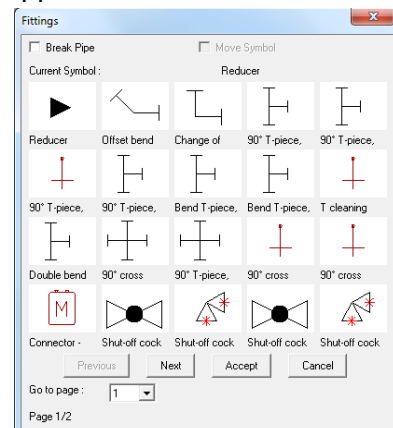


**Chimneys-uptakes:** When the network has been drawn, the chimneys have to be placed in order to be transferred to the calculations. The program provides the following options:



- **Orthogonal/circular chimney:** Select one of the two types, click on the location where the chimney will be placed, define the elevations of the starting and the ending points (e.g. from 3 m to 10 m) and press <Enter>. Automatically the chimney with its number is placed on the drawing.
- **Modify chimney:** If you want to make changes in the chimney, with this command you select the chimney and automatically the data window appears.
- **Assign chimney to the receptor:** In order to assign an appliance to a chimney, select the appliance, press <Enter> and in the command line type the number of the chimney on which will be assigned.
- **Uptake:** If you want, with this command you can design a drawing for the uptake (connecting flue pipe) that connects the appliance to the chimney. Select the command and in the window that opens define the uptake's characteristics (width, hatch etc.) and press either "Points" or "Polyline":
  - **Points:** Click on several points highlighting the shape of the uptake (starting point, middle ones and ending point) and when you press <Enter> define the "side point" which will be on one of the two sides of the line that has been created.
  - **Polyline:** Select the polyline you have already drawn (from Draw > Polyline menu) and when you press <Enter> define the "side point" which will be on one of the two sides of the line that has been created.
- **Modify uptake:** If you want to make changes in the uptake, with this command you select the uptake and automatically the data window appears.

**Placement of the Fittings to the drawing:** We select "Fittings" either from the AutoNet menu or from the "Natural gas" toolbar. In the window that appears, select the type of fitting, which will be placed at the specific point (detailed information in paragraph 2.5).

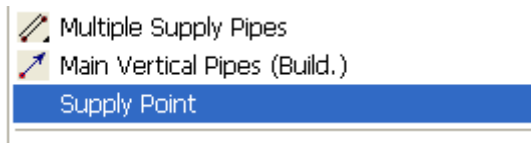


**Horizontal networks design:** After you place the appliances, as described earlier, you proceed with the design of the vertical and horizontal pipes of the level. After the design of the vertical and horizontal pipes, you connect the pipes to the receptors. For the reduction of both the time needed for the specific work and of the errors we suggest the use of the "**Connect receptors to existing pipe**" command. When you select it, the program asks you to define the appliance(s) which will be connected to a specific part of the pipe, then the pipe and when you press <Enter> they are automatically connected.

**Design of the networks' Vertical pipes:** You draw the vertical pipe as it was described in the previous chapter, giving the pipe position, the starting and the ending points. We must point out that the heights of the vertical pipes are dependent upon the level heights of the building that you defined in "Building definition". All pipes will be connected to the vertical pipes through the "Perpendicular" snap point.

**Set of Supply Point(s):** From the edge of the vertical pipes connect the horizontal pipes, and place the supply point to the end of it. It is important to use the **Endpoint** snap to connect the supply point to the horizontal pipe.





**Network recognition:** Select “Network recognition”, in order for AutoNET to recognize the network and prepare the files for the connection with the calculations. During the recognition, messages warn you for potential drawing errors. For example there might be closed routes of the horizontal pipes, points at which different types of pipes end, lack or false placement of the supply point, elements unconnected etc. Besides, no “white” parts should appear on the network, which means that they haven’t been “recognized”. As soon as you correct them, run again the “Network recognition” and if there are no mistake messages, proceed to the calculations.

**Calculations:** When the control is completed we’re ready to proceed to the computing environment. By selecting “Calculations” the Natural Gas calculations are displayed in a new window. When select “Files” and “Update from drawing”, the data is transferred at the calculation sheet.

**Update Drawing:** After the calculation part of the program is completed, save the project file, return to the drawing program (FINE NGAS) and select “Update Drawing”. The following window will open and you will select the information you want to be shown on the drawing. Particularly:

- In the *left part of the window*, you select the information you want to be shown regarding the **pipes**. You can select to see information for all the pipes (choose “Select All”), some of them (choose “Select from Drawing” and select the pipes from the drawing) or none (choose “Deselect All”). Furthermore, below this list, you can choose which information you want to be shown, such as the length, the flow rate, the diameter etc. If you do not want, for example, the “Velocity” to be shown, select it and uncheck the “Selection” button.
- In the *right part of the window*, you select the information you want to be shown regarding the **receptors**. You can select to see information for all the receptors (choose “Select All”), some of them (choose “Select from Drawing” and select the receptors from the drawing) or none (choose “Deselect All”). Furthermore, below this list, you can choose which information you want to be shown, such as the receptor name, the flow rate etc. If you do not want, for example, the “Type” to be shown, select it and uncheck the “Selection” button.

Finally, to place the information on the drawing, select either “Manually placement” or “Auto Placement” (the program automatically chooses to place the information for each pipe and receptor in the best position without covering each other).

**Drawing Update**

Application: Gas ☒ Columns ☒ Vertical Pipes

Level: All ☐ Min Length: 0.20

**Select Network Segments**

Segment	Length	Flow Rate	Diameter	Velocity	Total friction
1 2	2.9 m	1.807 m <sup>3</sup> /h	1/2"	2.583 m/s	0.383 mbar
2 3	1.2 m	0.807 m <sup>3</sup> /h	1/2"	1.153 m/s	0.022 mbar
V 3 4	0.7 m	0.807 m <sup>3</sup> /h	1/2"	1.153 m/s	0.044 mbar
4 5	0.0 m	0.807 m <sup>3</sup> /h	1/2"	1.153 m/s	0.003 mbar
2 6	0.3 m	1.000 m <sup>3</sup> /h	1/2"	1.429 m/s	0.010 mbar
1 7	0.3 m	3.500 m <sup>3</sup> /h	3/4"	2.745 m/s	0.091 mbar
C 7 8	3.0 m	3.500 m <sup>3</sup> /h	3/4"	2.745 m/s	0.384 mbar
8 9	3.2 m	3.500 m <sup>3</sup> /h	3/4"	2.745 m/s	0.372 mbar
V 9 10	0.5 m	3.500 m <sup>3</sup> /h	3/4"	2.745 m/s	0.038 mbar
10 11	0.2 m	3.500 m <sup>3</sup> /h	3/4"	2.745 m/s	0.030 mbar

Select from Drawing< Select All Deselect All

**Select Receptors**

Node	Receptor Name	Thermal Load	Flow Rate	Type	Type
5	Gas Stove 4	11.0 kW	1.3 m <sup>3</sup> /h	H	A1 C
6	Gas-fired flow water heater 8.7 kW	8.7 kW	1.0 m <sup>3</sup> /h	DWH	B11 C2
11	Gas-fired heating boiler 30.0 kW	30.0 kW	3.5 m <sup>3</sup> /h	UWH	B23 C1

Select from Drawing< Select All Deselect All

**Select values to apply**

Order Name Prefix Selection

1 V Length  
2 V Flow Rate  
3 V Diameter  
4 V Velocity  
5 V Total friction

Prefix: << >>

☒ Frame  
☒ Line  
Line Style: PLANVIEW

**Select values to apply**

Order Name Prefix Selection

1 V Receptor Name  
2 V Thermal Load  
3 V Flow Rate  
4 V Type  
5 V Type  
6 V Connected chimney  
7 V Cross section for connecting flue pipe

Prefix: << >>

☒ Frame  
☒ Line  
Line Style: PLANVIEW

**Update**

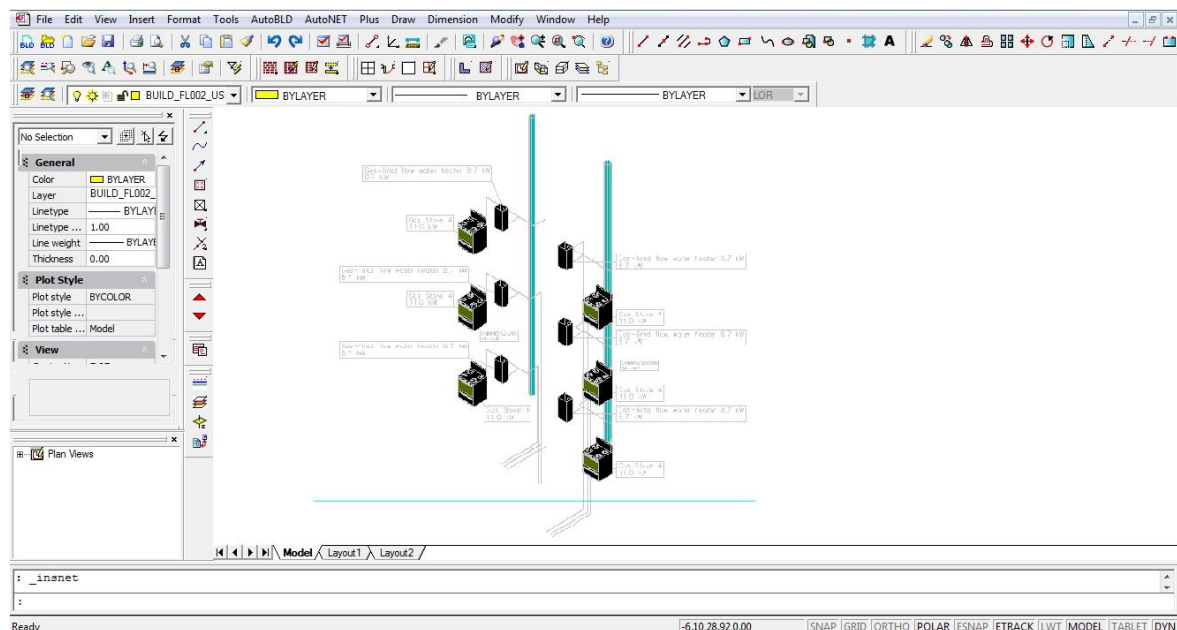
Type: Best Manually Placement Delete Close

Auto Placement Manually Movement

You can easily move the position of the data on the drawing selecting "Move drawing update Data" (from the AutoNET menu) or change their height and style from the "Space text style" command (from the AutoBLD -> Element parameters) and run "Update drawing" again.

**Convert single line to 3D:** This command converts the single line pipes of the drawing to 3D. The diameter of the 3D pipes is directly related to the calculation results and therefore, before you run this command, you have to update the drawing.

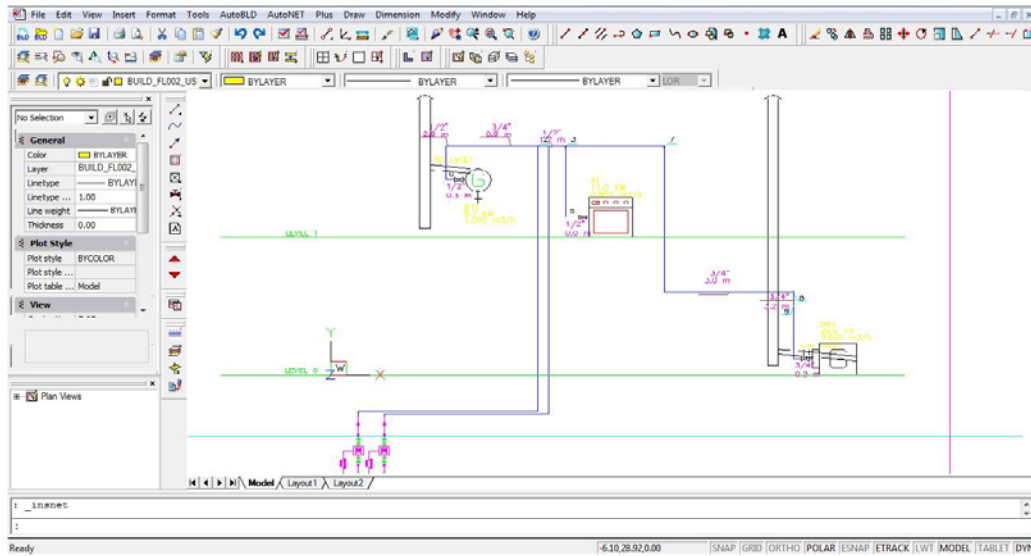
**Axonometric Drawing:** The axonometric drawing is produced automatically with the use of FINE NGAS provided that the network was designed at the specific program. From AutoNET we select "Axonometric Drawing" > "Create". Then the following window appears.



Detailed information for the axonometric diagram parameters are given in paragraph 2.5.



**Vertical Diagram:** The vertical diagram is produced automatically with the use of FINE NGAS provided that the network was designed at the specific program. From AutoNET we select “Vertical diagram” > “Create”. Then the following window appears:



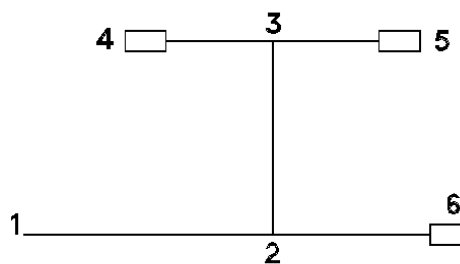
You can apply all the desired changes as it was described in the previous chapter or close the window, which will result to the display of the design in a DWG form, which you can modify using the provisions of CAD programs.





In order to make the network understandable by the program, a specific standardization should be followed, which is more or less the same in all applications. The standardization becomes easily understood with the following simple example.

Suppose we have the network shown in the adjacent figure. This network comprises several branches (i.e. parts of the network), junction points and terminals (end points). Thus in this network, we have assigned arbitrary numbers to both the junction points (1,2,3) and the hydraulic terminals (4,5,6). Each junction point may be assigned a number or a letter (lower or upper case, e.g. A, d etc) or a combination of letters and numbers (e.g. A2, AB, eZ, 2C etc.). The main



logical restriction is that in the starting point is always assigned the number 1. Also, assignment of the same number twice in the same network is not permitted for obvious reasons, with the exception of junction point 1 for which the assignment may be repeated as desired (for networks with more than one starting points). After numbering the junction points and terminals according to the above rule and in order to represent the network in the spread sheet, it is enough to give a name to the various sections of the network entered in the first column of the spread sheet. Having in mind that the order of network sections is not important, we fill in the first column with the two junction points of each section (putting a dot in between) so that the sequence of junction points matches the direction of gas flow in the pipe. In the above example the sections 1.2, 2.3, 2.6, 3.4 and 3.5 should be filled in (order is arbitrary). In some other columns of the row we fill in a series of data (e.g. length of section, fittings included in the section etc.) which depend on the type of installation while the output resulting from calculations and updating the remaining columns depend also on the particular installation.

Taking as a reference point the above spread sheet and ignoring initially the filled in values, we can see the **columns heading zone** (every column has its title and units), **the zone for filling in values** with a number of rows (separated with dotted lines for better supervision and clarity) and a **status bar** where helpful information appear depending on the position in the spread sheet we are in. Since the spread sheet contains usually a lot of information and is the core of the calculations in each application, it is particularly useful to have it maximized on screen by clicking on the upper arrow (located at top right of the window), so that the whole computer screen area is utilized.

The next section will familiarize you with the "Calculation Sheet". The Calculation Sheet provides its user with all the editing functions, which are described below:

First of all, the user has the possibility, as stated earlier, to use **in the frames where the Calculation Sheets appear** the "Font" option for both the calculations zone (so that values appear with the desired size and style) and the headings zone (so that headings are shown to the user satisfaction).

As far as the **headings zone** is concerned, the user can also increase or decrease the column width using the mouse: As long as the mouse pointer rests on the vertical line separating two adjacent columns, it takes the form of a double arrow and then by pressing (and keeping pressed) the left mouse button and dragging, the column width is increased or decreased depending on the direction of mouse movement.

The above alternative supervision possibilities available to the user depend on several factors such as the resolution of the graphics card and screen size, and for this reason any possible interventions are left to the user discretion. For that matter, there is also the possibility of "Load Prototype" from the user. Note however, that best supervision results are achieved with higher resolutions and large screens.

---

Access to the positions of the zone for filling in values is carried out with the use of the mouse and the arrow keys of the keyboard. Moving the mouse pointer in the zone for filling in values we can see that in some columns the pointer takes the form of a vertical line (|) while in other columns it takes the form of a prohibitive traffic sign. We cannot modify the values contained in these last columns (because they result from calculations).

If we move the mouse pointer (having the form of a cross) in to a cell or small square and click the left mouse button, we'll see that the cell contour (outline) becomes dark and we can fill in a value or modify the cell content. In the same way we can move to any other cell, or using the <Enter> key we move to the next cell below and using the <Tab> key we move to the next cell at the right and so on. Besides, in case the window width is not large enough to accommodate all columns, we can review the entire calculation sheet by manipulating it up-down or left-right using the vertically or horizontally sliding keys (potentiometer like). In addition, when access to a column for filling values is denied, the mouse pointer takes the form of a prohibitive traffic sign. This way, the user is informed that the quantity under examination is a derivative one i.e. resulted automatically from calculations.

The user should keep in mind the following useful commands when **entering values** in the Calculation Sheets of any application:

**Deleting cell content:** Pressing the <Del> key on a cell, the value it contains is deleted, and the cell is blank.

**Deleting a row:** Pressing the keys <Ctrl>&<Del> in combination, the row we are in is deleted.

**Inserting a row:** Pressing the keys <Ctrl>&<Ins> in combination, a new (blank) row is inserted immediately below the cell we are in.

**Moving to the beginning of a row:** Pressing the <Home> key, we move automatically to the first column of the row we are in.

**Moving to the end of a row:** Pressing the <End> key, we move automatically to the last column of the row we are in.

**Moving to the upper part of the sheet** (first column-first row): Pressing the keys <Ctrl>&<PgUp> in combination, we automatically move in the first column-first row of the calculation sheet.

**Moving to the lower part of the sheet** (first column-last row): Pressing the keys <Ctrl>&<PgDn> in combination, we automatically move in the last row of the calculation sheet.

Finally, you can move **from an upper to a lower cell using the <Enter> key** and **from a left cell to a right cell using the <Tab> key**.

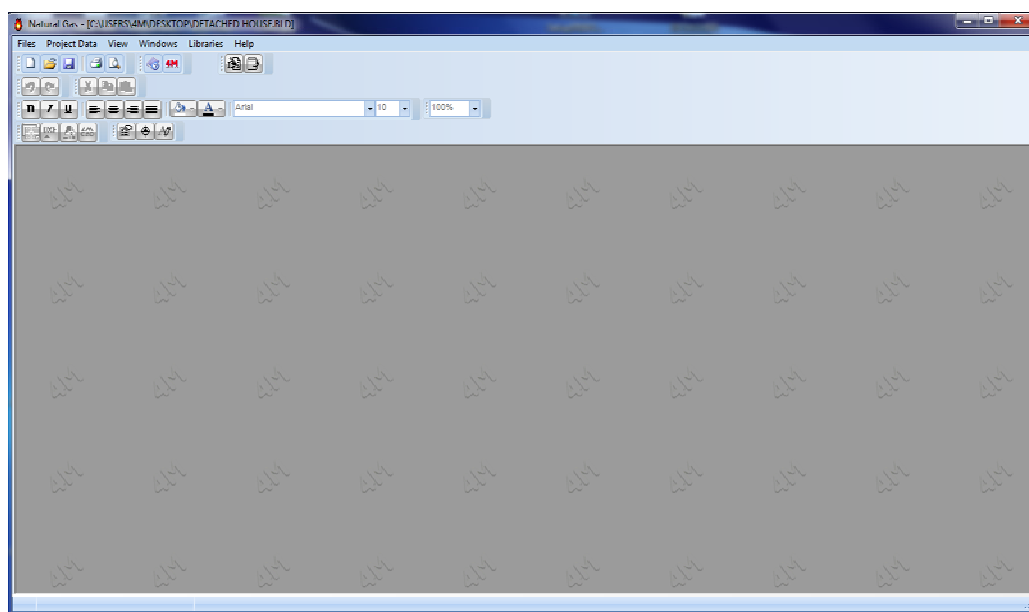
In addition, the calculation sheet provides the user with a **set of Spreadsheet Functions, which are available in most windows applications**, such as the Cut-Copy-Paste type of commands of a subset of lines (or even the whole calculation sheet), the row and columns width definition, the font type (as well as font attributes, justification etc) of a selected area, and so on. By selecting a certain area of the sheet (or all of it by "select all") and then pressing the right button of the mouse, a small menu appears on screen, with the relative commands. Another useful command is the Undo/Redo command concerning the calculations. All those editing commands are also applied to other windows. Apart from the copy-paste command, in case we want to repeat a row (typical branch), it is sufficient to fill in the content of the first column, i.e. the section name, that will make a copy of the row except for the section name which remains blank.

---

When the calculation sheet is activated, you will see in the main menu options an additional one namely "Calculation Sheet" with a secondary option "Printing Parameters". Selecting "Printing Parameters" the adjacent dialog box appears from where the user may affect the appearance of the printed Calculation Sheet. Specifically, the user may define a bold outline (frame), a normal outline, or no outline, horizontal and/or vertical lines, as well as a raster for the titles (headings) of the spread sheet with the desired shading of tints (using the sliding key). As previously emphasized, the Calculation Sheet window is the core for all applications. Since, however, not all calculation results related to a study can be confined within the Calculation Sheet, every application has additional windows where these complementary results are accommodated to form the complete set of the study. The advisability and functional description of these windows is the subject matter of each application. For all that, we can pick out, among the available windows, some of them with common philosophy regardless of application (e.g. "Bill of Material – Costing" Window, "Technical Description" Window etc.). The "forms" of these windows are described in the following sections not necessarily in the order found in the applications.

## 3.2 Main Menu

If you want the GAS network application to be executed, point with the mouse and double click on the relevant icon, and the main menu window will appear:



As you can see, the basic menu options are divided into the groups **"Files"**, **"Project Data"**, **"View"**, **"Windows"**, **"Libraries"** and **"Help"**, which are described below along with their secondary options.

## 3.3 Files

The "Files" option includes the usual file management options according to the windows standards:

**New project:** Type a name for the new project you want to be saved in a file.

**Select project:** A window appears where you can select the desired (existing) project file and load it.

**Caution!** *If neither a new nor an existing project is selected, the programme automatically considers that the UNNAMED project is active. If you add new data to the UNNAMED project and you want to save it with a different name, select "Save as" and type the new project name.*

**Update from Drawing:** In the case of co-operation with the FINE package, the project calculation sheets are updated with the drawing data.

**Caution!** *If the option "Update from Drawing" is selected, without previously opening a project and inserting rooms in the ground-plans using the FINE package, any existing data in the calculation sheets will be replaced with blanks.*

**Save Project:** The project you are currently working on is saved to the hard disc (with the previously given name).

**Save Project as:** The project you are currently working on is saved in a different file with a new name.

**Load Prototype:** The saved prototype appears on the screen.

**Save as Prototype:** The form, which has been created by the user and is displayed on the screen when this option is selected, is saved as a Prototype.

**Printing Prototypes:** The printing prototype management window is activated.

**Printing:** The project issue is printed according to the previously selected options in "Printing Contents" and "Printing Parameters" as well as according to the print preview output.

**Printing Contents:** You can select the project items you want to be printed:

**Printing Parameters:** The desired printing parameters can be selected in this window.

**Print Preview:** The complete project issue appears on the screen, exactly as it will be printed, page to page.

**Export to RTF:** An rtf file, containing the project items, is created (within the project directory, with the name KAER.RTF).

**Link to WORD:** A .doc file, containing the project items, is created (within the project directory, with the name KAER.DOC). At the same time, the MS-Word application is activated (if it is installed in your PC).

**Link to 4M editor:** An rtf file, containing the project items, is created (within the project directory, with the name KAER.RTF). At the same time, the 4M text editor is activated for further editing.

**Link to Excel:** An xls file, containing the project items, is created (within the project directory, with the name KAER.xls).

**Export to PDF:** A pdf file, containing the project items, is created (within the project directory, with the name KAER.pdf).

**Exit:** Exit from the application.

## 3.4 Data

This is about the basic project data, which are divided into "Project Info" (project headings) and "Network" data. The "Project info" refers to titles and headings related to the project identity, while the "Network" option refers to the general network data that the project designer should specify and are related to:

Field	Value
Equipment Schedules GI Type	1
Gas category	Category H
Main Pipe Type	Steel Pipe
Main pipe Standards	EN10255
Main Pipe Roughness Factor (µm)	500
Secondary Pipe Type	Polyethylene Piping
Secondary pipe Standards	prEN 1555-1
Secondary Pipe Roughness Factor (µm)	15
Max Air Velocity (m/s)	3
Number of Chimneys (1 - 20)	2
Operating Pressure (BAR)	0.02
Operating Temperature (°C)	15
Building geodetic height in relation to sea level	0.00
Import data from the Vertical Diagram	<input type="checkbox"/>

**Equipment schedules GI type:** The equipment schedule of the appliances of GI type is entered in this field. The “GI” refers to gas-fired appliances that are commercially or industrially used.

**Gas Category:** The gas category corresponding to the specific project should be filled. These are either earth natural gases, synthetic gases (SNG= Synthetic Natural Gas), or their alternative gases. They are divided according to the range width of the Wobbe index in the families L, H and E.

**Main Pipe Type:** With the key <F11> or by pressing the key in the field, you can select from the pipe library window that appears, the pipe type which shall be used in the project (e.g. Copper pipe)

**Main Pipe Standards:** The standard corresponded to the main pipe must be defined.

**Main Pipe Roughness Factor (µm):** The above selected pipe roughness appears automatically (µm) but it can be modified from this field.

**Secondary Pipe Type:** The type of the secondary pipe should be selected, following the instructions provided above for the main pipe type.

**Secondary Pipe Standards:** The standard corresponded to the secondary pipe must be defined.

**Secondary Pipe Roughness Factor (µm):** The above selected pipe roughness appears automatically (µm) but it can be modified from this field.

**Maximum air velocity:** It is the maximum air velocity based on which the cross-sections are calculated. That is to say the smallest possible cross-section, for which the velocity does not exceed this value, shall be selected. This value can be modified here as a whole from the user (and it shall be applicable in every part of the network), and also within the calculations sheet, in the “Max velocity” column.

**Number of chimneys:** The total number of chimneys should be filled. The user can fill from just one (1) up to twenty (20) chimneys in a project.

**Operation Pressure:** The operation pressure of the network should be filled with maximum value of 0,1 bar (100 mbar). Usually the default value is equal to 0,02 bar.



---

**Operating Temperature:** The operation temperature of the network should be filled in °C. The default value is equal to 15 °C.

**Building height above sea level:** This height in meters should be filled with respect to the sea level and is necessary for the chimney calculation according to EN 13384.01 standard.

**Import data from the Vertical Diagram:** This option enables you to draw the vertical chart firstly with the aid of an expert system and thereafter to transfer the data in the calculation sheet. If the check box is checked, the option "Insert vertical" appears in the program menu and you can use it in order to draw the vertical chart.

## 3.5 Vertical Diagram entry

This option includes the two secondary options "Vertical Diagram Creation" and "Update from vertical Diagram", which are explained in the following sections.

### 3.5.1 Vertical Diagram Creation

This option has the following secondary options:

#### 1. Diagram type

- **Receptors > Transfer as simple receptors**

It is specified whether each receptor is defined separately in the vertical diagram or if it will be integrated in a receptor group. In case that the check box is checked, the network is analyzed thoroughly in nodes.

#### 2. Building parameters

If you select the building in the upper left part of the screen, the building data will appear.

The topology of the building as well as the layout of the installation network in it is included in the building data. In the box "Building options" the levels with gas consumption are checked. In particular, if you select the ground floor and the floors, an additional field will appear where the user should fill in their number. You can also define the number of the apartments in each floor as well as the height of each floor.

After the user has defined the building data, he can press the key "Apply" and the building will appear in the left column in a tree form with an icon in front of it that shows the floor type.

#### **Working with levels**

If you click with the left mouse key on the level name, the level data "Level Name", "Level Height (m)" and "Number of Apartments" appear on the right side. The user can enter the fields "Level Name" and "Level Height (m)" and edit the level data.

If you double click with the left mouse key on the level name, the floor topology with the apartments will appear in tree form.

If you click with the right mouse key on the floor name, a menu with the following options will appear:

- **Insert new Apartment:** You can insert a new apartment besides the ones that are defined so far.
- **Copy FLOOR:** Copies in the PC memory (clipboard) the floor network in order to paste (copy) it later on a different floor.

- 
- **Paste FLOOR:** Copies the network data that are saved in the clipboard to the selected floor. The data that are already entered in the floor shall be overwritten by the new data.

### **Working with apartments**

If you double click on the floor the apartments will appear and in the right side will appear the option “Insert space with Gas”, where a list with various room types will be shown. If you double click on each room the room drawing will appear underneath it. If you double click on the desired room, this will be transferred automatically in the apartment selected in the left column. In the same way you can add more rooms in the same apartment or in different apartments.

If you right click on an apartment room the list shown in the next window will appear, with which you can manage the rooms:

- The command “Delete“, deletes a room.
- The commands “Move upwards” and “Move downwards” alter the order of the room inside the apartment.
- If you click with the left mouse key on an apartment room the room drawing will be shown enlarged.

### **3. Network data**

In the option connections you can see the network structure from the gas network until each apartment. In this option, the user can manage the gas consumptions. At first, there is a separate gas installation for every apartment. If for example, there are two apartments in the building with common ownership, which are located in different floors, then the user can transfer the gas network from a column to a different one, to delete a column or a certain flow gauge. This is achieved by right clicking on an element. Depending on the "tree" element on the left side of the window that you right click, a menu appears which enables you to move or delete or even add an element. Hence, the user is able to modify the network as it is desired.

### **4. Diagram**

With this option, the vertical chart of the previously mentioned installation appears. Moreover, you can change the colors that appear in the chart.

#### **3.5.2 Update from vertical diagram**

This command is used to transfer the network data in the vertical chart in the “Calculation sheet”.

### **3.6 View**

This option includes the secondary option "Toolbars" and follows in general the windows standards.

### **3.7 Windows**


The option “Windows” includes a series of calculation and result windows, in which the analytical project calculations are presented. The main window which comprises the core of the application calculations is the Calculation Sheet, and is described in the following section.

---

### 3.7.1 Cover Page

The “Cover page” window is the first printed page of the project and you can select among different types of cover pages or even create your own, exactly as you want it.

To load or create a different cover page, in the program menu go to Cover page -> Prototypes, select the prototype you want and press “Load”. If you want to save the cover page you made and use it in other projects, in the same window enter a name and a number (e.g. 02 Simple Alignment) and choose “Save as”.

 **Note:** The cover page files are stored in \CALC\KAERI\ , named KAERCV01.RTF, KAERCV02.RTF, etc; their descriptions are stored in the file KAERCV.LST.

### 3.7.2 Assumptions

The text of the project general assumptions, which may be included in the printed project as long as it is selected in "Printing Contents", is stated. If the option “Assumptions” is selected, you can load a different prototype or create your own from the program menu Assumptions -> Prototypes.

### 3.7.3 Systems of Appliances

Here the receptor systems that are used in the specific project are presented and analyzed in the receptors constituting them.

### 3.7.4 Receptors Legend

The receptors used in the project are presented.

### 3.7.5 Fitting systems

Here the fittings systems that are used in the specific project are presented and analyzed in the fittings constituting them.

### 3.7.6 Legend of fittings

The fittings used in the project are presented.

### 3.7.7 Calculation Sheet

The Calculation Sheet of the GAS network is the core of the calculations for the Gas installation application and conforms to the general rules of the Network Calculation Sheets, which are described in the first section. Therefore, each row of this sheet corresponds to a different network section while each column refers to the data that will be filled-in or will ensue automatically during the procedure of completing data. Help instructions concerning data entering appear at the bottom of the screen (status bar). In each row, the fields of the first column, which refer to section designations, should be filled-in first of all.

The method for the network standardization is based exactly on the standardization rules that were explained earlier. Here is a short description of the columns of the calculation sheet:

Network Section	Pipe Length m	Type of Appliance	Appliance flow rate m³/h	Peak Capacity (m³/h)	Desired Pipe Size mm	Pipe Size mm	Max Velocity m/s	Gas Velocity m/s	Type of Fittings	Fittings Friction losses mbar	Levitation Friction losses mbar	Pipe Friction losses per meter mbar/m	Pipes Friction losses mbar	Total Friction Loss mbar	Permissible Segment Pressure loss mbar	Total friction loss of the line (mbar)	Permissible Network Pressure loss mbar	Polar Angle F	Pipe passing
1	1.2	2.9	2.300	1.807	1/2"	3	2.583	F-1	0.074			0.107	0.309	0.383		0.383	2		
2	2.3	1.2	1.300	0.807	1/2"	3	1.153	3	0.004			0.015	0.018	0.022		0.405	2		
3	3.4	0.7	1.300	0.807	1/2"	3	1.153	3	0.004	0.030		0.016	0.011	0.044		0.449	2		Descend
4	4.5	0.0	1.300	0.807	1/2"	3	1.153	19	0.003				0.000	0.003		0.452	2		
5	2.6	0.3	1.000	1.000	1/2"	3	1.429	19	0.004			0.020	0.006	0.010		0.393	2		2
6	1.7	0.9	3.500	3.500	3/4"	3	2.745	3	0.020			0.079	0.071	0.091		0.091	2		
7	7.8	3.0	3.500	3.500	3/4"	3	2.745	3	0.020	0.129		0.078	0.235	0.384		0.475	2		Descend
8	8.9	3.2	3.500	3.500	3/4"	3	2.745	F-2	0.121			0.078	0.251	0.372		0.847	2		
9	9.10	0.5	3.500	3.500	3/4"	3	2.745	3	0.020	-0.021		0.078	0.039	0.038		0.885	2		Upward
10	10.11	0.2	3.500	3.500	3/4"	3	2.745	19	0.014			0.080	0.016	0.030		0.915	2		1
11																			

**1. Network section:** In the rows of the first column you should fill-in all the network sections (one section in each row) one by one. The sections are defined by their end nodes. You may assign a number (from 1 to 9999) or a letter (lower or upper case, e.g. A2, AB, 3c, Aa etc.) to each node. The basic restriction in numbering is that number 1 is always assigned to the point that is connected to the gas meter. Except the number "1", the same number should not appear in the network twice. After the numbering you can enter in the calculation sheet all the sections independently one by one (the sequence of the sections is not important), by filling-in the first column:

In the case of typical (similar) sections it is possible to recall them (with their name from the first column) in order to automatically transfer them.

**2. Pipe length:** You must state the pipe length in (m) in every section between two nodes (e.g. section 2.3).

**3. Type of appliance:** If there is an appliance at the specific network section, that is to say if the section ends up to an appliance, you can select it from the library with the GAS installation appliances, after pressing the key **F11** or by pressing the key inside the field in this column. A window will appear with the list of the appliances.

No	Name	In.Diam.a	Qa	In.Diam.b	Qb	Number
1	Gas Stove 4	13	1.5	13	1.3	
2	Gas-fired flow v	13	1.2	13	1.0	
3	Gas-fired flow v	13	2.4	13	2.0	
4	Gas-fired flow v	13	3.2	13	2.6	
5	Gas-fired flow v	13	3.9	13	3.2	
6	Gas-fired stora	13	0.9	13	0.8	
7	Gas-fired stora	13	1.0	13	0.9	
8	Gas-fired stora	13	1.1	13	1.0	
9	Gas-fired stora	13	5	13	1.0	
10	Gas-fired stora	13	1.4	13	1.2	

The type of the appliance (hydraulic receptor) is defined when a unit is filled-in in the last column, in the row that corresponds to the receptor, and the key "enter" is pressed after that in order to validate the data entered. By pressing "OK" you can return to the calculation sheet where you can notice that the order number of the appliance is filled-in. Alternatively, except defining only one appliance, you can also define an appliance system with up to 10 different types of appliances in each section.

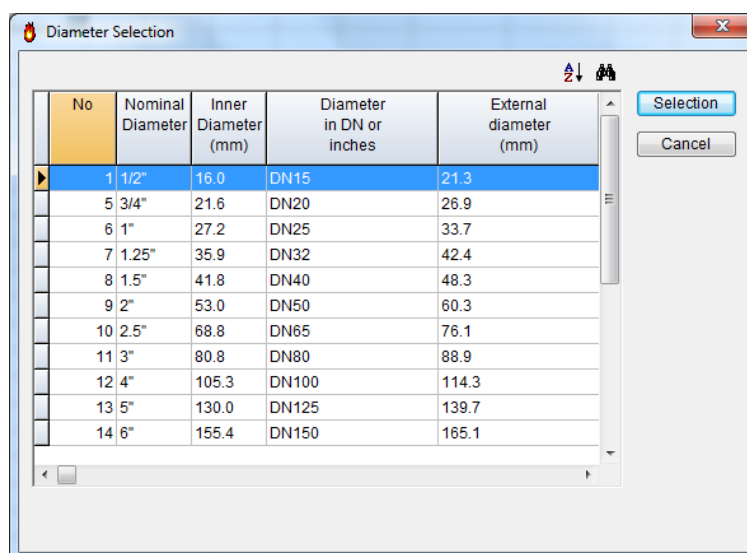
Since you are working with systems, the program enables you to define an order number of the hydraulic appliances system  $i$  in the upper part, and the relevant System appears in the column with the form S- $i$ , where  $i$  represents the order number of the defined System.

With this feature you avoid filling-in again the same appliances, since you can fill-in directly the order number of the System. The method with appliance systems is indicated in the case of very large installations, where the appliances are grouped in a few systems and the data volume is radically decreased in the calculation sheet.

**4. Appliance flow rate ( $\text{m}^3/\text{h}$ ):** The appliance flow rate  $Q_R$ , or in general the network section flow rate (theoretical sum of the flow rate if all the appliances work at the same time) is automatically calculated. The program calculates automatically the flow rate of the intermediate sections based on the flow rate of the appliances that are supplied from this section.

**5. Peak capacity ( $\text{m}^3/\text{h}$ ):** The peak capacity  $Q_s$  is calculated based on the total capacity of the previous column. The peak capacity is calculated from the relevant peak curve that depends on the equipment schedule which has been initially selected by the user for the GI type of appliances, or it is loaded automatically for all the other types.

**6. Desired pipe size (mm):** From this column the user can select a different diameter from the one that is calculated by the program (and is shown in the next column). If you press **F11** or the key in the field, the list with the diameters of the selected in the "Network Data" pipe type is shown.



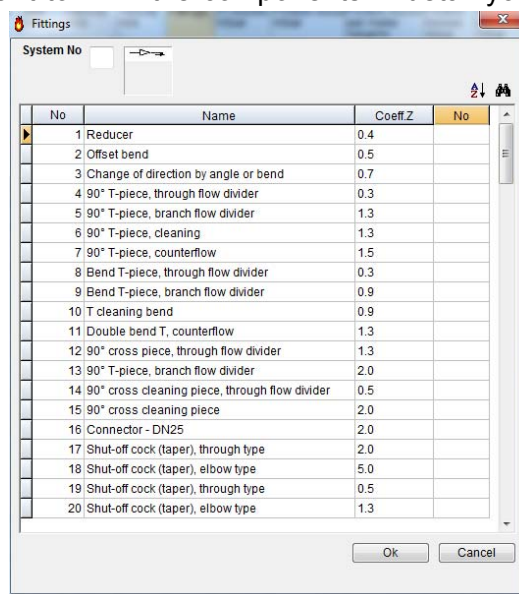
With the option "Select" you can select the desirable diameter, and you can see the effects of your choice in the rest of the network parameters (e.g. velocity, friction values etc). If you press <Del> in the relevant component of the calculation sheet you can delete the desired diameter, and the program calculates again the diameter.

**7. Pipe size (mm):** The pipe size of the network section, as it is calculated by the program, is shown in this column.

**8. Maximum velocity (m/s):** The velocity limit that is defined in the "Network" (in m/s), which you can modify if you want for the particular network section, is shown here. You must keep in mind that, if you change later the general limit in the "Network" this will not affect the modified values, but only the initial ones (the ones that have the same value with the "Maximum gas velocity" which is defined in the "Network").

**9. Gas velocity:** The gas velocity in (m/s) as it is calculated in the specific network section is shown here.

**10. Type of fittings:** This column refers to the type of fittings (elbows, tee sections, valves etc) that appear in the network section. If you want to fill-in the components in detail you press F11 or the key in the field in this column, and the window with the list of the gas system components from the relevant library will appear. If only one component exists, this is defined by writing the number 1 in the last column, in the row that corresponds to the component. By pressing "OK" you can return to the calculations sheet where you can notice that the order number of the defined component is filled-in.



For more than one component you can fill-in correspondingly the last column of the table with components, giving also the quantities of each component (up to 10 different types of components per section). In the case that you have more than one component, the indication F- appears in the column, which means in general "Fittings". If, in the upper part of the fittings table, you define a fitting system number a, the indication F-a will appear in the column. In this way you can group the fittings and also avoid filling-in the same groups (systems), since you can fill-in directly the fitting system number.

In this way you can group the fittings and also avoid filling-in the same groups (systems), since you can fill-in directly the fitting system number.

**11. Fittings friction losses (mbar):** The calculated fitting friction loss for the specific network section in mbar is shown here.

**12. Levitation friction losses (mbar):** The calculated friction loss caused by the gas levitation is shown here for the specific network section in mbar. In case of upward sections (segments) the value is negative, whereas for downward sections is positive. For horizontal sections it is equal to zero. The direction of a section is being defined in the column "pipe route".

**13. Piping friction losses per meter (mbar/m):** The calculated friction loss for the piping in the specific network section in mbar/m is shown here. This friction loss is calculated from the gas flow in the respected section, based on hydraulic calculations.

**14. Pipes friction losses (mbar):** The calculated friction loss for the entire pipe in the specific network section in mbar is shown here. This friction loss is calculated from the gas flow in the section, based on hydraulic calculations.

**15. Total friction loss (mbar):** The total friction loss in the section, that is to say the sum of the Friction of Fittings and the Friction of Piping, also in mbars is shown here.

**16. Segment permissible pressure loss (mbar):** This value depends on the specific standards adopted.

**17. Total friction loss of the line (mbar):** The respective calculated value is shown.

**18. Network permissible pressure loss (mbar):** This value depends on the specific standards adopted.

Following you will see the items "Polar angle", "Pipe route", "Chimney number", "Connecting flue pipe type", "Connecting flue pipe cross section". These items can be filled-in directly in the relevant columns, or from an additional window, which appears when you press the key F12 in any column or if you press the right mouse button and select from the list "Options" that appears. The use of these columns is explained below:

---

**19. Polar Angle:** It is necessary to fill-in the polar angle of the network only in the case that you desire to draw a vertical diagram (and a rough drawing) from the calculation sheet, that is to say in the case that you have not drawn the ground plans with the FINE NGAS application. The vertical diagram that is drawn takes into consideration the length and the polar angle (in relation to the horizontal axis) of each branch.

**20. Pipe passing (upwards - descending):** This field is empty by default and should be filled properly, or to be left empty in case of a horizontal branch.

**21. Chimney number:** The number of the Chimney to which the connection pipe (if any) is being directed should be filled.

**22. Connecting flue pipe type:** If a connecting flue pipe is required at the specific section, we need to fill its geometrical type (i.e. circular, square etc).

**23. Connecting flue pipe cross section (cm<sup>2</sup>):** The calculated cross section of the connecting flue pipe is shown in this column. The user can easily change it by pressing the key F12 and filling in the "Desired cross section for connecting flue pipes" field.

**24. Coincidence GI:** The default value of the equipment schedule that was defined in the "Network" window can be modified here for each specific section. The "GI" refers to gas-fired appliances that are commercially or industrially used.

**25. Type of pipe:** The user can define whether this branch refers to a main or secondary pipe (in case there are two different types in the same network). The user defines the types of the main and the secondary pipes from the "Project Data" > "Network" window.

**26. Pipe design length (m):** Sometimes, when drawing a vertical diagram, you may design very short or very long sections. For that reason, you can define an arbitrary pipe length in this column, which however can result to the desired **aesthetical** aspect in his design. Regardless of the length that you define in this field, the program shall write the **real pipe length** on the drawing. It should be pointed out that in case you do not fill-in this field, the program shall draw the vertical diagram based on the column "Pipe Length" of the Calculation Sheet.

**27. Floor of appliance:** The number of level is filled here automatically and updates the "technical report" of the project.

**28. Main gas stopcock:** Depending on the value ("Yes" or "No") the main gas stopcock is shown at the axonometric drawing.

**29. Passing through the soil:** The user fills "Yes" or "No" depending on the case. If the field remains empty the program assumes that the passage is out of the ground. This information is used for the update of the technical description.

**30. T part exists:** Depending on the value "Yes" or "No" the respective fitting appears on the axonometric drawing. For instance, if we define this fitting on the 2.3 section, then it will appear on the node "2".

#### ***Fields Calculated***

Fields corresponding to results calculated by the program are the following:

**31. Fittings Sz:** The total sum of the local z resistances is being shown.

**32. DH Elevation difference (m):** The height difference is shown (in m) with a positive prefix for ascending network sections and negative prefix for the descending sections.

**33. Number of H appliances** (gas stoves, gas cookers, gas stove tops, gas ovens and the cooking and baking part of gas ranges).

**34. H appliances gas flow rate (m<sup>3</sup>/h)**

- 
- 35. Coincidence Factor H**
  - 36. Gas flow rate multiplied by concurrency H appliances**
  - 37. Number of DWH appliances** (gas flow water heaters).
  - 38. DWH appliances gas flow rate (m<sup>3</sup>/h)**
  - 39. Coincidence Factor DWH**
  - 40. Gas flow rate multiplied by concurrency DWH appliances**
  - 41. Number of RH appliances** (gas space heaters, gas storage water heaters and the heater part of gas ranges).
  - 42. RH appliances gas flow rate (m<sup>3</sup>/h)**
  - 43. Coincidence Factor RH**
  - 44. Gas flow rate multiplied by concurrency RH appliances**
  - 45. Number of UWH appliances** (gas circulating water heaters, gas combi water heaters and gas heating boilers where  $Q \leq 30$  kW).
  - 46. UWH appliances gas flow rate (m<sup>3</sup>/h)**
  - 47. Coincidence Factor UWH**
  - 48. Gas flow rate multiplied by concurrency UWH appliances**
  - 49. Number of GI appliances** (commercial or industrial gas-fired appliances and central hot water heaters and heating systems in conjunction with gas heating boilers where  $Q > 30$  kW).
  - 50. GI appliances gas flow rate (m<sup>3</sup>/h)**
  - 51. Coincidence Factor GI**
  - 52. Gas flow rate multiplied by concurrency GI appliances**

Summarizing, if you follow the above standardization and fill-in the data for all the network sections, then the flow rates in the sections without appliances are summed and are presented automatically in the column for the flow rate. Based on the peak capacity in each network section and given the maximum velocity that corresponds to this section, the cross-section of the section pipe is established. Despite all this, you can select another standardized diameter from the list of standardized diameters of the library that appears on the screen. In whatever way the section has been defined, the effective gas velocity and the friction losses of the piping and components in the respective section of the network will be exactly calculated.

**Fixed column:** Moreover, through the "Calculation Sheet" menu, you can select the "Fixed Column" so that the first column of the sheet "freezes" on the screen. Thus, as you fill in the fields of the sheet and you are "led" towards the right, you will know at any time on which network section you is working. When the "Fixed Column" option is active, a yellow point (pin) appears at the bottom of the "frozen" field.

### 3.7.8 Chimney calculations

This window presents the results of the calculations concerning the chimney (or chimneys) included in the project.



Chimney No	
Appliances of chimney gas flow rate	2.40
Nominal Power of appliances of the chimney (kW)	20.70
Number of chimney appliances	3
Type of appliances in chimney	no fan
Minimum hydraulic diameter required for the chimney (cm)	12.50
Chimney cross section area type	Square
Length or diameter of the selected chimney (cm)	14
Width of the selected chimney (cm)	
Hydraulic diameter of the selected chimney (cm)	14.00
Wind velocity pressure (Pa)	0 Pa (proptious conditions outside of reverse pressure zone)
External air temperature TL (°C)	15.000
Mean ambient air temperature of the chimney Tu (°C)	0.000
Ambient air temperature at the chimney outlet Tuo (°C)	0.000
Flow safety coefficient SE	1.5
Additional data for the connecting flue pipe	
Connecting flue pipe 1 temperature for temperature instability (°C)	121.000
Connecting flue pipe 2 temperature for temperature instability (°C)	121.000
Connecting flue pipe 3 temperature for temperature instability (°C)	121.000
Connecting flue pipe 4 temperature for temperature instability (°C)	
Connecting flue pipe 5 temperature for temperature instability (°C)	
Additional data for the chimney	
Extended length of chimney (m)	1.4
Chimney effective height (m)	12
Sum of local resistivities of the chimney Sz	1.150
Chimney Thermal resistance (1/L)v (m2KW)	0.564
External heat transfer coefficient aa (W/m2K)	23.000
Chimney Draughtness factor z (m)	0.004

**Chimney no** = In this field the user types the number of the chimney which will be calculated (in case there is more than one chimney).

**Appliances of chimney gas flow rate** = The total flow rate of the appliances that are connected to the chimney is presented in m<sup>3</sup>/h.

**Nominal power of appliances of the chimney** = The total nominal power of the appliances that are connected to the chimney is presented in KW.

**Number of chimney appliances** = The total number of the appliances that are connected to the chimney is presented in this field.

**Type of appliances in chimney** = In this field the type of the appliances that are connected to the chimney is presented. The type is loaded automatically from the Libraries > Appliances.

**Minimum hydraulic diameter required** = When all the fields of the “Chimney calculations” window have been filled in correctly, the minimum hydraulic diameter of the chimney (in cm) is calculated automatically. In case a necessary field of this window has been left blank, in this field an appropriate message will be shown.

**Chimney cross sectional area type** = select the type of the chimney (square, circular etc.)

**Length or diameter of the selected chimney (cm)** = When the minimum diameter of the chimney has been calculated, fill in this field with the length (in case of square or orthogonal) or diameter (in case of circular) of the chimney to be placed. The diameter must be equal or higher than the calculated minimum diameter.

**Width of the selected chimney (cm)** = When the minimum diameter of the chimney has been calculated, fill in this field with the width of the chimney to be placed (in case of square or orthogonal). The width must be equal or higher than the calculated minimum diameter.

---

**Hydraulic diameter of the selected chimney** = The diameter of the chimney is calculated automatically taken into account the previous dimensions.

**Wind velocity pressure** = In this field the user selects from the list the pressure which is generated on the chimney due to the wind.

**External air temperature TL** = In this field the user defines the external air temperature in °C. The external temperature shall be taken as the maximum temperature of external air at which the chimney is intended to be used (for heating systems is normally 15 °C).

**Mean ambient air temperature of the chimney Tu** = In this field the user defines the mean ambient air temperature in °C.

**Ambient air temperature at the chimney outlet Tuo** = In this field the user defines the ambient air temperature at the chimney outlet in °C. For chimneys operating under dry conditions Tu = 0 °C while for chimneys operating under wet conditions Tu = -15 °C.

**Flow safety coefficient SE** = In this field the user selects the flow safety coefficient in order to take into account the local resistance in the flue gas collector.

Additional data for the connecting flue pipe

**Connecting flue pipe 1 temperature for temperature instability** = When the chimney has been calculated, the temperature of the connecting flue pipes is automatically calculated. By pressing the selection button in the field, the user can edit the data of the connecting flue pipe.

Additional data for the chimney

**Extended length of chimney** = In this field the user defines the extended length of the chimney (in m), that is the difference in height between the axis of the lower flue gas inlet into the chimney and the outlet of the chimney.

**Chimney effective height** = In this field the user defines the effective height of the chimney (in m), that is the difference in height between the axis of the higher flue gas inlet into the chimney and the outlet of the chimney.

**Sum of local resistivities of the chimney Sz** = In this field the sum of the coefficients of flow resistance due to directional and/or cross sectional and/or mass flow changes in the flue is calculated by pressing the selection button.

**Chimney thermal resistance 1/L** = The thermal resistance of the chimney shall be obtained from the product manufacturer (in m<sup>2</sup>K/W). For multiwall custom built chimneys, the thermal resistance shall be determined from the layers data which will be defined by pressing the selection button in the field.

**External heat transfer coefficient aa** = The external heat transfer coefficient shall be 8 W/(m<sup>2</sup>K) for connecting flue pipes and chimneys internal to the building or 23 W/(m<sup>2</sup>K) for connecting flue pipes and chimneys external to the building.

**Chimney roughness value r** = In this field the user selects from the list the chimney roughness value depending on the chimney material.

**Ventilation installation** = In this field the user selects whether there is a ventilation installation in the space.

**Chimney temperature for temperature instability** = In this field the temperature of the chimney (in °C) for temperature instability is calculated automatically.

**Stability conditions satisfaction for the desired diameter** = When the minimum hydraulic diameter of the chimney has been calculated and there is "YES" in this field, the chimney has been calculated correctly, otherwise the user has to make changes in order to satisfy the stability conditions.

### 3.7.9 Network Drawing

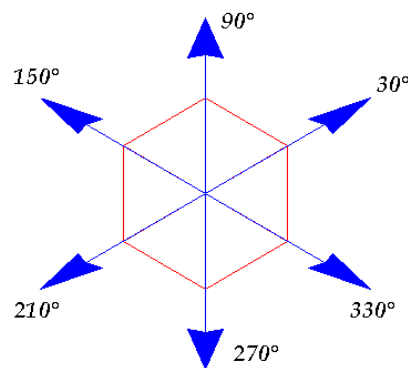
The numbered network drawing is shown on the screen, provided that polar coordinates have been inserted in every network branch (see calculation sheet).

### 3.7.10 Vertical diagram (Chart)

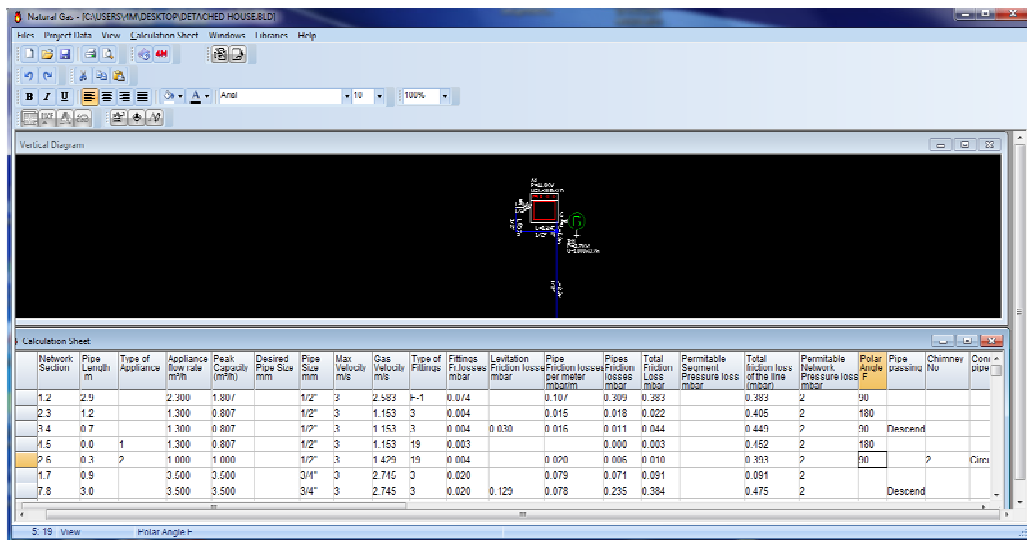
If the user wants to create a vertical diagram using the calculation sheet (and not automatically, using FINE NGAS package), the above option creates the vertical chart provided that the polar coordinates have been inserted in every network branch. At the bottom we find the Counters.

It is pointed out that it is possible to select a drawing for the engine room from the menu "Vertical diagram" > "Select side for machine room".

**Axonometric drawing:** The axonometric drawing is being generated exactly as the vertical drawing, by filling a value in the "Polar Angle" column, on the "Calculation Sheet" (instead of the angle values 0, 90, 180, 270 the values 30, 150, 210, 330 could be used).



By activating the two windows "Calculation Sheet" and "Vertical diagram" as in the following screenshot, and by modifying some values, the user can understand easier this functionality, given that the drawing is being shaped on real time basis.



---

### 3.7.11 Sections Friction losses

In this window you can calculate and view in the screen the total frictions in all sections (or routes), starting from the gauge and ending in each terminal node. Moreover, for more convenience, the most unfavorable section appears in the bottom of the window.

### 3.7.12 Bill of Materials-Costing


The bill of materials-costing results regarding the specific project is presented. You can edit the bill of materials-cost estimation sheet, modify the costs or quantities, insert discounts and add jobs or materials followed by their costs and quantities.

### 3.7.13 Detailed Bill of Materials

The exact quantity of the materials that have been used in the specific installation (i.e. pipes, appliances etc.) is being generated by the program in an editable table form.

### 3.7.14 Technical Description

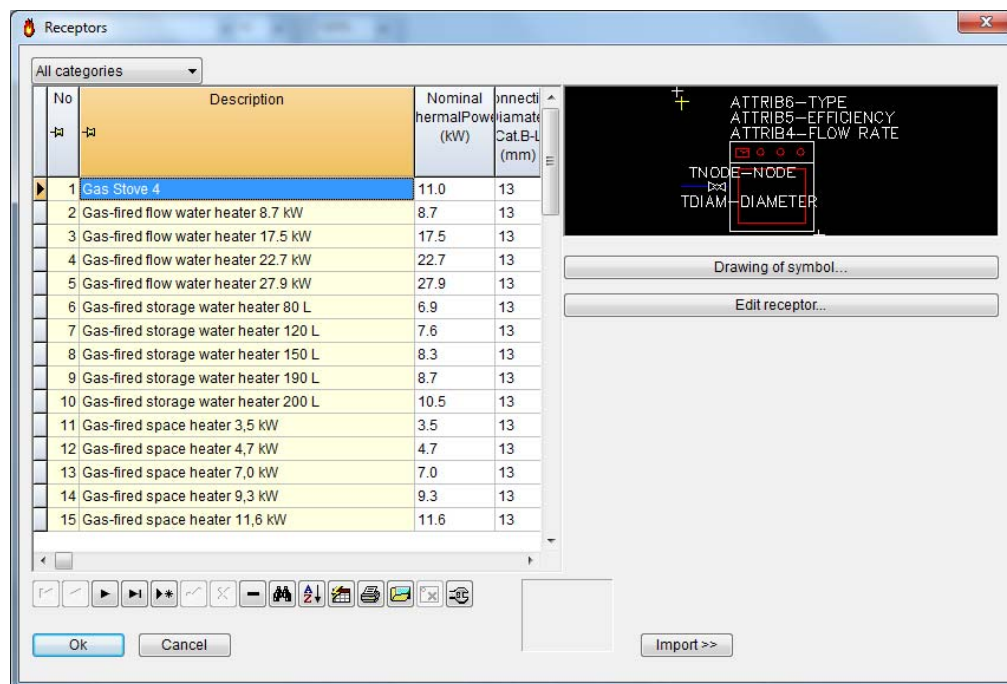
The window "Technical Description" supports the creation of the project technical description. You can select among various technical description prototypes and text editing styles. To load a different technical description, in the program menu go to Technical description -> Prototypes, select the prototype you want and press "Load". If you want to save the technical description you made so as to use it again, in the same window enter a name and a number (e.g. 02 Gas technical description) and choose "Save as".

 **Note:** The technical description files are stored in FINE14\_NGAS\CALC\KAERI\ , named KAERTP01.RTF, KAERTP 02.RTF, etc. their descriptions are stored in the file KAERCV.LST.

## 3.8 Libraries

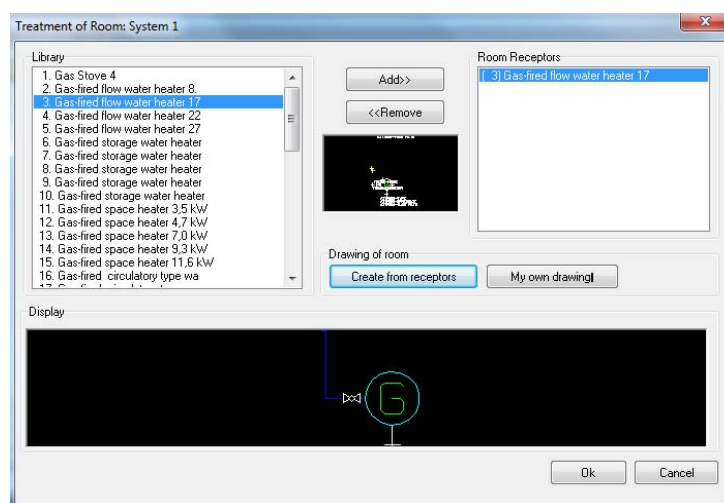
The FineNGAS Libraries include the following categories, with their respective features:

- **Fittings**, with specified characteristics (description, coefficient z, cost)
- **Pipes** (pipe types) with a given roughness, standardization (nominal, internal diameter, weight, capacity etc.), material and cost.
- **Appliances** with lots of appliances of all five categories (DWH, GI, H, RH and UWH), their thermal power, gas flow, type, connection diameter etc.





- **Systems of appliances:** The library is fitted with a tool that allows the design of new receptor systems as well as the modification of the existing ones.

The creation and/or modification are being easily performed through a smart wizard who allows the user to add or remove a gas appliance in a visual way.



The user has also the option to import any dwg file.

- **Offer libraries** where the user can add his own libraries.

In any of the above libraries, you can easily change or add your own information. In order to add a new fitting, appliance etc. press the "Append record" button  at the bottom of the page. The program automatically creates a new blank row where you add the information you want and after pressing the "Post edit" button  you press OK. In this way, you can use your own libraries in every project you are working.

---

## 3.9 Help

This option provides the user with the instructions of the program, according to the windows standards.

## 3.10 Examples

In this chapter two step-by-step examples are presented, which will help you understand in practice the basics of drawing in FINE NGAS. These simple examples aim mainly at helping you start drawing building entities (walls, openings etc.) along with the gas network. More specifically:

- *Example 1*, describes the steps of drawing a simple level (optional).
- *Example 2*, describes the steps of drawing a simple fire gas installation network, proceeding to the calculations and generating the necessary drawings.



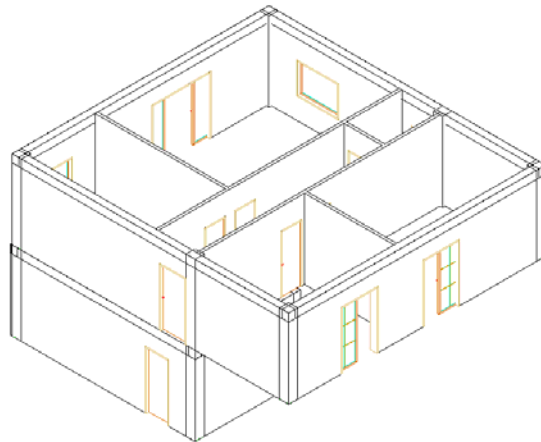
**Important:** *It isn't necessary to draw the levels of the building with walls, openings etc. in order to calculate the gas network. If you already have 2D architectural drawings of the levels, you can insert them and place on them the receptors, the pipes etc. Nevertheless, if you do not have architectural drawings, it will be more helpful to draw the walls etc. so as to be able to design the network more accurately.*

Apart from the following examples, you can watch the same steps (and much more) in the videos you will find in <http://www.4msa.com/FineGasENG.html> or in <http://www.youtube.com/watch?v=zAP91yhK2D8>.

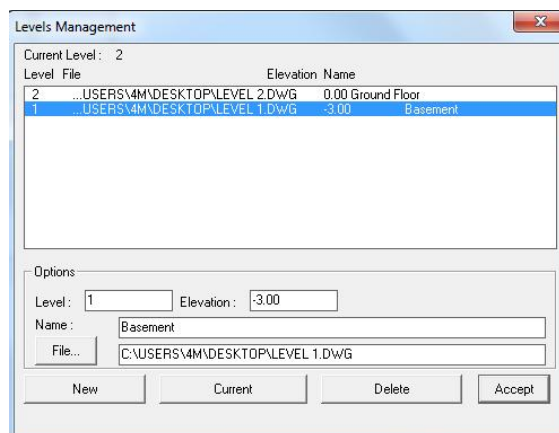
### 3.10.1. Building design example

In this example, the drawing of the adjacent simple building is described, including the steps from the beginning (creating a new project) to the end so that you can easily understand the function of FINE NGAS in practice.

For this example, we will import two architectural plan views that have been designed in AutoCAD although you can start drawing from scratch. If you have already designed the building in IDEA Architecture 14, you can skip this example.

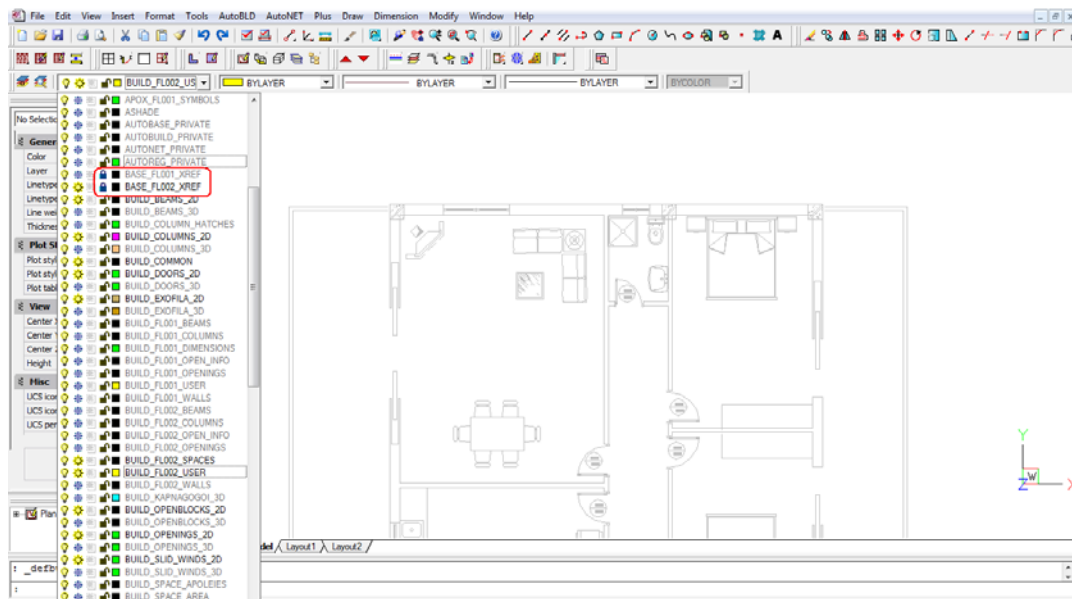


1. From FILES menu, we select “New project”, type “Example\_1” and press “Accept”.
2. From AutoBLD menu, we select “Building definition”, define as level 1 the “Basement” in elevation “-3” and from “File” we load its architectural drawing. Before we define the second level, we press “New” (so as to save the changes) and we follow the same steps for the second level, as it is shown below.



When we finish, we press once again “New” and then “Accept” and automatically we defined the two levels with their architectural drawings.

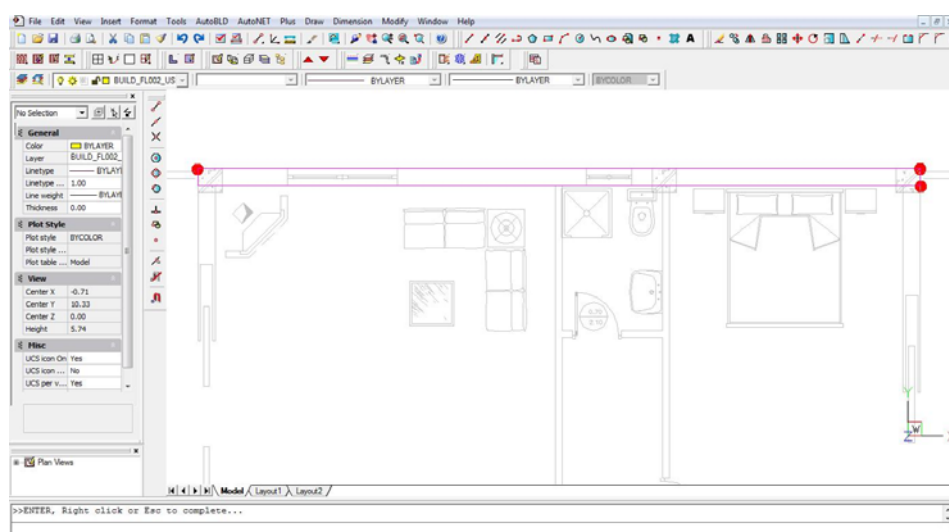
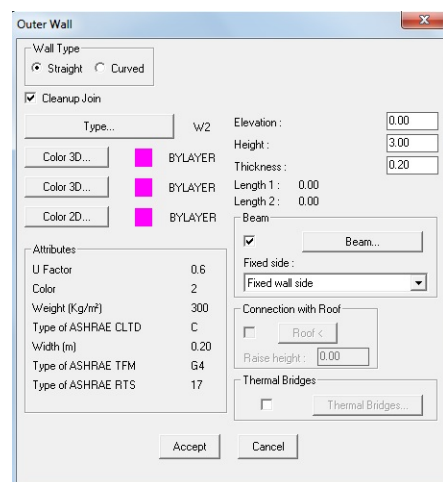
3. Before we start drawing, we “Lock” the architectural drawings from the layers list on the left by pressing the “Lock” symbol as it is shown below. The name of the architectural drawing will be BASE\_FLOO1\_XREF for the first level, BASE\_FLOO2\_XREF for the second level etc.



4. From AutoBLD, we select Walls > Outer wall and in the properties window we define the height (3 m in our example) and the width (0.25 m) of the wall, the height and the width of the column (it will be automatically designed along with the wall), its type etc.

When we finish, we press “Accept” and we design the wall by defining three points:

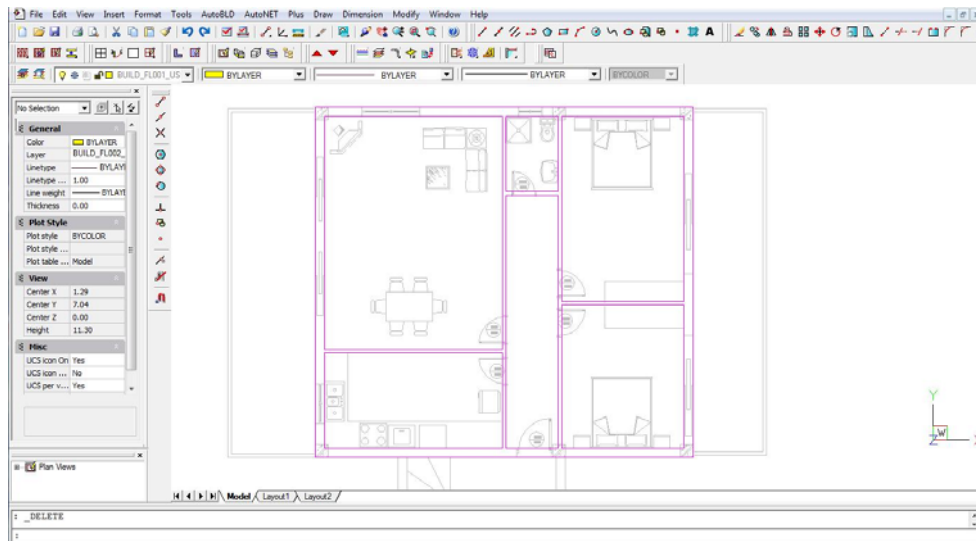
- First, we click on the beginning of the wall (where the left red spot is, in the following image)
- Then, we click on the end of the wall (where the upper right red spot is) and
- Finally, we click on the side that the wall “grows” (where the down right red spot is).



You can enable the “Entity snaps” toolbar (or the “ESNAP” setting) so as to select easily the edges of the wall.



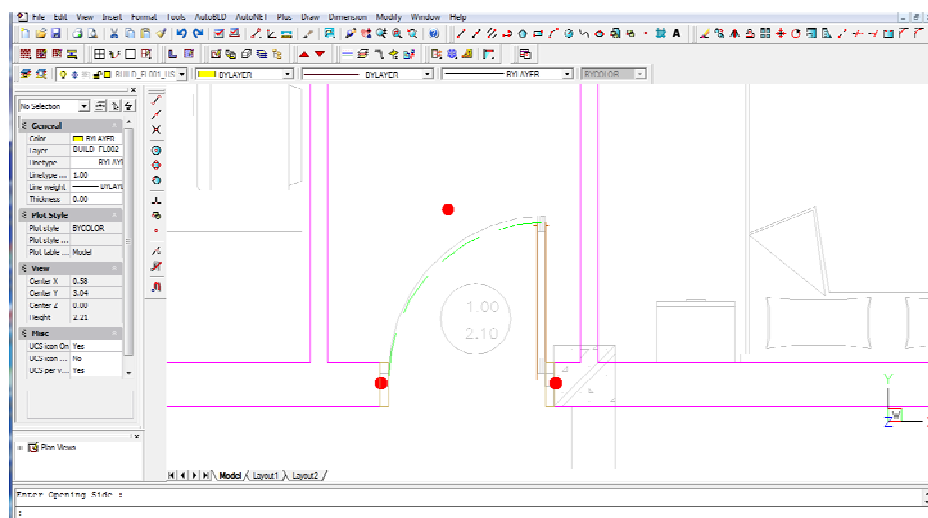
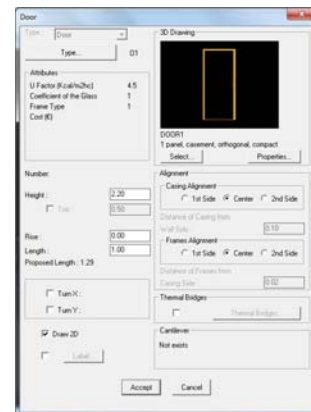
Continuing in the same way, we draw all the walls (inner and outer) of the ground floor.



5. From AutoBLD, we select Opening > Door and in the properties window, we select the height (2.20 m in our example), the length (1 m), the type of the door etc.

When we finish, we press “Accept” and we design the door:

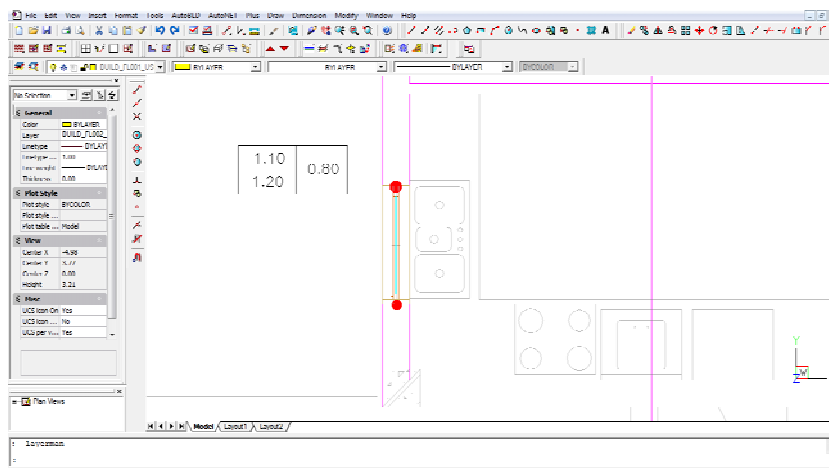
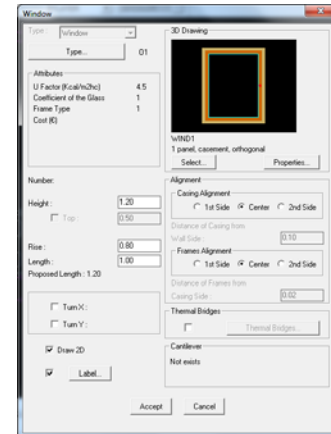
- First, we select the wall upon which we want to place the door.
- We click on the starting point of the door (where the left red spot is, in the following image)
- We click on the ending point of the door (where the upper right red spot is). As long as you have specified the length of the door, it is not necessary to specify this point exactly. You only need to specify a point near the start towards the side that the second point is located.
- Finally, we click on a point that shows the direction towards which the door will open (where the down right red spot is).



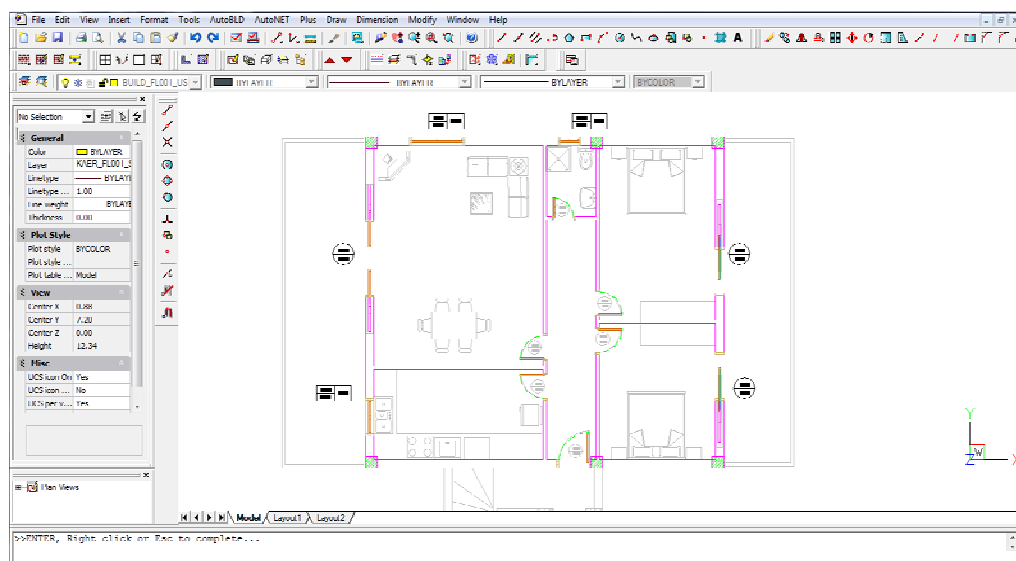
6. From AutoBLD, we select Opening > Window and in the properties window, we select the height (1.20 m in our example), the length (1 m), the rise (0.80 m), the type of the window etc.

When we finish, we press “Accept” and we design the window:

- First, we select the wall upon which we want to place the door.
- We click on the starting point of the window (where the upper red spot is, in the following image)
- We click on the ending point of the window (where the down red spot is). As with the door, as long as you have specified the length of the window, it is not necessary to specify this point exactly. You only need to specify a point near the start towards the side that the second point is located.



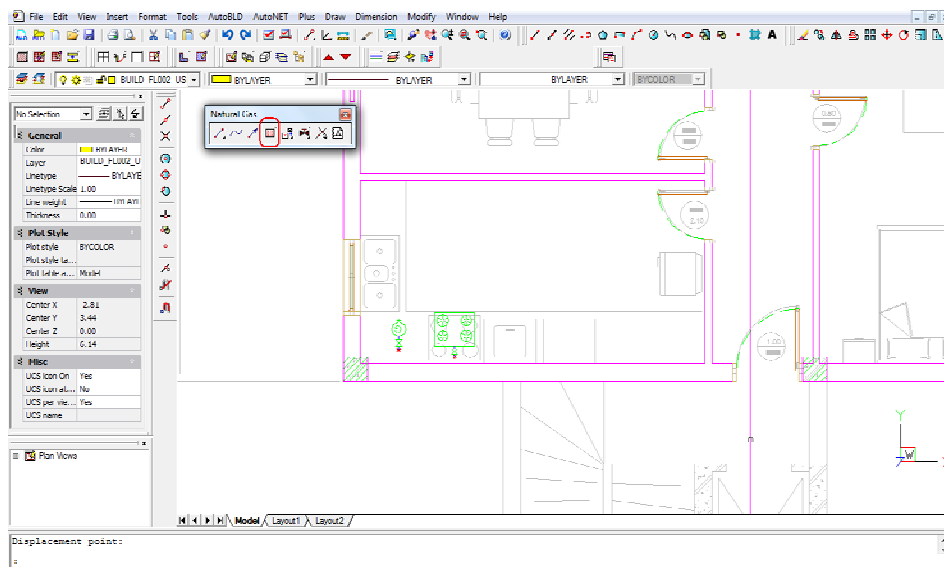
In the following image, we have deactivated the architectural drawing, so as to have a better inspection of the drawing. To do this, go to AutoBLD > Layers management > and uncheck the “XREF” box.



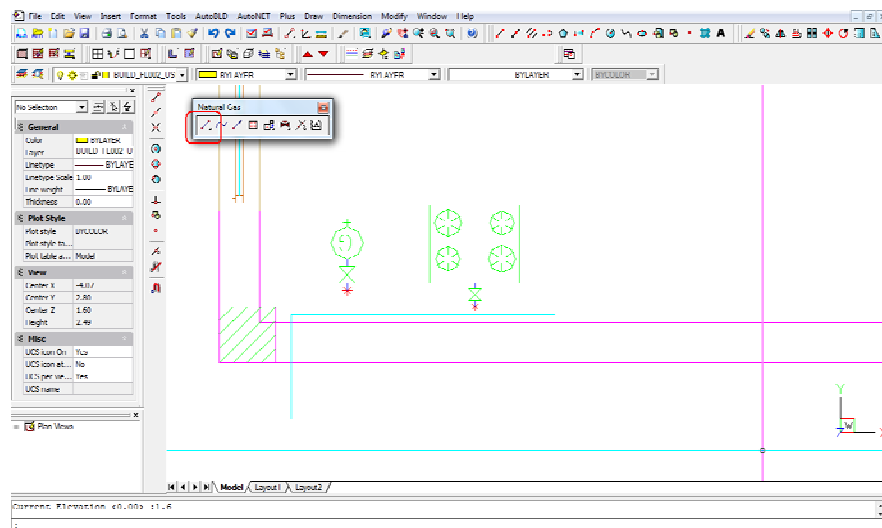
### 3.10.2. Gas installation Network design example

1. First of all, we place the appliances in the rooms of the ground floor. From AutoNET > Receptors (or directly from the “receptors” symbol of the toolbar), we select the “Gas stove”, press “Accept” and place it in the first room. Repeating the same step, we place in the same room the gas-flow water heater and in the level of the basement the gas-fired heating boiler.

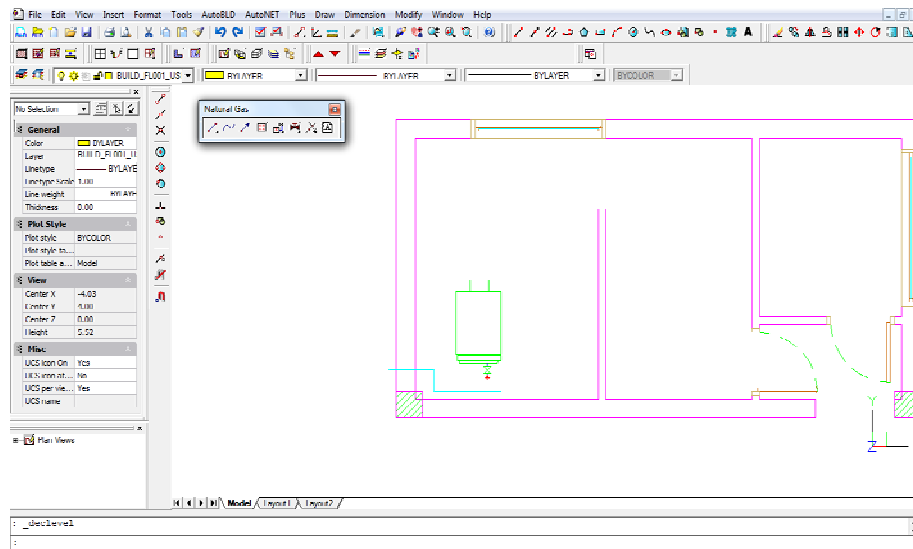
Before we place the water heater in the drawing, from the “Plus” menu > we can select “Set elevation” and in the command line type the elevation we want for the water heater to be placed (e.g. 1.6 m). In this way, the water heater will be placed in 1.6 m in z axis in relation to the ground floor level. After we place the water heater, we restore the elevation with the same command.




2. From AutoNET > we select “Pipe” (or from the gas toolbar as highlighted below) and start drawing the gas network of the ground floor. If you want, you can draw the pipes in a different elevation by repeating step 1 (in our example, we draw the water heater in an elevation of “1.6” m) or draw curved pipes (from AutoNET > “Curved pipe”).

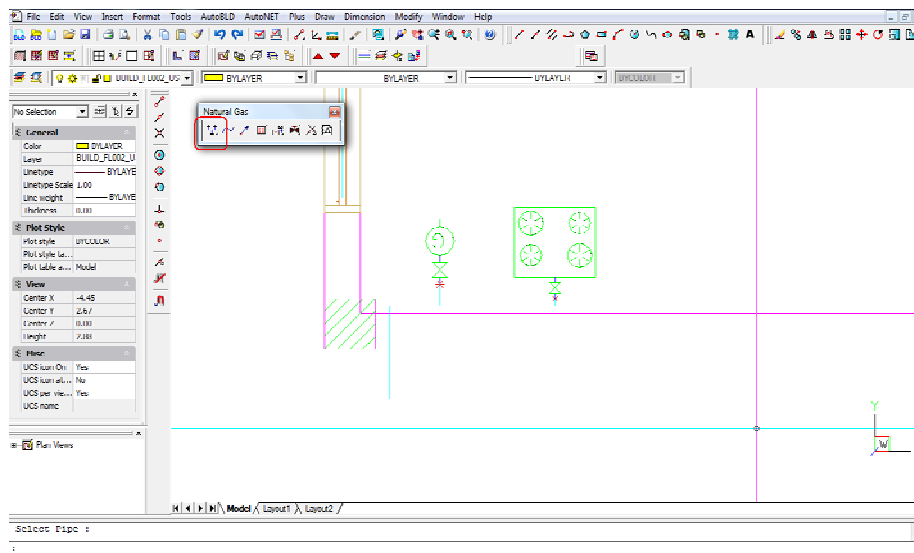


We repeat the same steps for the basement as it is shown below.



3. The next step is connecting the pipes with the receptors and we do this easily by selecting from AutoNET > “Connect receptors to existing pipe” (or from the corresponding symbol  of the toolbar).

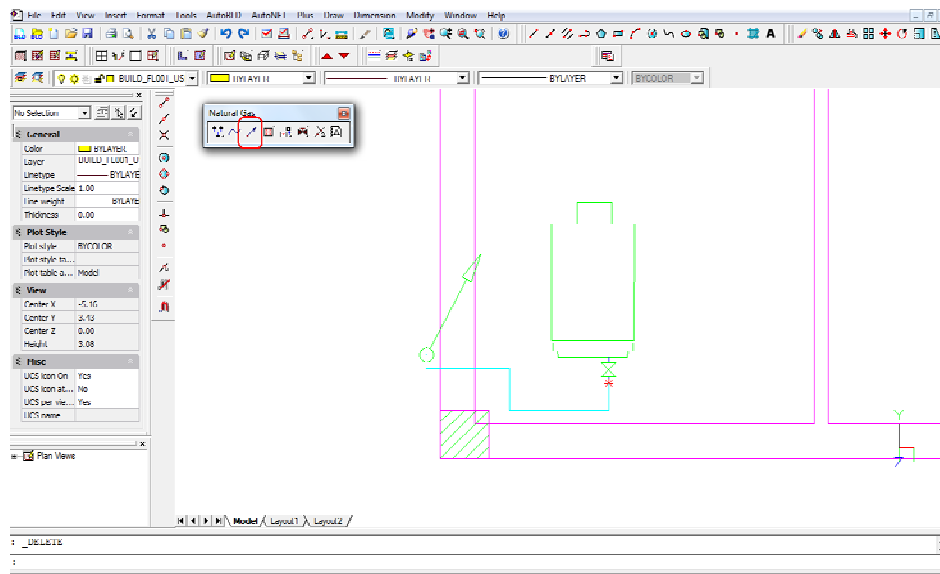
- We click on the receptor
- We click on the pipe next to it
- We press ENTER and they are automatically connected.
- We repeat the step for all the receptors (we can connect more than one receptor to the same pipe each time).
- We delete the part of the pipe that we no longer need.



4. If we want to place a vertical pipe that leads from the basement to the ground floor, we select from AutoNET > “Main vertical pipes”:

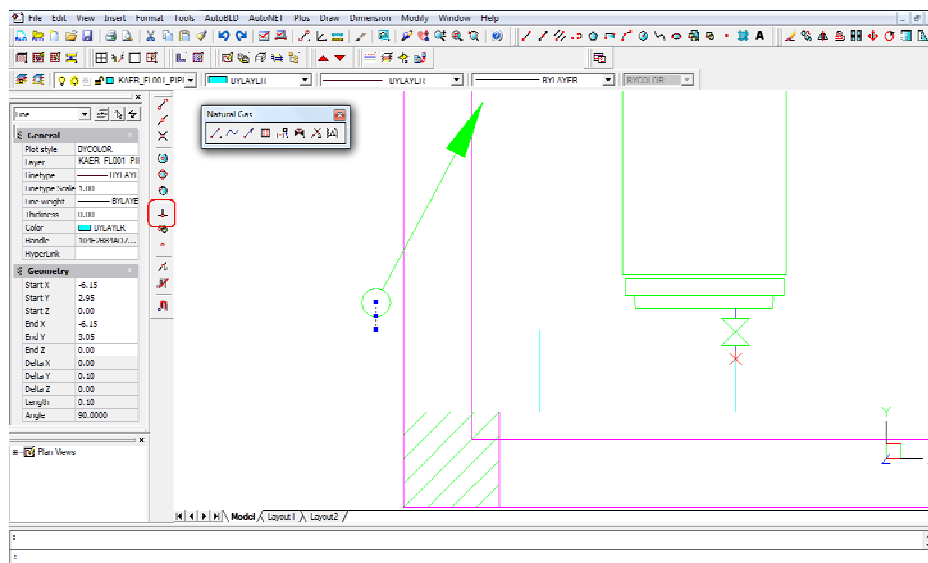
- We click on the location we want to place the vertical pipe (in the command line you can see the “Enter XY location” prompt message).
- We type the height “-3” which is the starting point of the vertical pipe in this example (the basement elevation).


- We type the height “0” which is the ending point of the vertical pipe in this example and the vertical pipe is created.



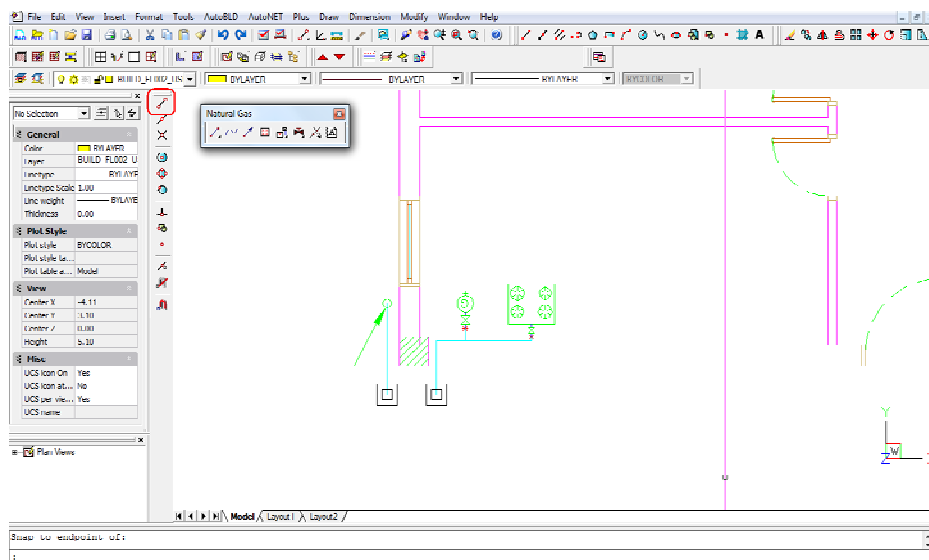
5. From AutoNET > we select “Pipe” and draw a pipe connecting the horizontal to the vertical pipe as it is shown below.

You can use the “Perpendicular snap” (it is highlighted in the picture) to connect precisely the horizontal to the vertical pipe.



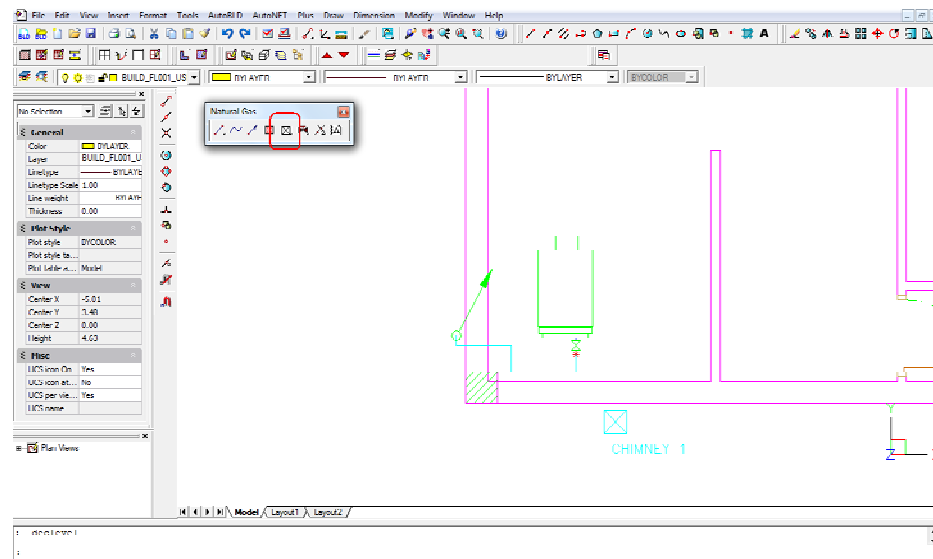
6. By selecting the level of the ground floor (you can use  to change levels or double click on the level's name in the left side of the screen) we draw a horizontal pipe starting from the vertical pipe as it is shown below.

Finally, from AutoNET > “Supply point”, we set the supply start point symbols (which is for the gas meter position). You can easily connect the symbol to the horizontal pipe using the “Endpoint snap” point (highlighted in the following picture).

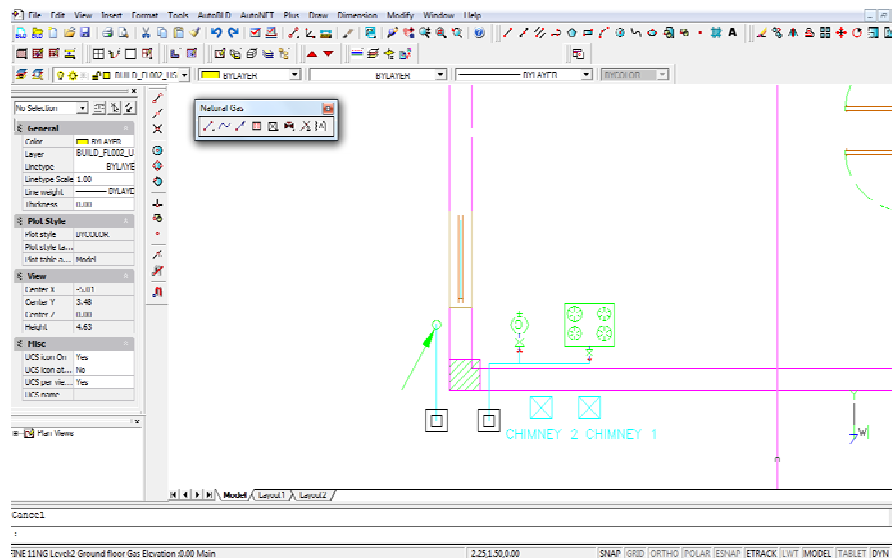


7. The next step is to place the chimneys on the drawing. On the basement, from the AutoNET menu > we select “Chimneys-Uptakes” > “Orthogonal chimney” (or from the toolbar as it is highlighted below):

- We click on the location we want to place the chimney (in the command line you can see the “Enter XY location” prompt message).
- We type the height “-3” which is the starting point of the chimney in this example (the basement elevation).
- We type the height “8” which is the ending point of the chimney and automatically the “Chimney 1” is placed.

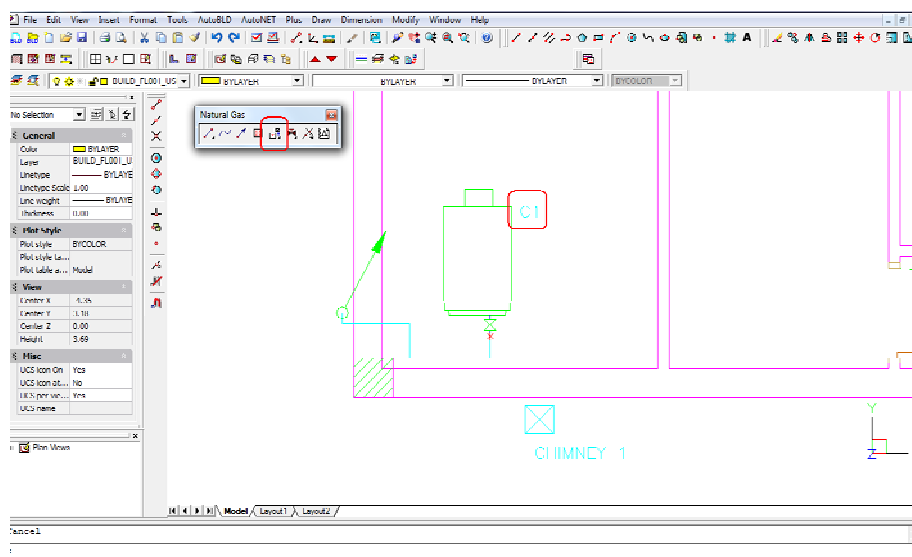


We repeat the same step for the chimney of the ground floor (starting point: 0 m, ending point: 8 m) and “Chimney 2” is placed on the drawing as it is shown in the following image.

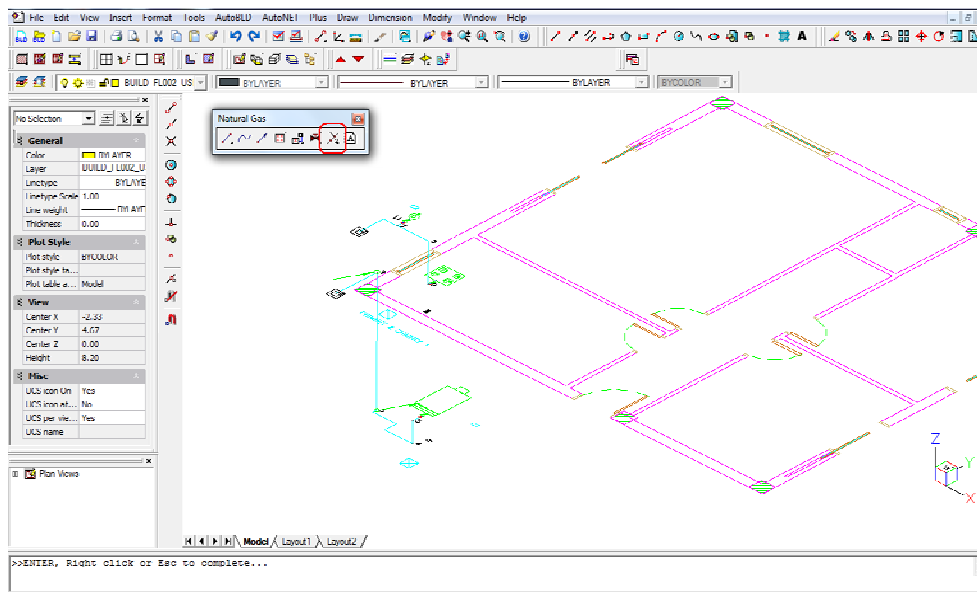


8. From AutoNET menu we select “Chimneys-Uptakes” > “Assign chimney to receptor” (or from the toolbar as it is highlighted below). On the basement, we select the heating boiler, press <Enter> and type the number “1” in the prompt message “Select Chimney that Receptors will Place Among”.

We repeat the same for the water heater of the ground floor, only typing “2” for the chimney number.



9. Before we proceed to the calculations, we select from AutoNET > “Network recognition”. This is an important step in order to make sure that the network has been drawn correctly. If there are any mistakes, a warning message will appear, otherwise, the network sections are numbered as it is shown below and we can proceed to the calculations.



10. From AutoNET > we select “Calculations” and in the window that opens automatically, we select Files > Update from Drawing. From “Windows” menu we open the Calculation Sheet where the network information have been transferred and make any changes we want (detailed information in Chapter 3.7.7).

Natural Gas - [C:\USERS\MM\DESKTOP\EXAMPLE 1.B.D]

File Project Data View Calculation Sheet Windows Libraries Help

B U Z U [Units: And] [10] [100%]

Calculation Sheet

	Network Section	Pipe Length m	Type of Appliance	Appliance flow rate (m³/h)	Peak Capacity (m³/h)	Desired Pipe Size mm	Pipe Size mm	Max Velocity m/s	Gas Velocity m/s	Type of Fittings	Fittings Pressure loss mbar	Levitation mbar	Pipe Friction losses mbar/m	Pipes Friction losses mbar	Total Friction Loss mbar	Permissible Segment Pressure loss mbar	Total friction loss of the line (mbar)	Permissible Network Pressure loss mbar
1	1.2	1.1		3.500	3.500	3/4"	3	2.745	3	0.020			0.078	0.085	0.106		0.106	2
2	2.3	3.0		3.500	3.500	3/4"	3	2.745	3	0.020	0.129		0.078	0.235	0.364		0.490	2
3	3.4	1.7		3.500	3.500	3/4"	3	2.745	F-1	0.081			0.078	0.133	0.214		0.704	2
4	4.5	0.5		3.500	3.500	3/4"	3	2.745	3	0.020	-0.021		0.078	0.039	0.030		0.142	2
5	5.6	0.2	29	3.500	3.500	3/4"	3	2.745	19	0.014			0.080	0.016	0.030		0.772	2
6	1.7	1.0		2.300	1.800	1/2"	3	2.583	1-2	0.056			0.100	0.100	0.163		0.163	2
7	7.8	0.8		1.300	0.807	1/2"	3	1.153	3	0.004			0.015	0.012	0.016		0.179	2
8	0.9	0.9		1.300	0.807	1/2"	3	1.153	3	0.004	0.039		0.015	0.014	0.056		0.235	2
9	9.10	0.0	1	1.300	0.807	1/2"	3	1.153	19	0.003			0.000	0.003			0.238	2
10	7.11	0.1	2	1.000	1.000	1/2"	3	1.429	19	0.004			0.020	0.002	0.006		0.165	2
11																		
12																		
13																		
14																		
15																		
16																		
17																		
18																		
19																		
20																		
21																		
22																		
23																		
24																		

1:1 View Network Section

11. In order to calculate the diameter of the chimneys, from “Windows” menu we open the “Chimney calculations”.



Chimney No.	1
Appliances of chimney gas flow rate	3.50
Nominal Power of appliances of the chimney (kW)	30.00
Number of chimney appliances	1
Type of appliances in chimney	with forced draught burner
Minimum hydraulic diameter required for the chimney (cm)	8.50
Chimney cross section area type	Square
Length or diameter of the selected chimney (cm)	10
Width of the selected chimney (cm)	
Hydraulic diameter of the selected chimney (cm)	10.00
Wind velocity pressure (Pa)	0 Pa (prospective conditions outside of reverse pressure zone)
External air temperature T <sub>1</sub> (°C)	15.000
Mean ambient air temperature of the chimney T <sub>a</sub> (°C)	0.000
Ambient air temperature at the chimney outlet T <sub>2</sub> (°C)	0.000
Flow safety coefficient SE	1.5
Additional data for the connecting flue pipe	
Connecting flue pipe 1 temperature for temperature inside	135.000
Connecting flue pipe 2 temperature for temperature inside	
Connecting flue pipe 3 temperature for temperature inside	
Connecting flue pipe 4 temperature for temperature inside	
Connecting flue pipe 5 temperature for temperature inside	
Additional data for the chimney	
Extended length of chimney (m)	11
Chimney effective height (m)	11
Sum of local resistances of the chimney (Pa)	1.150
Chimney Thermal resistance (1/kJ/h (m²K/W))	0.514
External heat transfer coefficient w <sub>a</sub> (W/m²K)	23.000
Chimney Roughness factor (m)	0.001 selected steel pipe weight aluminum

In the above window:

- In "Chimney no" field we type "1" so as to calculate the first chimney.
- We fill in the "Extended length of chimney" and "Chimney effective height" fields. In the first chimney, only one appliance is connected so we type the same value of "11" m in both fields (the chimney starts from an elevation of -3 m and ends in elevation of 8 m).
- Automatically in the field "Minimum hydraulic diameter required for the chimney" the calculated diameter of 8.50 cm is shown.
- We fill in the field "Length or diameter of the selected chimney" with the diameter 10 cm and the stability conditions are satisfied (in the last field of the window).

We repeat the same steps for the second chimney.

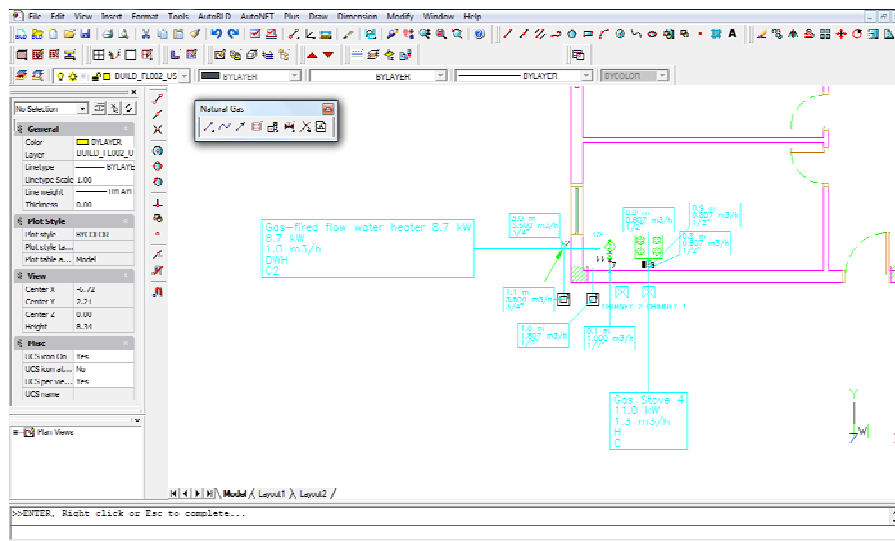
When we finish the calculations, we save and close the window and return to FINE NGAS.

- From AutoNET we select "Update Drawing" and in the opening window we choose which information we want to be show on the screen for the pipes and the receptors.

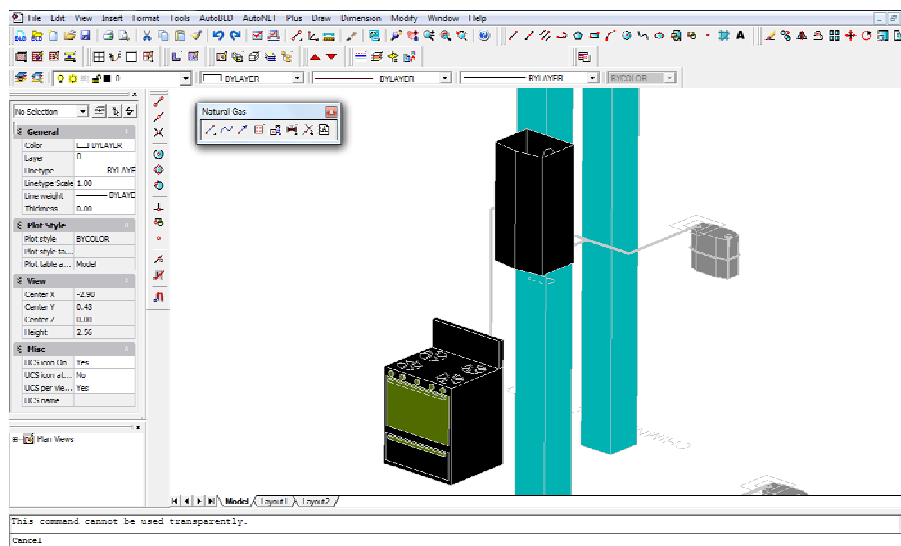
Segment	Length	Flow Rate	Diameter	Velocity	Total friction
1	7.1 m	3.500 m³/h	3/4"	2.725 m/s	0.111 mbar
2	3.0 m	3.500 m³/h	3/4"	2.745 m/s	0.054 mbar
3	1.2 m	3.500 m³/h	3/4"	4.292 m/s	0.048 mbar
4	0.5 m	3.500 m³/h	3/4"	2.745 m/s	0.036 mbar
5	0.2 m	3.500 m³/h	3/4"	2.725 m/s	0.020 mbar
6	1.0 m	1.007 m³/h	1/2"	2.503 m/s	0.023 mbar
7	0.8 m	0.807 m³/h	1/2"	1.153 m/s	0.025 mbar
8	0.5 m	0.807 m³/h	1/2"	1.153 m/s	0.025 mbar
9	0.5 m	0.807 m³/h	1/2"	1.153 m/s	0.025 mbar
10	0.5 m	0.807 m³/h	1/2"	1.153 m/s	0.025 mbar
11	0.1 m	1.000 m³/h	1/2"	1.425 m/s	0.025 mbar

Node	Receptor Name	Thermal Load	Flow Rate	Type	Type
1	Unconnected building node: 0.0 m³/h	38.11 kW	15 m³/h	Flow	1.1
10	Gas Stove 4	11.0 kW	4.0 m³/h	H	A1
11	Standard flow water heater 5.0 m³/h	6.7 kW	1.0 m³/h	Flow	1.1

By clicking “Auto Placement”, the selected information is shown on each level.



13. If we want, from AutoNET we can select “Convert single line to 3D” and the program automatically converts the network into 3D in relation to the calculation results.



14. Finally, we can generate the axonometric diagram of the network from AutoNET > “Axonometric diagram” > “Create”. Automatically a new drawing is created with the axonometric diagram and its data as it is shown below.

