

FineELEC

Quick Start Guide

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

Preface

This Quick Start Guide provides a fast and friendly introduction on FineELEC, describing its concept and functionalities. All the features and functions of the program are presented and explained in detail within the complete User's Guide, along with informative examples.

FineELEC, the **F**ully **I**ntegrated **E**nvironment for **E**lectrical Building 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 IntelliCAD engine adopting the common cad functionality and open dwg drawing file format. The CAD component helps the user to design and then calculates and produces completely automatically the entire calculations issue for every Electrical 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 Calculation component of FineELEC 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 module spreadsheets.

Despite its numerous capabilities, FineELEC 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 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 FineELEC, showing its philosophy and main features.
- Part 3 describes the operations of the calculation environment of FineELEC.

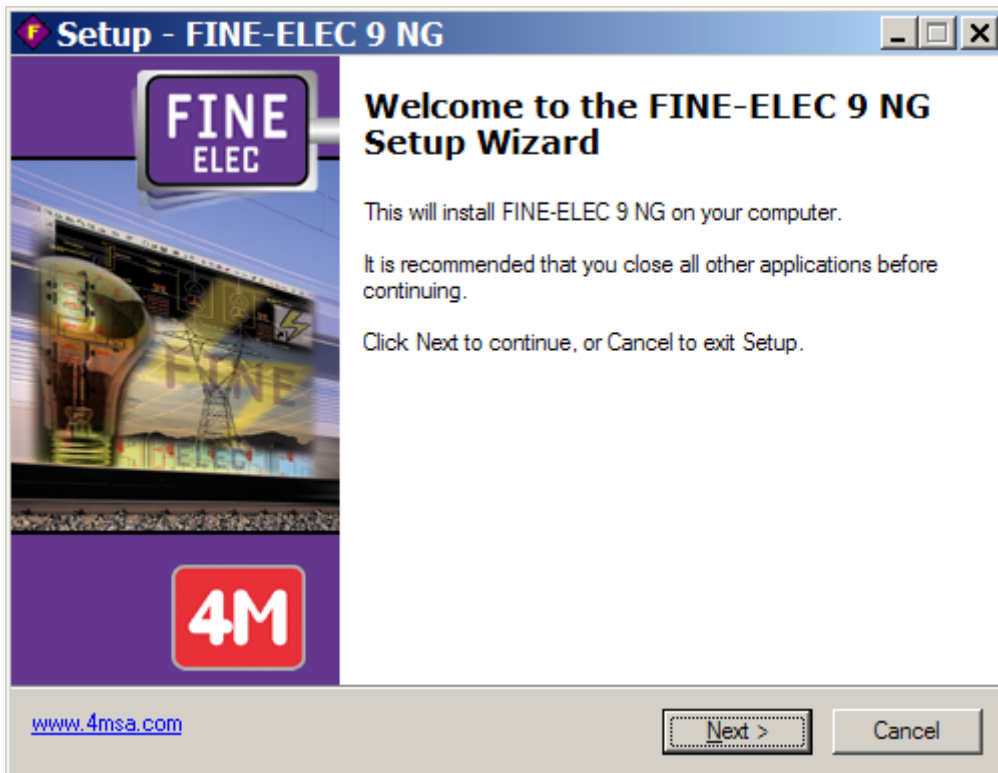
<i>Preface</i>	3
1. Installation - Launching	7
1.1 Installing FineELEC	7
2. CAD Component	9
2.1 Overview	9
2.2 Main menu	9
2.3 Drawing Principles & Basic Commands	10
2.3.1 Drawing aids	12
2.3.2 Drawing Coordinates	12
2.3.3 Drawing Basic Entities	13
2.3.4 Useful Commands.....	13
2.3.5 Grips	14
2.3.6 Print.....	14
2.4 AutoBUILD: Architectural Design	15
2.4.1 Building Definition	16
2.4.2 Drawing Walls	17
2.4.3 Drawing Openings	18
2.5 AutoNET: Electrical Installation Design	19
2.5.1 General Philosophy	19
2.5.2 Special Commands for Cable Construction	26
2.5.3 Modifying an existing network	27
2.5.4 Receptors	28
2.5.5 Symbols	31
2.5.6 General Symbols.....	31
2.5.7 Network Recognition and Numbering	31
2.5.8 Calculations	31
2.5.9 Drawing Update	32
2.5.10 Legend	32
2.5.11 Vertical Diagram.....	32
2.5.12 Library Management.....	32
2.6 A simple example	33
3. Calculations	37
3.1 Files	37
3.2 Options	38
3.2.1 Network data	38
3.2.2 Cables (VDE).....	40
3.2.3 Cable parameters (HD 384)	41
3.2.4 Typical Load Types	41
3.3 View	43
3.4 Windows	43
3.4.1 Calculation Sheet	43
3.4.2 Panel Calculation	53
3.4.3 Export to a DWG or DXF file.....	56
3.4.4 Substation.....	56
3.4.5 Fan	56
3.4.6 Block Diagram	56
3.4.7 Voltage Drops	57
3.4.8 Bill of Materials	57
3.4.9 Technical Reports, Assumptions, Cover Page	57
3.5 Libraries	57

3.6 Help 63

1. Installation - Launching

1.1 Installing FineELEC

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 Setup window appears, choose the language for the installation and click OK.
3. When the Welcome page appears (as shown below), click **Next**.



4. 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).
5. 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.
6. 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 FineELEC" checkbox is selected, the program will start running.



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

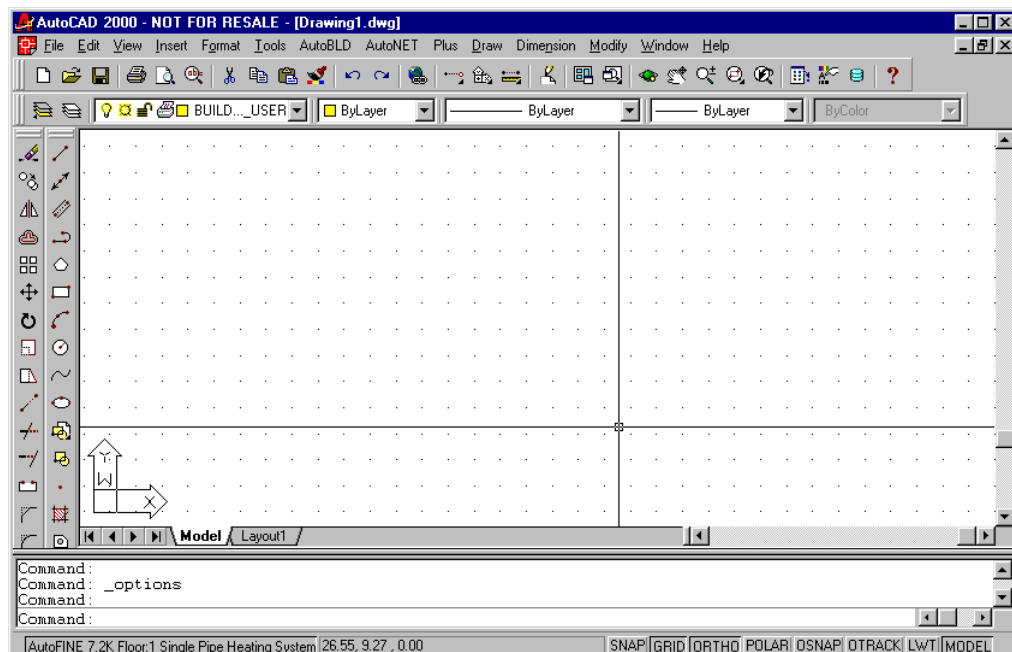
2. CAD Component

2.1 Overview

As mentioned above, FineELEC is a powerful workstation for electrical design of building installations. Based on HD384 regulations, the program performs automatically all the necessary calculations directly from the drawings, producing thus all the case study results (Calculation issue, technical descriptions, full-scale drawings properly updated, bills of materials etc). This chapter describes the operation of the CAD component of FineELEC, which operates on its own autonomous environment, embedding the IntelliCAD engine. This CAD component considers the building and its Electrical installation as being composed of intelligent entities with their own attributes, consistently related to each other. More specifically, the CAD Component includes 2 main modules interacting between each other, giving the impression to the Designer that he/she virtually works on the building: It is about a) AutoBLD for the architectural design (3D building model), and b) AutoNET for the electrical network design.

2.2 Main menu

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

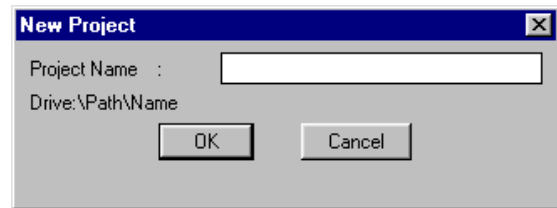


Among the commands of the designing environment, we notice the following main groups of commands:

1. **FILE** group of commands for the file management needs (New Project, Open Project and Project Information).
2. **AutoBLD** group of commands serving for the Architectural designing.

3. **AutoNET** group of commands, containing the instructions required for the designing and recognition of the electrical network.
4. **PLUS** menu including a series of designing facilities for the user.

To start drawing with FineELEC, a new project should be first defined by applying the **“New Project”** command, which is located at the **“FILE”** menu. Then a window appears on screen where the name of the Project must be filled. Then, click OK and you are now ready to start drawing and work on the project.

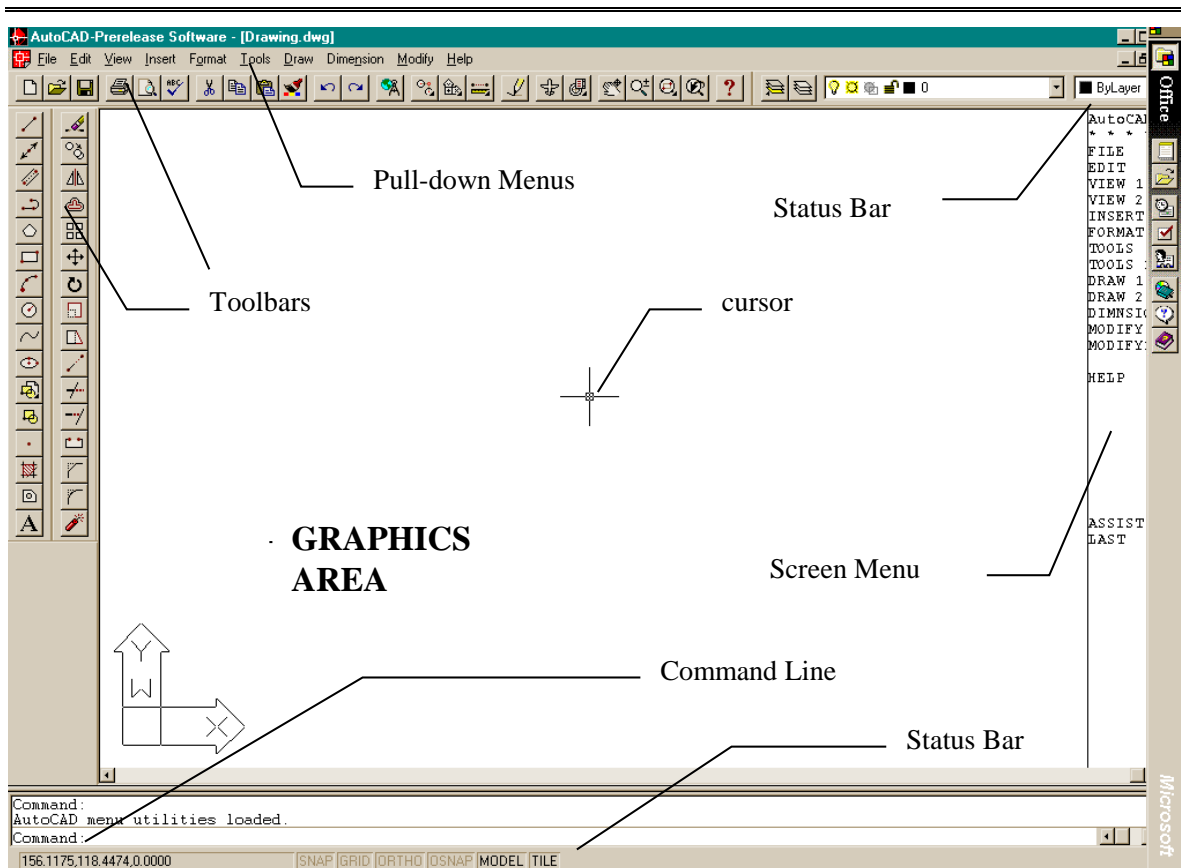


In order to **“Open”** an existing project (i.e. for further editing or just for viewing), just select **“Open Project”**, and a list with the existing projects in your disk will be displayed on the screen. The functionality of this window list follows the windows standards.



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, IntelliCAD 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.
- **Graphics 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:** These menus appear by placing the cursor on the Status line.
- **Screen menu:** It concerns all the pull-down menus and submenus which are displayed on the right side of the screen (the screen menu can be activated and deactivated using the AutoCAD option "Preference").
- **Status Line:** It is the line on the top of the screen where the current layer (slide), the drawing status and the current cursor coordinates are displayed.
- **Cursor menu:** This menu appears when the cursor is in the graphics area and you press either the middle mouse button or <SHIFT> and the right mouse button simultaneously.

Please note that each mouse button performs a specific function. The functions of the mouse buttons are the following:

Left button: Selection of a command, point or object.

Right button: Enter

Middle button: "Osnap" command

2.3.1 Drawing aids

This section describes the basic drawing aids available to the user. These are the commands **Osnap** (object snap), **Ortho** (vertical/horizontal drawing), **Grid** and **Snap** (movement increment). More specifically:

OSNAP: The "Osnap" command forces the cursor to select a snap point of an object, which is within the Pick box outline. The snap 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 "Osnap" command can be activated either by holding down the "SHIFT" key and right clicking the mouse or by clicking the middle mouse button or through the additional toolbar.

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 (in AutoCAD 12, the indication "O" also appears on the top left side of the screen). The command is activated or deactivated by clicking the corresponding button-icon or by pressing **F8**.

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 button-icon or by pressing **F7** (If the grid is active, it appears on the Status Bar).

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). When "Snap" is on, the cursor seems to adhere, or "snap", to an invisible grid. "Snap" can be turned on and off either by clicking the corresponding button/icon or by pressing **F9**. (If it is activated, it appears on the Status Bar). The default Snap setting is **0.05 m** for both axes (X and Y).

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, either 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. If you do not insert a value for z or ϕ , 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 should 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: These are specified like the relative coordinates, but without using the @ symbol. The absolute coordinates are specified in relation with 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

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). The user 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 to the user. These are the commands "Zoom", "Pan", "Select", "Move", "Copy" and "Erase". In particular:

Zoom: "Zoom" increases or decreases the apparent size of the image displayed, allowing the user 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 / ±" 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) the user wants 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).

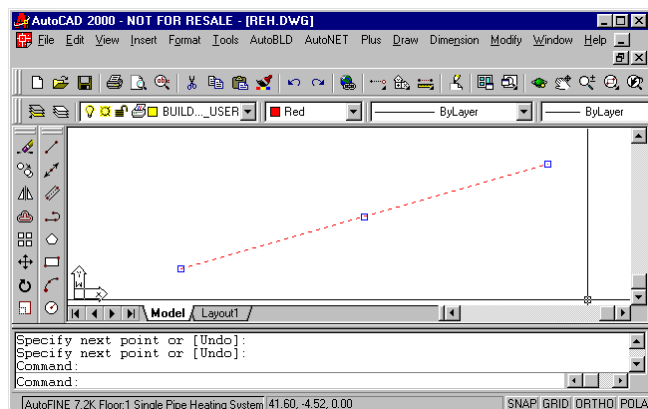
DDInsert (Insert Drawing): This command allows the user to insert another drawing (DWG file) or block in the drawing. When this command is selected, a window appears in which you should select block or file and then select the corresponding block or file from disk. Then you are prompted to specify the 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 should 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, which will be described in a following section. In order to insert a block in a drawing, you should use the "ddinsert" command described above.

Explode: The "Explode" command converts a block in a number of 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). Then object is displayed with grips (small squares), which mark control locations and are powerful editing tools. When you click a grip, it turns red and 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 the user has created a drawing and wants to print it. Any drawing can be printed using a printer or plotter or to a file. Printing is performed using

"PRINT" (or "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 colors or lineweights 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 > Print Preview.
 - On the Standard toolbar, click the Print Preview tool ().
 - Type ppreview and then press Enter.
3. After checking the preview image, do one of the following:
 - To print the drawing, click Print to display the Print dialog box.
 - To return to the drawing, click Close.

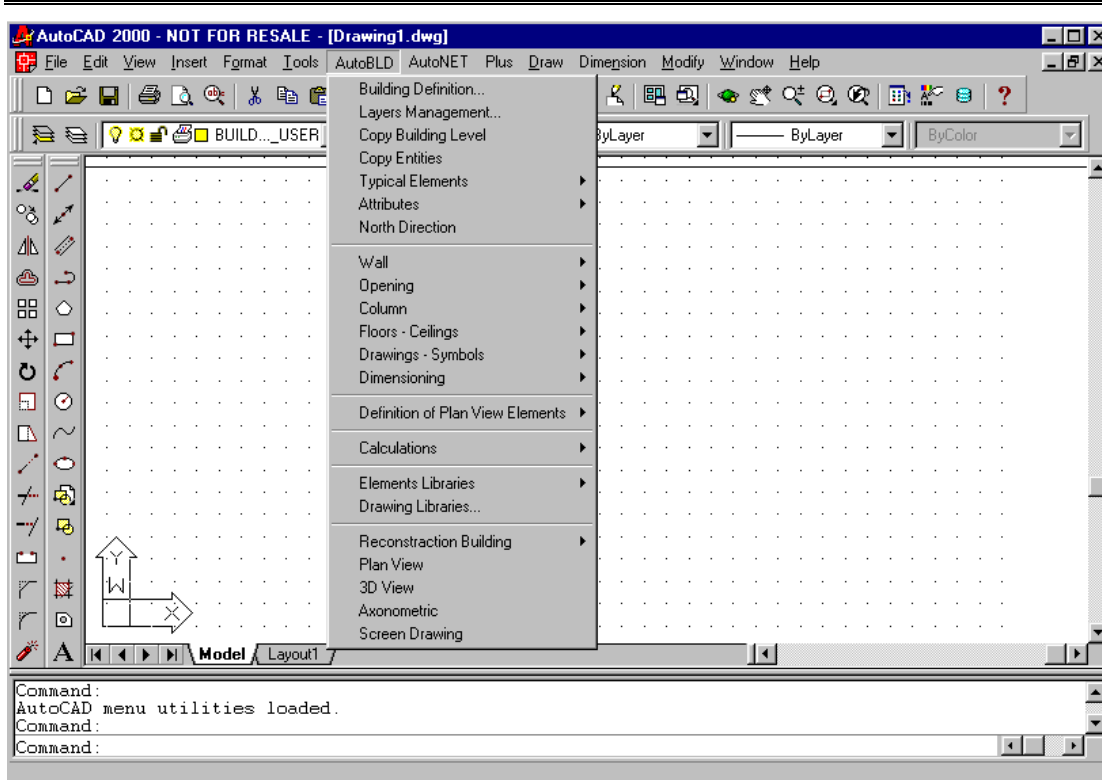
The Print dialog box is organized by tabs into two functional areas: scaling and viewing, and advanced printing options. For help defining print settings before you print, see Customizing print options.

To print a drawing

1. If necessary, click the desired Layout tab or the Model tab.
2. Do one of the following:
 - Choose File > Print.
 - 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 Print dialog box, make any adjustments to the settings.
4. Click Print.

2.4 AutoBUILD: Architectural Design

As already mentioned, AutoBLD group of commands includes all the necessary tools to shape the 3D building model and therefore the Architectural drawings. Those tools are organized in sub-groups.

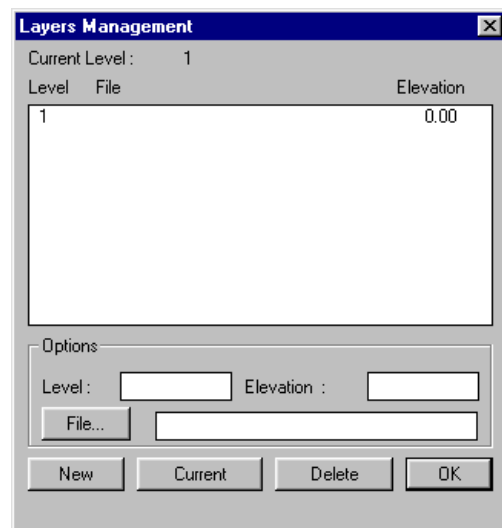


The first sub-group includes commands for the definition of the project parameters, the second sub-group includes drawing commands, the third sub-group commands related to the calculations, the fourth sub-group includes management options for the AutoBLD libraries and the fifth sub-group includes viewing commands of the building.

2.4.1 Building Definition

This window summarizes the building levels along with the files assigned to, in the case that the user wishes to import existing dwg drawings for the building levels (i.e. provided by the Architect). More specifically:

- In the "Level" field, the user must define the Level (floor) number.
- In the "Elevation" field, the height of the floor level (i.e. 3 meters).
- In the "File" field, the path and the name of the relevant DWG drawing-file, only if the user wants to refer to an already existing drawing, otherwise if he is intended to draw the architectural drawing from scratch he/she must leave this field blank.



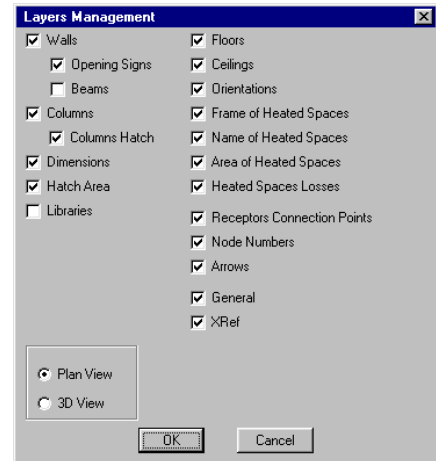
The insertion and the management of ground plans are performed with use of the xref command. At the bottom of the dialog box there are three functions available which are actually used to manage the floor files. More specifically:

- 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 desired floor. The "Delete" command removes the ground plan of the relevant floor in the project without deleting the original architectural DWG file.

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

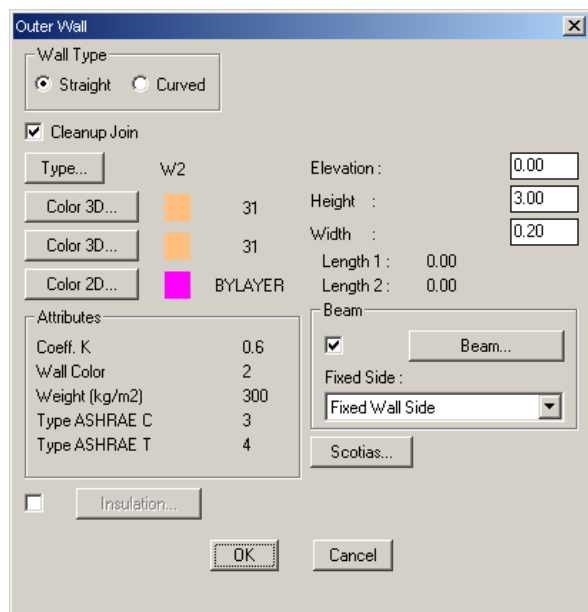
Thought the "Layers Management" option the user can disable any element group, by simply clicking inside the check-box of the respective group.



2.4.2 Drawing Walls

AutoBLD 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.

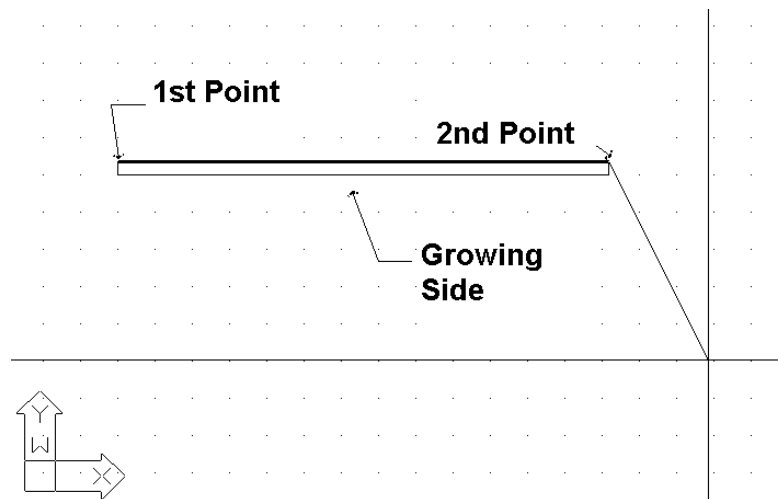
The **Wall** option, located at the second subgroup of the AutoBLD group of commands, includes the Outer, Inner, Outer wall from polyline, Inner wall from polyline and Outline options as well as the option subgroup Modify, Delete, Extend, Break, Join, Trim and Move. The first subgroup concerns the wall drawing, while the second with their further processing after being drawn. Finally, there is also the



Elevation of Merge Intersection option, which affects the view plan drawing presentation. By selecting Outer Wall, first of all its attribute dialog appears with a series of parameters (type, dimensions, colors etc). In order to start drawing a wall, you should 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. You can change the wall drawing from linear into circular, typing **T** in the following programme prompts and pressing <Enter>. During drawing, one can come to the conclusion that the ability of drawing consecutive walls is very convenient, since it saves the user 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 the user with 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 also 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 the user to reverse the previous command executed and b) the Properties command, which enables the user 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. Besides, at the bottom of this menu lies the option “Libraries”, which enables the user to define his/her own opening freely, to create various shapes of windows.

Window: The option "Window" demands that you 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 “Attributes”, namely the corresponding values for the height, the rize, 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 to the user 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 coordinates 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) are 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.

Besides walls and openings, 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 FineELEC.

The Building model of a FineELECproject 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: Electrical Installation Design

2.5.1 General Philosophy

The option group AutoNET includes all those tools the designer needs in order to draw (and then calculate) the cabling network of the electrical 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 colour to each network while the command "Linetype" is used to select the desired line type.

Applications Layers Management: This command leads to the adjacent dialog screen, where you can activate more than one mechanical installations and monitor those which are possibly overlapping (i.e. both Water Supply and Sewerage networks at the same time).

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.

Select Application: This option enables selection of the desired application of FineHVAC, the Water Supply or the Sewerage. Depending on the selected application, the section of the following AutoNET menu will be configured accordingly. 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.

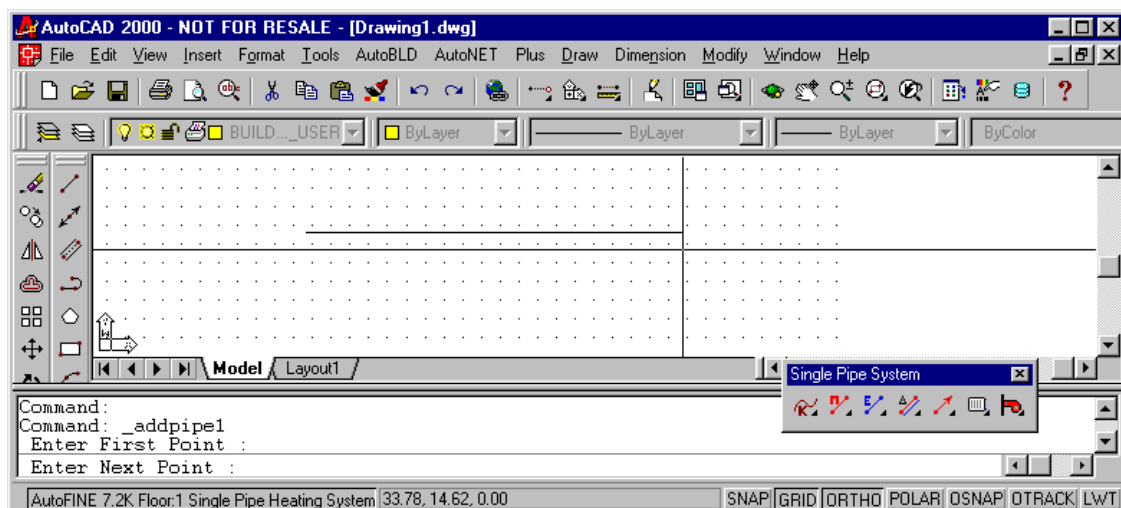
Horizontal & Vertical Cabling: In any case, the cabling drawing is carried out exactly as the line drawing in AutoCAD (or IntelliCAD). The user is able to draw horizontal or vertical network branches. Note that vertical branches are different from columns, which will be described below, as they are within the borders of the active floor and do not "cross" floors like columns.

The cable installation thickness (thickness of the line) is the current thickness. Modification of the cable installation thickness is possible through the AutoCAD (or IntelliCAD) command "elev". If you type "elev" (in the command line), you are prompted to determine the new current thickness. Press <Enter> if it is 0 or type 0 if there is another value but 0. At this point it should be emphasised that, if a horizontal cabling which is found on a specific level is drawn and it is connected to another cabling or a contact point (receptor or appliance), the program automatically "elevates" or "lowers" the cable so that connecting to the other cable or receptor, respectively, is possible. In this way, the programme facilitates the drawing of cabling in three dimensions while the designer is actually working in a two-dimension environment.

Caution! In case a horizontal branch is connected to a vertical one, "perpendicular" snap should be used so that there will be no more vertical branches created which would possibly complicate identification.

While designing a network, all facilities provided by AutoCAD or IntelliCAD can be utilised through relative co-ordinates.

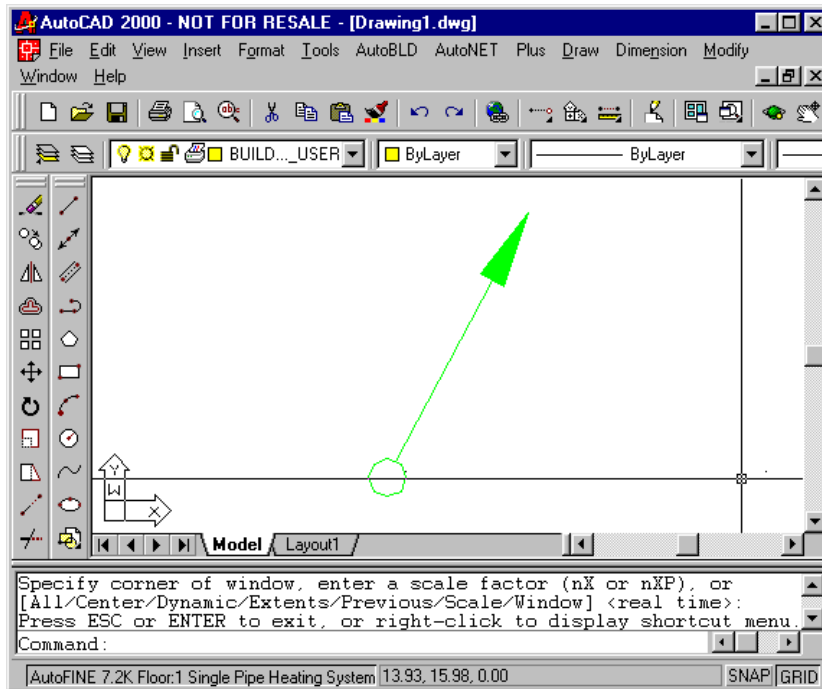
Example: Suppose you want to draw a horizontal network branch at the floor level which will be parallel to the wall (thickness 0). After selecting "Cable", use the mouse to define the first endpoint. Activate AutoCAD "ortho" to enable horizontal movements, as this facilitates the drawing of this example.



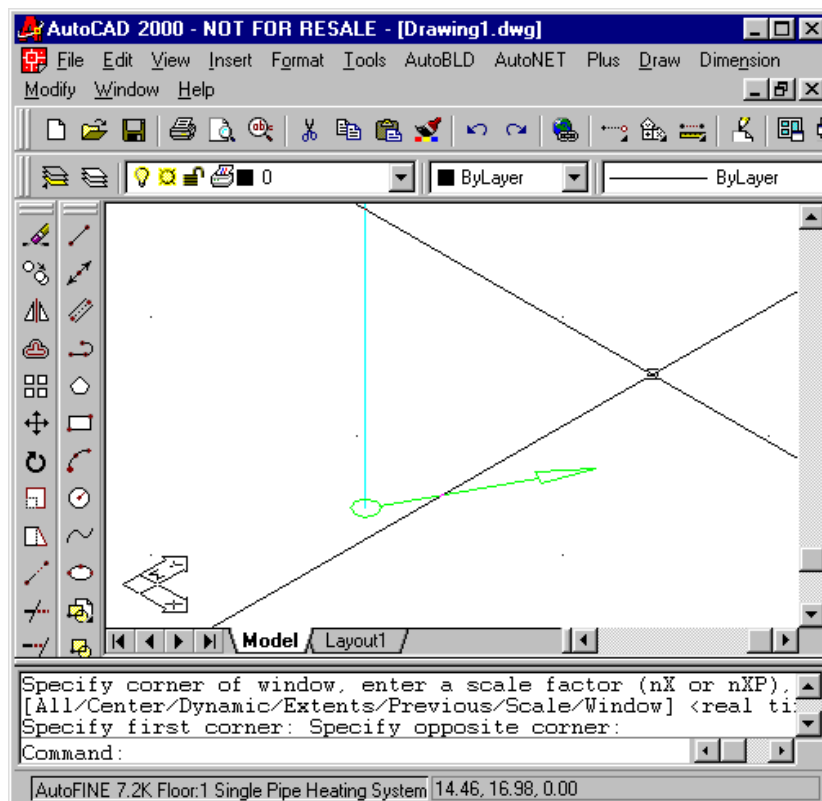
Move to the other side of the cable and left click to define the second endpoint. Right click to terminate the function of this command.

Column Drawing: Drawing vertical branches which cross floors (one or more) is possible through the option "(Building) Column". When the respective option is selected from the menu, the programme 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").

Example: Suppose you want to draw a vertical branch (column) from 0 to 3. If you insert the location point (XY) and then the numbers 0 and 3 successively, the symbol for direction change appears on the ground plan.



If "3D View" is selected, the column is displayed in 3 dimensions, exactly as drawn:



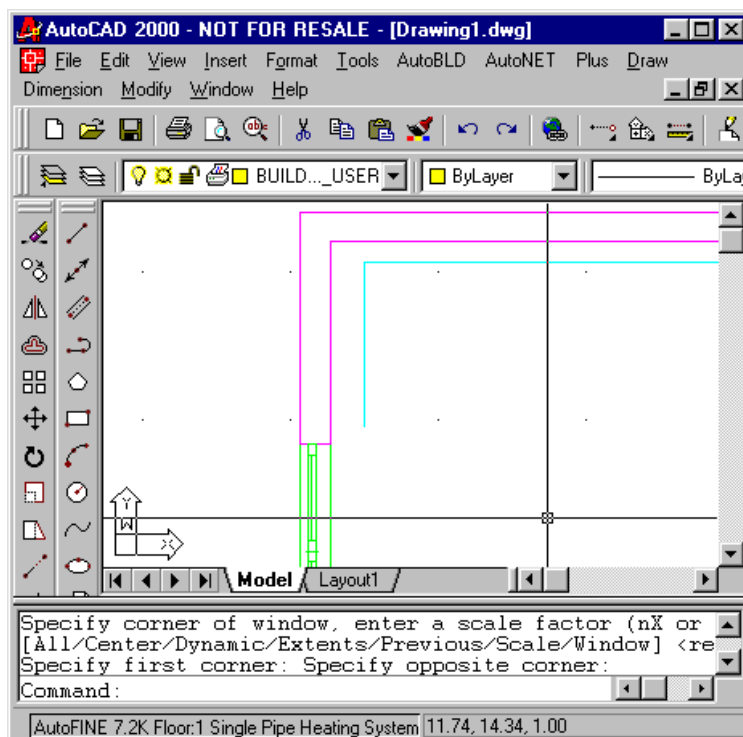
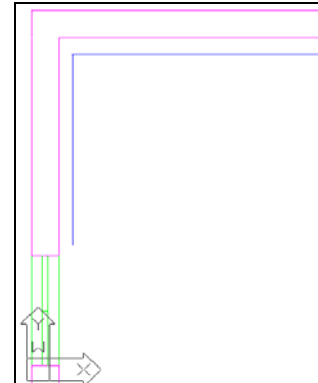
The programme automatically draws the symbol for a vertical column at the column trace, as shown in the image.

Note: The symbol is "block" with the name Mark1. After the symbol is placed, the user can modify it, draw any other symbol he wishes or "Replace Block".

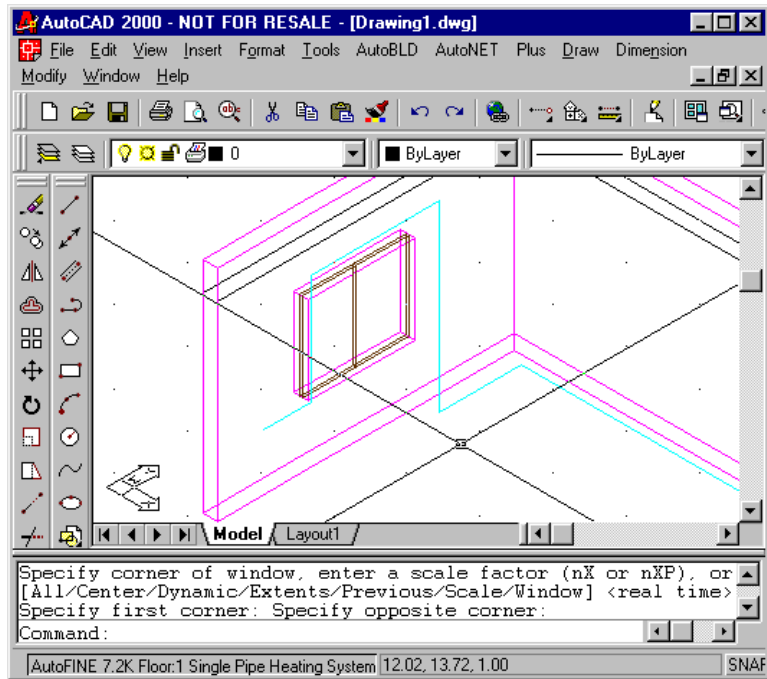
Vertical branches within the same floor: If you want to elevate or lower a cable within the same floor without having the elevation-lowering symbol inserted (Mark1), you can use the command "Cable".

Example: Suppose you want to elevate by 2.5 m from the free end of the cable shown in the adjacent screen, to continue over and along the window and finally descend by 1.5 m.

Run the command "Cable" and select the cable end through "Endpoint" snap. After that, in the prompt "Enter next point" insert @0,0,2.5 which means that the next cable end is found at the same co-ordinates x, y (relative co-ordinates 0,0) and height (z) 2.5 m.



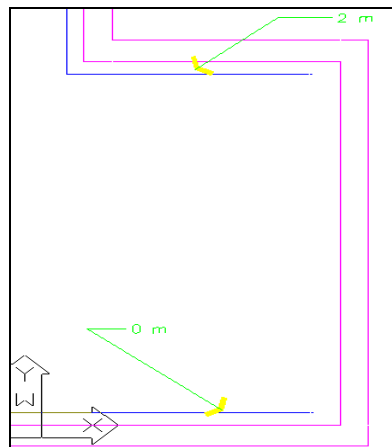
Then you can continue drawing the cabling, using the mouse to move towards the lower part of the window and "descend" from that point by typing @0,0,-1.5 as next end. This means that there will be a lowering of 1.5 m. Right clicking terminates the drawing of the cabling. For better monitoring of the drawing, run "3D View" command and the following screen appears:



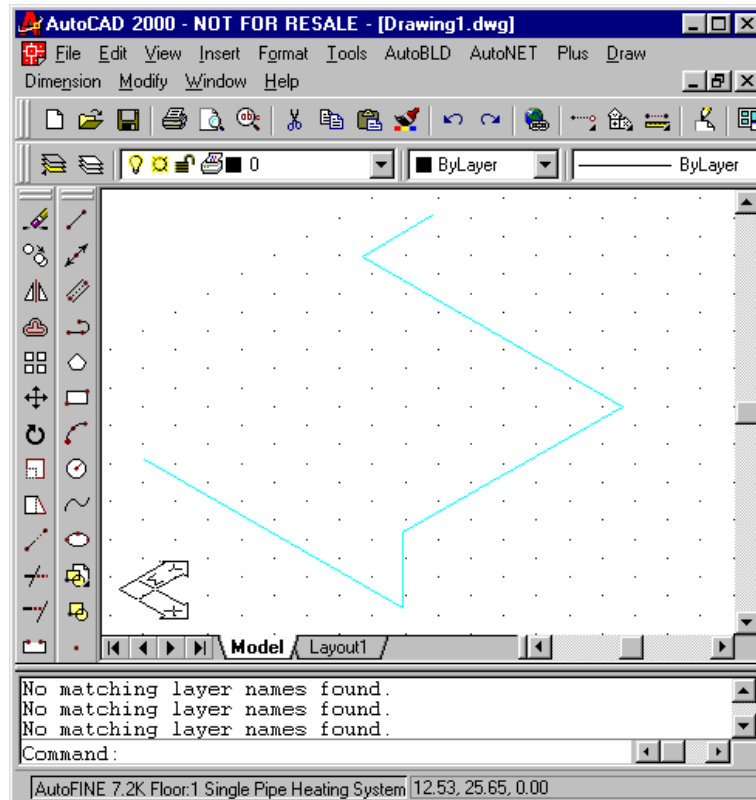
Note that you can draw cabling in 3 dimensions very easily, if you use the above drawing mode.

Connecting network sections: Connections between network sections (horizontal, vertical or both) as well as between network parts and receptors can be easily executed by using the AutoCAD "Snap" commands.

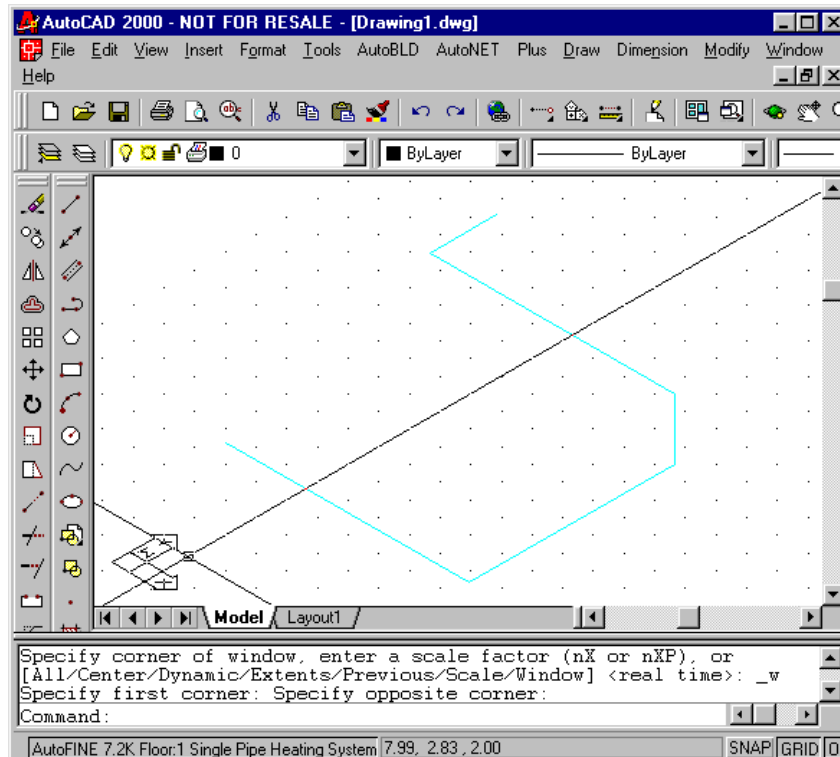
Example: Suppose that the two horizontal parts of the ground plan below, which are placed in different heights, have to be connected.



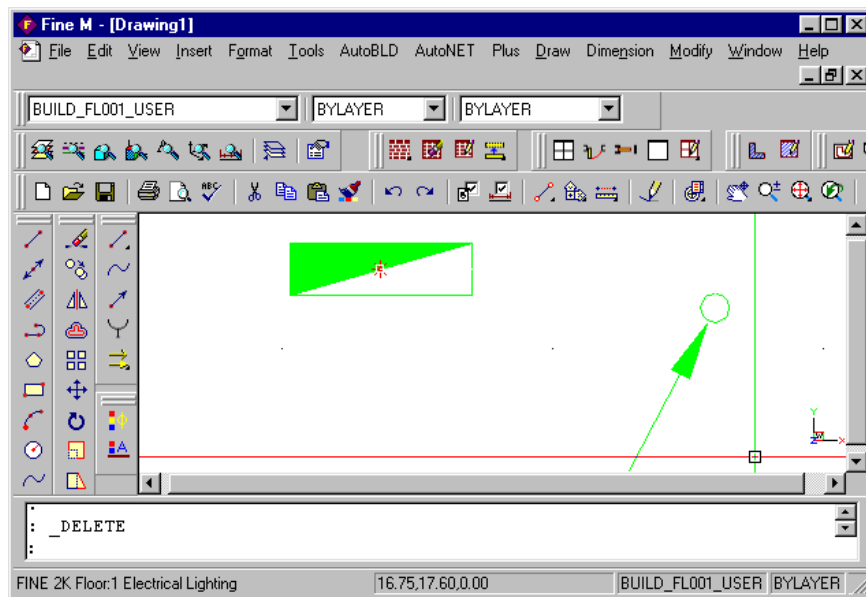
If you start by "grabbing" the "upper" cable end and then end up at the "lower" cable end, the result in the three dimension representation will be as follows:



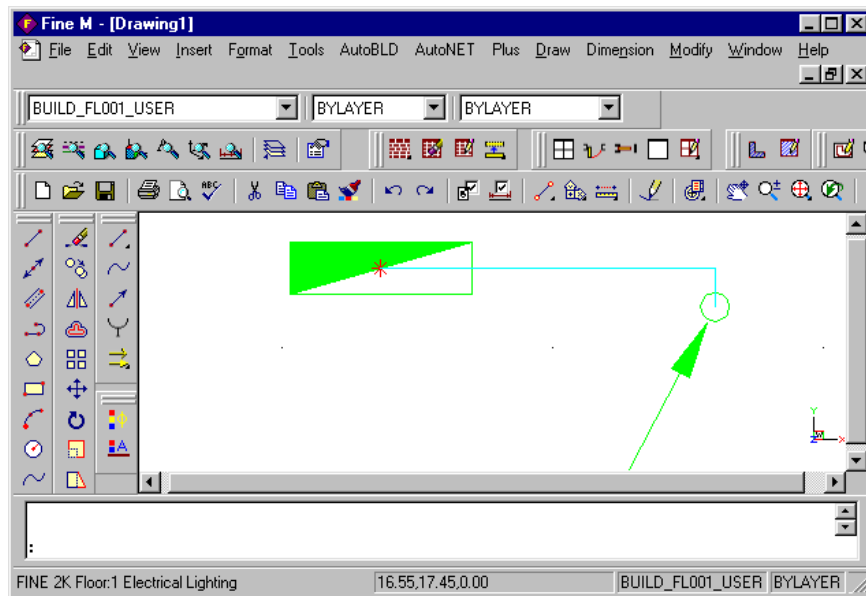
On the other hand, if the cable is drawn by using the "lower" cable as the first end and the "upper" cable as the second end, the result will be as follows:



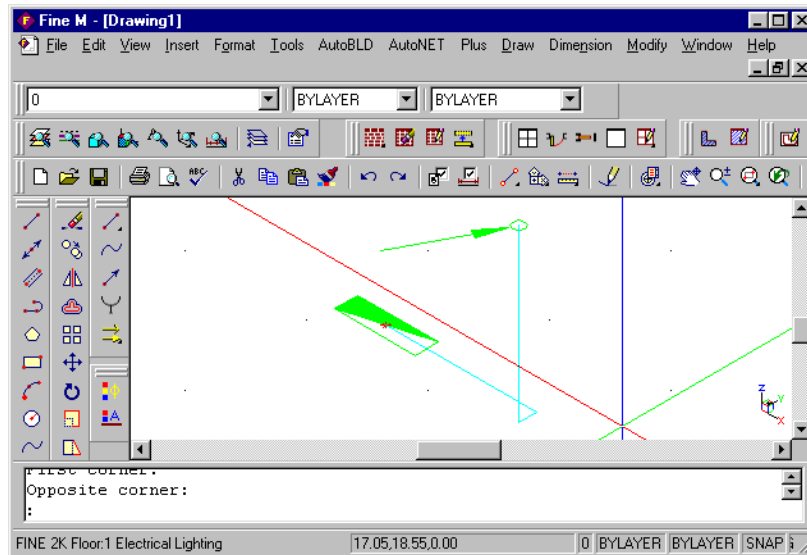
Example 2: Suppose the panel of the ground plan below has to be connected to the bordering column.



With the connection point as the first end (using a "connection point" snap), the second end in between and the third end on the column (using a "base point" snap) the section shown below is created:



In the vectograph, the result will be as follows:

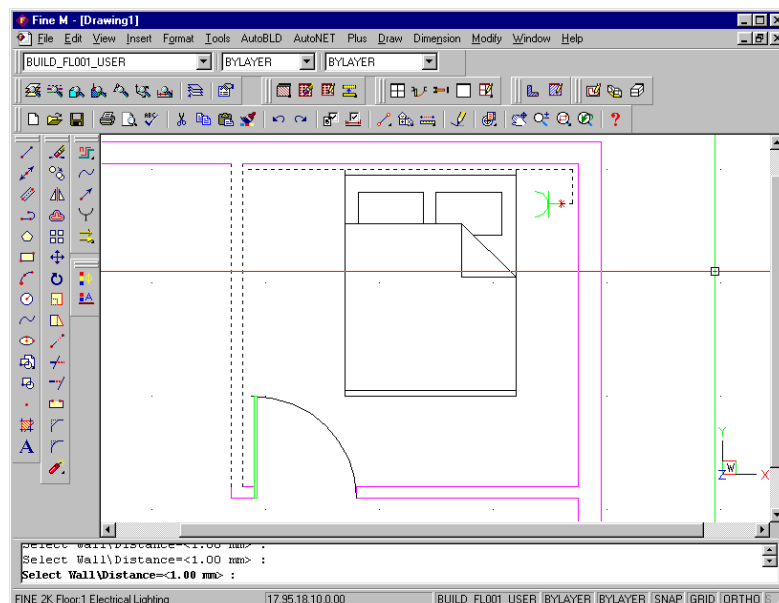


Attention! If an "endpoint" or "nearest" snap had been used for the column (in the ground plan above), the connection would not have been properly executed. It would have also been possible to work in 3 dimensions and enter a "perpendicular" snap by showing the column and therefore achieve a proper connection.

2.5.2 Special Commands for Cable Construction

This is actually a set of commands aiming at the facilitated drawing of the installation cabling. More specifically, there are two basic commands:

Cable parallel to Wall: A cable parallel to the wall (walls) marked by the user 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 cable is to be drawn. For instance, if the connection point of the tub is inserted as the first point in the ground plan detail shown below and then the three walls of the room are "marked", a cable parallel to these walls will be constructed.



The reason for that is that the program draws a vertical line from the first point to the parallel line defined by the other two points.

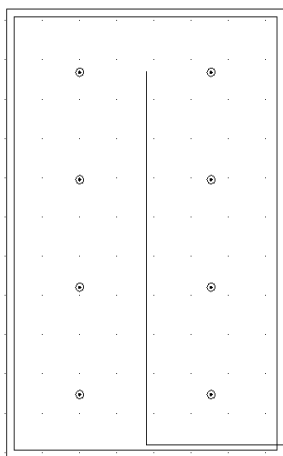
Cable parallel to Points: A cable is drawn parallel to the points defined by the user (supported by automatic snap), with a given distance from the crooked line defined by these points. The program asks for the first point and then for the other points (successively) parallel to which it is desired to have the cable drawn. When all points are inserted (and you right click), the distance will be requested.

Cable parallel to Wall (or Points) and Receptor Connection: This is a particularly useful command similar to the two commands above "Cable parallel to wall" and "Cable parallel to points", which, however, enable selecting the receptors to be connected on the routing (cabling or wiring) which will be drawn parallel to the walls or the points. Therefore, it is possible to connect a whole set of illuminators to the main panel, with just a couple of moves. More specifically, by selecting the "cable parallel to points and receptor connection" command the following options appear:

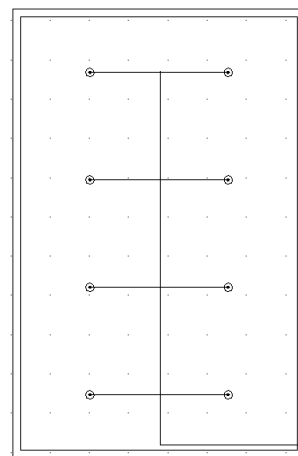
- *Select receptors:* Select the receptors to be connected to the cable applied in a parallel arrangement against the wall by defining certain points on the wall.
- *Enter the 1st point & Enter the next point:* Provide the points parallel to which you want to install the cable. The points are shown on the drawing with an X.
- *Distance from a point <1.00>:* Provide the distance in printing mm where the cable is going to be drawn starting from the inserted points.

The program draws the cable and connects it to the receptors (i.e. lighting bodies).

Connecting receptors to an existing line: This command enables you to select the



receptors to be connected to an already existing routing. This command requires to select the receptors and the line to get connected. The program draws network sections from the receptor connection point vertically on the existing line. Suppose that the spot lights have been placed on the view plan and you want to have them connected to a central supply cable.



Select the command "Connect receptors to an existing cable" and the following options will appear:

- *Select receptors:* Select the receptors you want to have connected to the existing line.
 - *Select a cable:* Select the cable to which it is desired to have them connected.
- Then the program will automatically connect the receptors to the central cable.

Multi cabling: This command supports the drawing of multiple parallel cables.

2.5.3 Modifying an existing network

The user can modify an existing network by using AutoCAD or IntelliCAD commands and the program will continue identifying this network. Thus, it is allowed to copy, move or erase etc a network section. The usefulness provided by the AutoCAD or IntelliCAD "grips" utility should be underlined (e.g. to move the mutual junction point of two cables etc).

During drawing the rules below should be followed: Cables supplying the receptors (electrical devices) should be connected to the touch points of these receptors. Obviously only one cable can be connected to a touch point. The connection with the touch points which appear as "stars" in the ground plan can be executed with the AutoCAD "snap" function which can be activated by clicking the middle mouse button (for a 3-button mouse) or by pressing Shift and the right mouse button (for a 2-button mouse). Therefore, if you press the middle button while drawing a cable, the screen shown next will appear, where the "touch point" is selected (NODE).

Cabling 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 identification procedure) will perform all checks and indicate the mistake and its location to the user. A necessary step before the "identification" is defining the point (a) where the network starts, that is the supply point (a). In reality, this point corresponds to the main electrical panel. This point appears on the screen as a square symbol (see figure).

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

1. Receptor Placement
2. Drawing the cabling columns
3. Drawing the horizontal cable sections
4. Defining the Supply point(s)
5. Network Recognition (Identification)

The program senses the receptor supplies by their features within the numeric libraries.

2.5.4 Receptors

The "Receptors" selection shows the screen including the receptors of each installation in the form of slides. Since the receptors of an installation can not usually fit to a screen, it is possible to move to following pages where additional receptors appear. Placing a receptor can be done simply through the following steps:

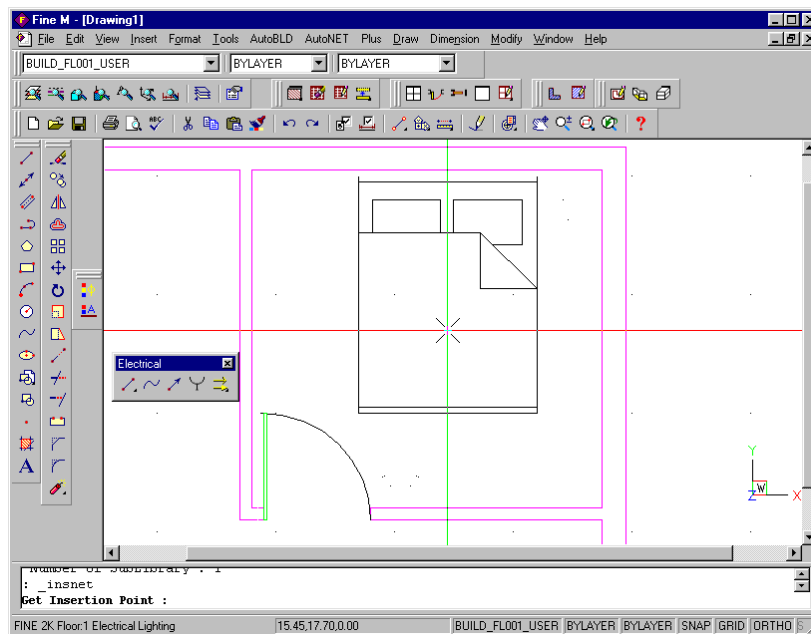
1. In the receptor screen select a receptor, press <Enter> and then press "OK" (or alternatively double click). Then it can be observed that the receptor moves on the ground plan with the graphic cursor.
2. If you move the mouse properly, the receptor can be carried in such a way that its base point (which coincides with the cross of the graphic cursor) can be placed in the desired point. Right click to confirm your selection.
3. If you move the mouse again, the receptor will rotate around the base point. Thus, if you confirm the angle in which you desire to have the receptor placed (again by right clicking), the receptor "freezes" in its final position.

The above procedure is quite similar to the block insertion procedure of AutoCAD or IntelliCAD. Another option related to the placement of the receptors is that: By checking properly the box upper-right on the window, the user can insert and place either the whole receptor or only its touch points (connection points) in the ground plan. This is significant 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 receptors is available for the calculations component.

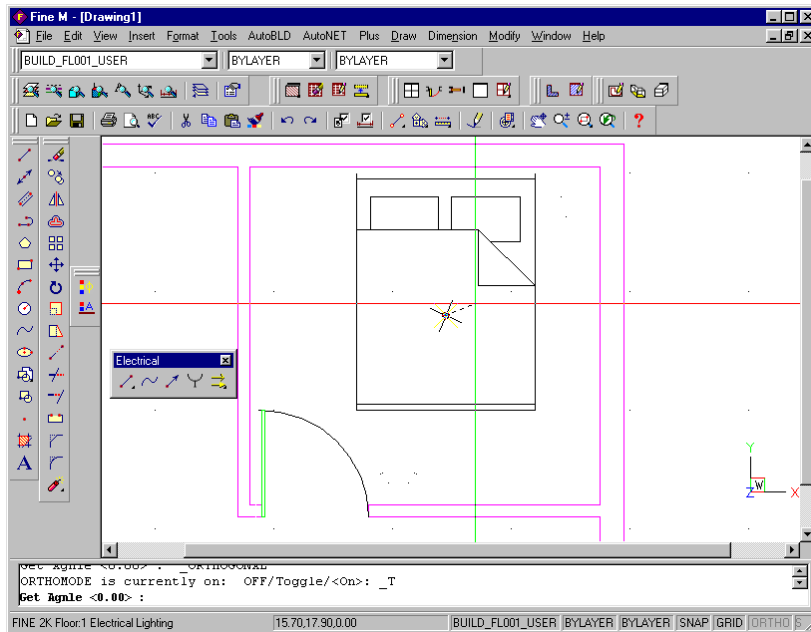
Note: Since names might be "cut" in the receptor slides, on the upper part of the dialogue box exists the indication "Current symbol" with the full name of the selected receptor next to it.

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 "Change Height" command.

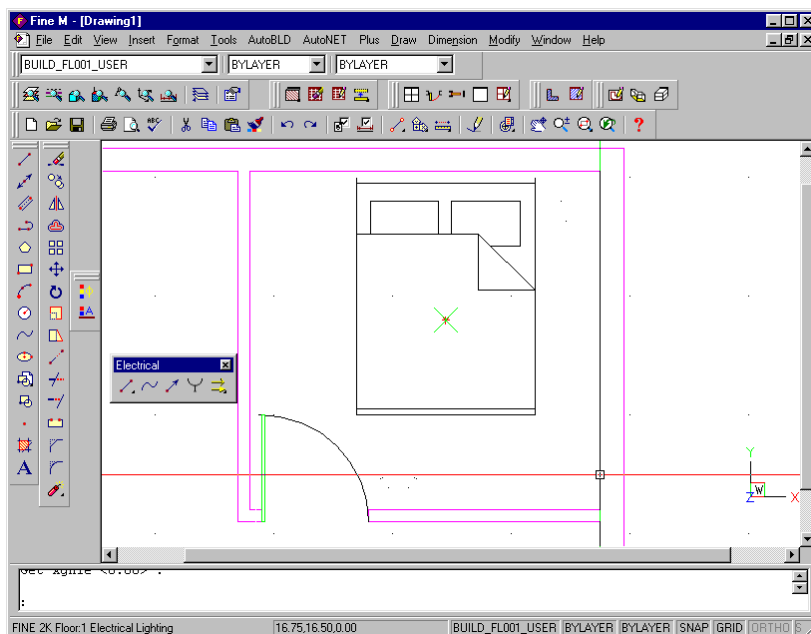
Example: Suppose that an illuminator has to be installed in 2.85 m height from the floor. After selecting "Change Height" or executing the "elev" command by typing it in the command line and inserting value 2.85, press <Enter> in the receptor screen "on" an illuminator and afterwards press "OK" (or alternatively double click). Then you can see the illuminator moving on the view plan together with the graphic cursor.



If the mouse is moved properly the illuminator can be carried in such a way that its base point (which coincides with the cross of the graphic cursor) will be placed in the respective point.



It can now be observed that if the mouse is moved, the illuminator rotates around the base point. Thus, if you confirm the angle in which you desire to install the receptor, the illuminator can be seen in its final position.



Two additional commands related to the receptors location are described below:

Installing many receptors in grid: It is also possible to draw grids (eg. from illuminators) thus producing detailed ground plan draws with limited moves. The same task can be executed by the "Symbols grid" command in the "Plus" menu but, in this case, the receptor should be inserted in the drawing in advance.

Automatic receptor installation: It is possible to install the receptors automatically in the ground plan and more specifically to install receptors at the space centre. Of course the user can proceed to any modifications (e.g. move, erase etc).

2.5.5 Symbols

"Symbols" include various general symbols, configurations and other drawings that can be used in the electrical installation.

2.5.6 General Symbols

The "General Symbols" option "opens" the general symbol library, which includes sub-libraries in which the user can save (and recall) various drawings. Just drawings without numerical attributes.

2.5.7 Network Recognition and Numbering

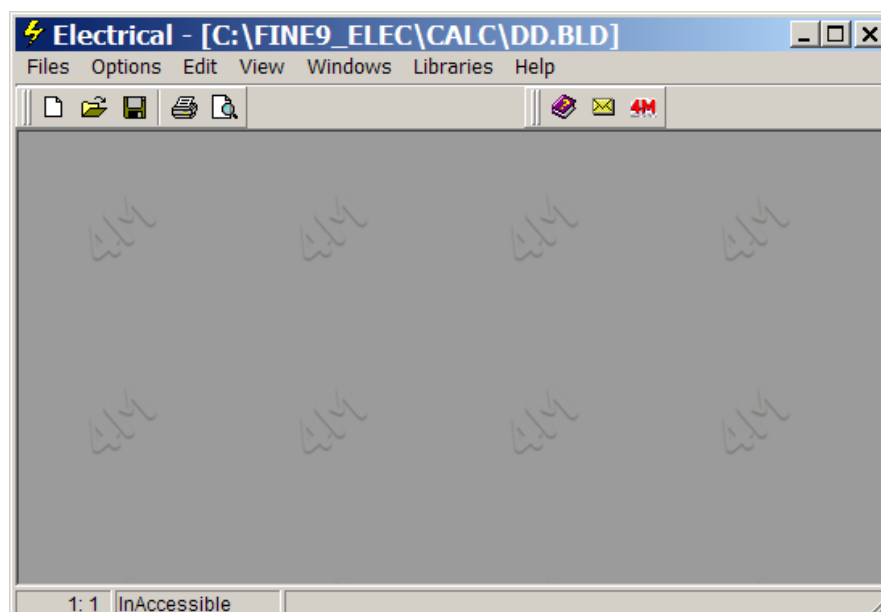
Once a network has been drawn according to the rules stated above and the supply point has been defined via the respective command, then the "Net Recognition" option converts the network into a mathematical model, which updates properly the calculation sheets. Through the recognition, junction points and receptors are numbered on the ground plan.

Notes:

1. If a receptor is not numbered, that means that the receptor is not connected to the network.
2. If a network section has a different colour, it cannot be connected to the network. Connect it or select "Break at selected point" at the connection point with the previous cable.
3. Junction points are numbered with font 4MS1 (included within the attributes of the block NODE.DWG).
4. If the user wants to move the number of a junction point or a receptor, he can do it with the grips.

2.5.8 Calculations

The "Calculations" option results to the activation of the calculation component of FineELEC.



For transferring the data from the drawings, the user has to select "Update from Drawing" in the menu "Files" of the corresponding calculating application and press "Yes" to the question "Calculate"). As a result, all the network sections along with their lengths, the receptors with their power values etc, are transferred within the calculation sheet.

2.5.9 Drawing Update

This option is used for the automatic updating of the ground plans with the required information from the calculations. This information is saved automatically in a determined layer in which the user can intervene, according to the above-described overall layer management.

2.5.10 Legend

The "Legend" option creates a legend (table) with all the symbols that have been used in this specific project. By selecting it, the program asks for the location to insert.

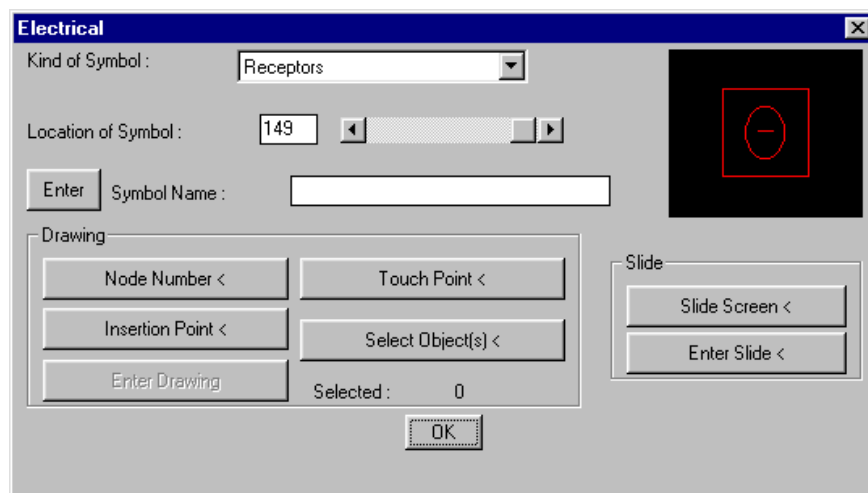
2.5.11 Vertical Diagram

This option creates two kind of drawings the "Panels Diagram" and "Panels Distribution Diagram". If the drawing exists then the program asks the user about updating it. All the drawings are editable by the user.

Note: For the automatic management of the vertical charts, FINE uses a specific file terminology. Particularly, vertical charts have names such as <ONOMA>DP.DWG, and <ONOMA>DD.DWG where <ONOMA> is the project name.

2.5.12 Library Management

This option leads to a submenu including the options "Numerical data" and "Drawing data". The first option shows the numerical data while the "Drawings" option opens the dialog box corresponding to the drawings:



Kind of Symbol: This the category type where the symbol belongs.

Location of Symbol: It refers to the position of the symbol, which you want to view or insert in the library, in the drawings as well as in the numerical data library.

Symbol name: This is registered in the "numerical data" libraries. There is a set of commands on the right side of the screen, which enable the insertion of a slide in a position within the library. More specifically:

Slide screen: The slide screen provides a dynamic zoom in the drawing. It initially carries out "zoom all" in the whole drawing. Afterwards it is possible to define the screen desired to be saved as a slide by using the zoom. This capability is particularly useful in case it is desired to insert a lot of symbols at the same time.

Enter Slide: This option enables saving the current screen as a slide. On the left side, a set of commands helps you to define a block and insert it in the library. It consists of the following commands which should be followed in the order given.

Node Number: Determine the position where the numbering of the receptor should be shown, as well as the font type and height (or simply press <Enter> twice after providing the receptor location).

Touch point: Insert and place the receptor touch points (connection points).

Attention! *If you do not want to have the touch points printed in the final drawings, the "touchpoints" layer should be de-activated (select "Freeze") in the "Layer Manager".*

Insertion point: Determines the point according to which the drawing will be inserted.

Select Object(s): Select which entities of your drawing will comprise the Block. It should be pointed out that, in case the symbol type stands for a receptor or an accessory, the receptor tpoints (touch points) as well as the receptor attributes should be also determined.

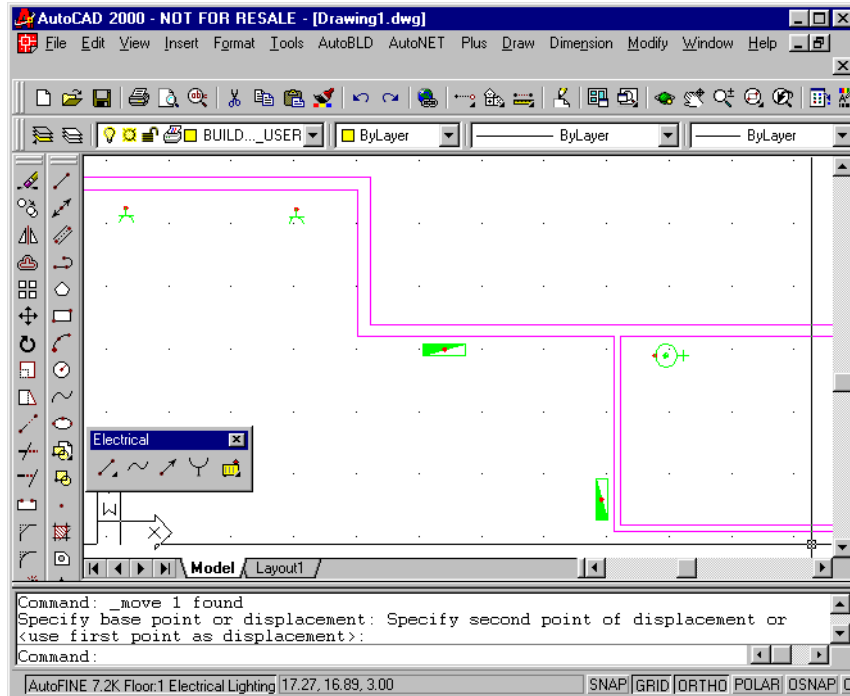
Enter drawing: Run this command to save the selected block in the respective library directory.

2.6 A simple example

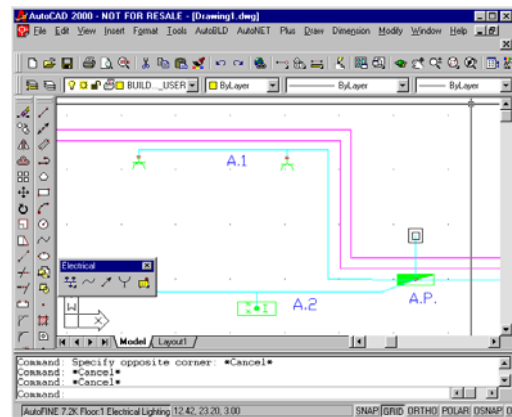
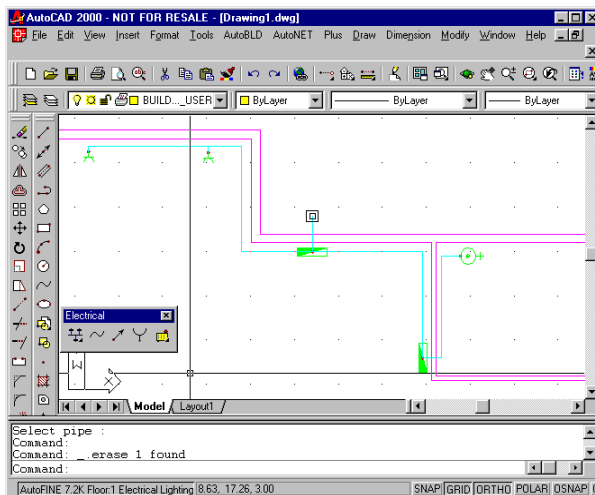
The example presented in this section helps the user comprehend in practice the contents of Chapter 4. As mentioned above, the steps according to which a standard electrical network is drawn, are the following:

- Select from the libraries and install on the ground plan all the load receptors (sockets, lights –standalone or in a grid etc), the panels and the other installation elements (switches etc).
- Draw the horizontal and vertical routes from and to the panels, according to the existing layout of connections.
- Define the point of supply of the main panel.
- Net Recognition
- Calculations
- Update Drawings
- Panel Charts - Distribution diagram

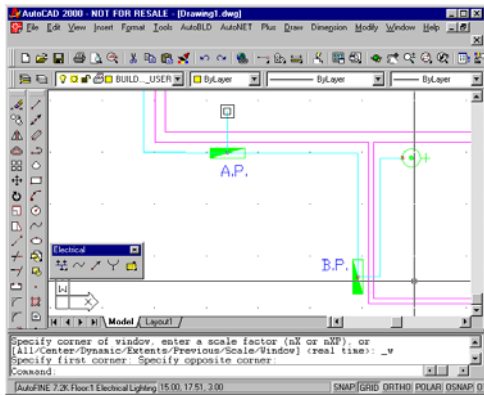
In this simple example suppose that you want to connect the panel of the following ground plan to a line that supplies the 2 sockets on the left and to a second line that supplies the sub-panel (on the right), the latter being connected to the hot water storage tank. After placing the receptors (also using the "elev" command to move to different levels), the ground plan is the following:



Run the “Cabling” command, which is completely similar to the “Cable” command of the example in section 5.1, to join the connection points (by elevating or lowering, wherever needed). Finally, the following ground plan comes up:



If you have defined a supply point and select “Net Recognition”, the network is identified and the panels are automatically “labelled” with letters, plus the letter P (for Panel), that is panel A.P (which is always the main panel), panel B.P and so on. In case some panels are not “labelled”, this means that they are not connected to the network, in which case you should check the connection points. If none of the panels is labelled, you should check if the supply point (network starting point) has been defined correctly.



As soon as the network is identified and you enter the calculation sheet, you will see those panels along with their lines (e.g. A.1 for the line connecting the two sockets, B.1 for the line connecting the sub-panel to the hot water storage tank and so on if there were more lines) and with all their data.

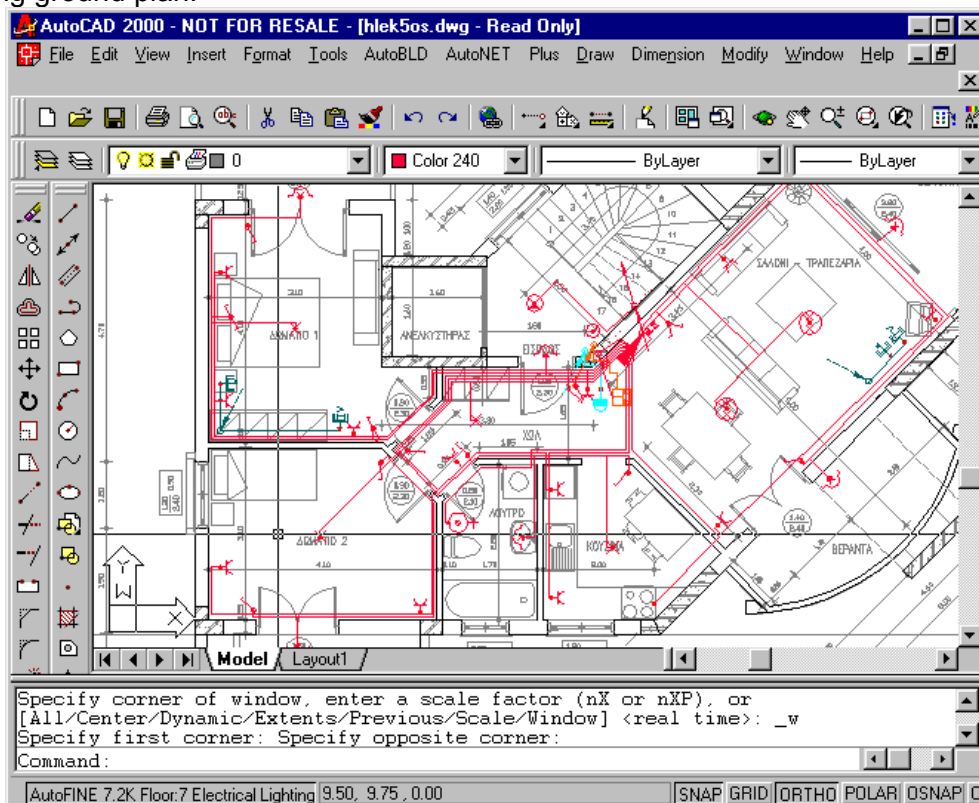
Return from the calculation sheet to the ground plan and run the command “**Update Drawing**”. The ground plan layouts of the Electrical installation are automatically updated, especially

the distribution lines, so that a full and immediate supervision of the installation network can be possible, in reference with the calculation sheet data and the panel charts.

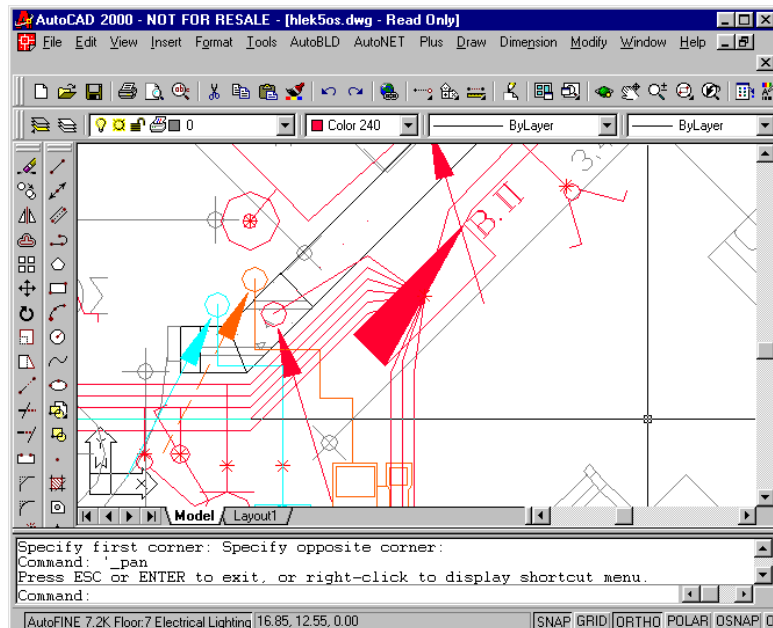
Two useful commands with respect to the final ground plan layouts of the electrical installation are the following:

- **Control Receptors from switch:** This command enables the user to quickly and easily attach the line number and the serial number of the controlling switch to lights and their switch.
- **Line Numbering at El. Receptors (Drawing Update):** Numbering at electrical receptors (e.g. B1, B2 etc) is performed by automatically placing the line numbers, similarly to the numbering of the other networks. This option enables the user to print the final layout of the electrical installation without including the wiring, which often renders the drawing complicated and illegible.

Working in a similar manner, you can draw larger networks, like the one depicted in the following ground plan:



In a large installation like the above, you can observe in detail how lines should start from the main panel (radially), so that clarity and increased supervision are possible.



Furthermore, the user should pay attention to the following details:

- Lines should start from the panel touch points and follow a completely separate route onwards (meaning that they are not allowed to have a common section or having a common starting section and then split).
- Lines between panels start from the supply panel touch point and end at the sub-panel touch point. Especially for the initial panel (A), the supply line, which also ends at its touch point, should start from any point (other than a touch point). This way, the panel supply line is actually drawn; the first point of which should be defined as the "Supply Point".
- The horizontal section that is connected to the panel touch point should be at the same level with the touch point. That is, no vertical section should exist at the touch point.
- In order for the panel and the distribution diagram layouts to be created, you should have created a DXF file for the panels and the charts respectively, within the calculating environment, (as in the "Single-pipe System" application).

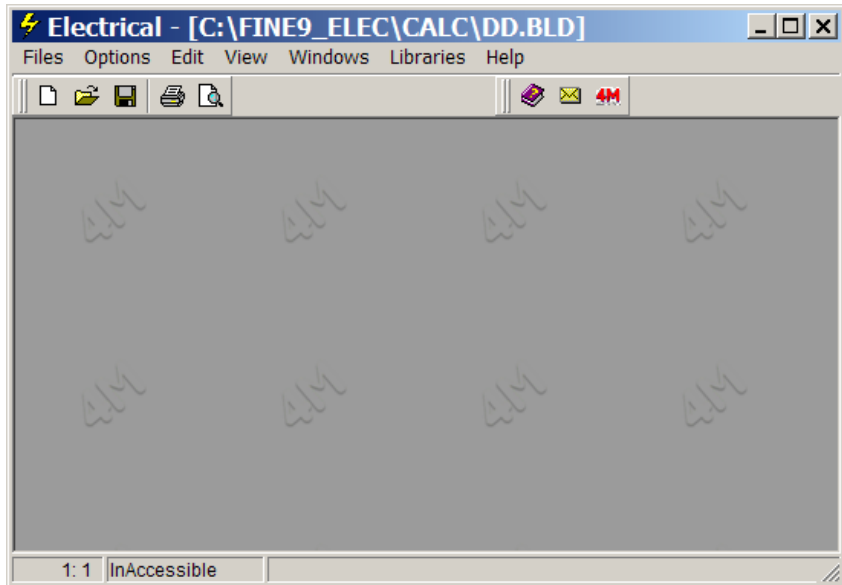
Finally, it is noted that the application outputs the error message "WRONG SYMBOL", when more than two cables end at a single receptor.

Notes:

1. The logical parameters - drawing commands are again defined in the "Autofine.ini" file (in the section \AFINE\14\w95), in the part of the text referring to Electrical Networks, with commands similar to those of cable networks.
2. In case you want to perform a "Weak Current" study, you should select the "Weak Current" project in the "Select Application" option, in order to have an independent layer management.

3. Calculations

This chapter provides a description of the Calculations Component of FineELEC. Each module can be used either independently, by filling the respective numeric data, or in cooperation with the CAD component of FineELEC, in which case the calculation environment acquires automatically the data directly from the drawings.



At the top of the application window appear the general options of each application menu, constituted of the group options "Files", "Options", "View", "Windows", "Libraries" and "Help".

3.1 Files

The "Files" option deals with the file management and printing options according to the known windows standardization. By summarizing:

New project: Type a name in order to save the new project in a file.

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

Update from Drawing: As pre-mentioned earlier, through this option the project calculation sheets are updated with the drawing data.

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

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

Restore 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", following the print preview output.

Printing Contents: You can select the project items you want to print, as shown in the respective window.

Printing Parameters: The desired printing parameters can be selected in this window according to the procedure already mentioned in Chapter 1.

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

Export to RTF, MS-Word and 4M Editor commands create respective text files.

Exit: With this command, the application stops running.

3.2 Options

This group of commands includes the global project data, which are divided into general data (project headings) and network data.

3.2.1 Network data

The network options dialog contains the main parameters of the installation. More specifically:

Network Options

Phase Circuit Voltage (V)

Max Voltage Drop (%)

Cable Type

Thermal Conductivity (S m/mm²)

Supply Type

Ambient Temperature (up to 50 °C)

NY Y Cables Location

Max Permissible Temp. of NY Y Cables (up to 85°C)

Protection Devices Calculations

Calculate Short Circuit Currents

Short-Circuit Time (s)

Calculation Method

Panel Loads Grouping for Coincidence

Cable Standardising

Clemens Standardising

Capability of Definition of Panels Different Lines (Arrival-Departure)

Maximum Cable than can be Used

Ok Cancel

Phase Circuit Voltage: This value refers to the difference of voltage between the wire and the neutral (phase voltage 230V). The polar voltage is automatically calculated by the program. The designer may modify this value either for the entire network (for each network line) or selectively (for a specific line) in the calculation sheet.

Maximum voltage drop (in the wires): It is about the maximum limit of voltage drop, according to which the cross-section areas will be calculated. The designer has the option to modify this value here too, either for the entire circuit or selectively, for a specific circuit line.

Cable Type The construction material (copper, aluminium etc) is defined, which affects the conductivity coefficient of the cables.

Thermal Conductivity (of the cables): Depending on the construction material, the conductivity coefficient is automatically inserted but it can be modified optionally by the designer.

Voltage supply type (single-phase or three-phase): In case three-phase voltage supply is provided in the network data sheet, there is the option to convert it into single-phase selectively for certain lines or panels within the calculation sheets.

Ambient Temperature (up to 50°C): It affects the maximum allowed current in the cables.

NYC Cables Position (1: Soil 2:Air): It affects the maximum allowed current in the cables.

Maximum Permissible temperature of NYC Cables (up to 85°C)

Protection Devices Calculations (1:by current, 2:by cable): One of these two ways can be selected, which can be specialised for use with the calculation sheets.

Calculation of short-circuit currents:

If **"Yes"** is selected, the program will additionally calculate the short-circuit currents for each circuit section. In order for the calculation of short-circuit currents to be performed, you should select the transformer type from which the circuit within the *"Substation"* window is supplied.

If **"No"** is selected, there will be no calculation of the short-circuit currents.

Short-circuit time (s): All the short-circuit current calculations will be performed according to this value.

Methodology of calculations: Following either the VDE or the HD 384 Standard.

Grouping of the panels' loads for coincidence: The two options «Subpanels» and «Analysis» affect the way in which the coincidence of the loads will be considered. As far as the first is concerned the subpanel loads of the main panels appear as "panel" and can be coincided totally. As for the second option, the subpanel loads are considered and shown in full details (lighting appears as lighting and so on).

Cable Standardization: Defines the way of appearance of the cable names.

Clemens standardization, between the options RST or L1L2L3.

Capability of definition of panels different lines (whether the arrival line is the same with the departure line from the central panel).

Maximum cable size that can be used

3.2.2 Cables (VDE)

Given that the selected standard is the VDE, then the following information on the cables of the specific installation has to be defined:

Single-Core Cables through the soil

- Cables arrangement (Linear or Trigonal)
- Total number of neighbor three-phase arrangements (1-4)
- Thermal resistivity of soil ($^{\circ}\text{C}\cdot\text{mm}/\text{W}$) (7-30)
- Operation (Continuous or Discontinuous)
- Installation Method (Free, Cover, Cover with sand)

Multicore Cables through the soil

- Cables arrangements (1-4)
- Thermal resistivity of soil ($^{\circ}\text{C}\cdot\text{mm}/\text{W}$)
- Operation (Continuous or Discontinuous)
- Installation method (Free, Cover, Cover with sand)

Single-Core Cables in the air

- Cable arrangement (Linear, Trigonal)

- Total number of adjacent three-phase arrangements (1-3)
- Transit (No correction, Soil, Closed trays, Open trays, Near Wall, Wall)
- Number of trays (Only for transit in trays) (1-6)

Multicore Cables in the air

- Air-Gap between cables & distance from the wall (Yes, No)
- Total number of adjacent cables (1-9).
- Transit (No Correction, Soil, Closed trays, Open trays, Wall)
- Number of trays (Only for transit in trays) (1-6)

3.2.3 Cable parameters (HD 384)

In case the user has selected to work according to the HD 384 Standard, then the following information on the cables of the specific installation has to be defined:

Insulated Cables

- Installation (laying) mode (In a pipe, inside the wall etc).
- Transit (on the surface of an element, on the wall or inside the wall, naked or in a pipe etc).
- Number of circuits

Multipole Cables

- Installation mode (naked in the wall, naked on the wall, inside the wall in a pipe, on the wall in a pipe, in a distance from the wall, on the ground etc).
- Transition (cables passing from the surface of a wall or through it, naked or passing through a tube etc)
- Number of circuits
- Number of carriers

Single-Core Cables

- Laying mode (in touch between each other- horizontal or vertical arrangement, In a distance among them- triangular arrangement, In a distance among them - horizontal arrangement, In a distance among them-vertical arrangement, on the ground etc).
- Transition mode (in a simple layer- in touch with a compact cable carrier, in horizontal perforated cable carriers on a straight line- in touch, in vertical perforated cable carriers on a straight line- in touch, in cable grills or wired grids on a straight line- in touch, no amendment etc)
- Number of circuits
- Number of carriers

Soil parameters

- Thermal Resistivity of Soil (°Cxmm/W)
- Soil Temperature (7 to 30 °C)

3.2.4 Typical Load Types

Through this option the user can define a set of typical load

	Load Type	Cos phi	Coincidence	Voltage Drop (%)	Connection Method	Line Type
1						
2						
3						
4						
5						
6						
7						
8						
9						
10						
11						
12						
13						
14						
15						
16						

types (optionally) to be used as templates. For each type of typical load – the name of which is inserted in the first column of the relevant screen - the following characteristics can be provided:

Load type: It is provided with a serial number (e.g. 1, 3 etc) which is related to the load types according to the standardisation used in the loads library (1:Lighting, 2:Socket, 3:Hot Water Storage Tank, 4:Cousine, 5:Motor, 6:Panel etc). The designer may consult the load library by pressing <F11>. There is also the option to define sub-categories of loads by adding a second number right after a dot is inserted (.) (i.e. 1.2, 3.2, 5.6 etc). For example, with the number 1.2 you may define a lighting load with fluorescent lamps and with the number 1.3 a lighting load with sodium lamps. The load type, as already mentioned in the libraries, "carries" the type of the drawing line too.

Cos phi: It refers to the cos phi of the load type inserted in the first column (the following auxiliary panel with typical values of cos phi appears on the screen).

HP	kW	cos phi	n%
¼	0.19	<u>0.7</u>	60
½	0.37	<u>0.7</u>	64
¾	0.55	<u>0.75</u>	69
1	0.75	<u>0.79</u>	74
1½	1.10	<u>0.8</u>	76
2	1.50	<u>0.8</u>	77
3	2.20	<u>0.8</u>	79
4	3.00	<u>0.82</u>	85
5	3.70	<u>0.82</u>	85
6	4.50	<u>0.83</u>	86
7½	5.50	<u>0.83</u>	86
10	7.50	<u>0.83</u>	86
12½	9.30	<u>0.83</u>	87
15	11.00	<u>0.83</u>	87
20	15.00	<u>0.84</u>	89
25	18.60	<u>0.85</u>	90

Typical coincidence: It refers to the coincidence of the load type inserted in the first column (by pressing F11 the following auxiliary panel appears on the screen, with the typical values).

Panel Type	Load type							
	Lighting		Socket		Thermal		Motor	
	from	to	from	to	from	to	from	to
Sub-panels	<u>1.0</u>		<u>0.6</u>	<u>0.7</u>	<u>1.0</u>		<u>0.7</u>	<u>1.0</u>
Main Panels	<u>0.8</u>	<u>0.9</u>	<u>0.4</u>	<u>0.5</u>	<u>0.6</u>	<u>0.8</u>	<u>0.6</u>	<u>0.7</u>
Main Panel	<u>0.7</u>	<u>0.8</u>	<u>0.2</u>	<u>0.3</u>	<u>0.3</u>	<u>0.5</u>	<u>0.5</u>	<u>0.6</u>

Auxiliary panel of Coincidence Coefficients

Voltage drop: It refers to the maximum voltage drop allowed corresponding to the load type that the first column is filled with and represents a % of the voltage supply.

Motor connection type: It is filled only for motors, provided that you want to define the starting method, in order for the cable cross-sectional area as well as the motor protection devices to be properly estimated.

Code	Starting type:
1	Direct on line
2	with Star-Delta switch

In case this field is left blank, starting is considered as direct-on-line.

Number of line phases: It is filled in order to specify whether the corresponding line is a single-phase (1) or a three-phase (3) one.



Notes:

- *It is not necessary or mandatory to insert values in all fields for each load type. As a result, the fields which have been left blank will be assigned either the generally defined values for the circuit (e.g. voltage drop) or the values in the libraries (cos phi) or finally they will be completed within the line calculation sheets and the panel calculation sheets.*
- *A great conveniency provided by the definition of typical data is the fact that every modification of such data, performed in any project phase, will affect the calculations correspondingly. In other words the user is provided with overall check over the project, since he is able to modify the circuit load parameters easily and quickly.*

3.3 View

This option follows the known windows standardization.

3.4 Windows

The "Windows" option includes the calculation sheet, as well as some other windows with important project results. In some cases, an additional basic option is automatically inserted in the main menu, named as the active window including some additional options.

3.4.1 Calculation Sheet

Each row of this sheet corresponds to a different circuit section while each column refers to data that will be provided or result automatically during the data insertion procedure.

Net Sec.	Line Length (m)	Load of the Line (kW)	Load Type	Cos phi	Des. Phase	Phase	Max Volt. Drop (%)	Voltage Drop V	Line Type	Des. Cr. Sect. (mm ²)	Cr. Sect. Calc. (mm ²)	Max Fuse (A)
A.P		13.52	6	0.966		123	2.500		3		6	25
A.B	16	6.761	6	0.966		123	2.500	1.269	3	4	4	20
A.C		6.761	6	0.966		123	2.500	0.000	3	4	4	20
B.P		6.761	6	0.966		123	2.500		3	4	4	20
B.1	11	1.6	1.1	1		1	2.500	1.905	1	1.5	1.5	10
B.2	14	1.4	1.1	1		2	2.500	2.121	1	1.5	1.5	10
B.3	7	1.550	2.1	1		3	2.500	0.705	1	2.5	2.5	16
B.4	5	2.5	5.1	0.8		123	2.500	0.235	3	2.5	2.5	10
C.P		6.761	6	0.966		123	2.500		3	4	4	20
C.1	11	1.6	1.1	1		1	2.500	1.905	1	1.5	1.5	10
C.2	14	1.4	1.1	1		2	2.500	2.121	1	1.5	1.5	10
C.3	7	1.550	2.1	1		3	2.500	0.705	1	2.5	2.5	16
C.4	5	2.5	5.1	0.8		123	2.500	0.235	3	2.5	2.5	10

Help instructions concerning data entering appear at the bottom of the screen (status bar). In each row, first of all you should fill in the fields of the first column which refers to each section coding.

Regarding the editing facilities of this calculation sheet, 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 has also the possibility to 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. In the spreadsheet below we can see columns having different widths.

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 by means 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, while:

- With the <Enter> key we move to the next cell below and so on.

-
- With the <Tab> key we move to the next cell at the right and so on.

Also, 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 key on a cell, the value it contains is deleted, and the cell is blank.
- **Deleting a row:** Pressing the keys <Ctrl>& 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 in the first column of the row we are in.
- **Moving to the end of a row:** Pressing the <End> key we move automatically in 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.
- **Freezing the first column:** Pressing the “stable column” command
- Finally, we 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.

Note that in order to fill in or select appropriate values you can activate dialog boxes in some positions in the filling in zone, using the keys <F11> and <F12>.

General editing commands described above (deleting cell content etc.) are also applied to these auxiliary windows. Finally, 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, (or simply mark the corresponding cell, proceed with copy & paste and finally press <Enter>). This 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).

Concerning now, the circuit designation, this is standardised as follows: The circuit panels and lines are named as below:

-
- Each panel and sub-panel of the network with one letter, two letters or a letter and a number (e.g. A ή AD ή C3 etc). Note that the main network panel is named A and that no panel designation should contain the letter "P".
 - Each line ending at a certain load with a number (e.g. line 1, line 12 etc). Note that if the line ends at a panel, it is not numbered (it is automatically defined by the pair of the two panel names).

At first the user should name the panels and the lines properly and then fill in the first column with the network panels and lines as shown below:

- Each panel is inserted with its name, a dot (.) and the extension "P" (e.g. A.P, B.P, C1.P etc).
- Each line is inserted with the panel name from which it starts, a dot (.) and the extension of the coding number (e.g. B.1, C.12 etc).

The simple example of the figure above makes the standardisation mentioned above easier understood.

Within the calculation sheet, you will define the panels A.P, B.P and C.P by naming them A.P, B.P and C.P correspondingly. The line between the panels A and C will be named A.C and the line between the panels A and B will be named A.B (**Attention!** The direction is important as it indicates that A supplies B), while the lines 1 and 2 which start from panel B will be filled as B.1 and B.2 correspondingly. Note that the panels and the lines should be filled in such a way that every line is placed under the corresponding panel. In addition, it is strongly recommended to provide the network data starting from the terminal panels in order to reduce the huge amount of calculations. Consequently, in the example above, the fields of the first column will be filled in succesively, from the top to the bottom, as follows: B.P, B.1, B.2, C.P, C.1, C.2, C.3, A.P, A.B, A.C).

According to the standardisation described above, the circuit loads are automatically summed up and calculated for each panel separately.

By entering the load type (column 4) into a section (column 1) the following columns related to the corresponding values are automatically filled:

(a) **cos φ**

Applies the value for load inserted in the typical data sheet or the value for the corresponding load type in the load type library. The designer can either insert it or modify it by typing in the relevant field.

(b) **the maximum allowed voltage drop**

Applies the value existing in general, or typical respectively, data sheet.

(c) **the line type**

Applies the value of load inserted in the typical data sheet or the value of the corresponding load type in the load type libraries. The designer may fill it or alter it by typing in the relevant field. Assign number 1 to a single-phase line or number 3 to a three-phase line.

(d) **the protection device type through the drawing line**

Note that the supply voltage of the specific section is the one provided in the general data sheet and, if needed, it may be modified by the designer after pressing <F8> in column 3.

These values may change, of course, whenever the user desires.

As for the calculations to be performed, the following data should be inserted for each line: a) the line length (otherwise the voltage drop of the line cannot be calculated), b) the line load, c) the line load type and d) the cos phi of the line load.

The load type is inserted in column 3 by defining a serial number according to the load type assigned to the load type library (The load type library is displayed if <F5> is pressed). There is also the option for an extension, such as 1.2 instead of 1, provided that the sub-category 2 of load type 1 (lighting) constitutes a load type sub-category with specific behaviour (cos. phi, coincidence coefficient etc) different from that of the load e.g. 1.3. In the case that the coding of the load type has been input into the typical load table of the project, then all the characteristics of the corresponding load "enter" automatically the row of the calculation sheet. Yet, a possible modification of them, within the typical lines sheet, will modify the corresponding values in the calculation sheets, affecting correspondingly all the circuit data (re-calculation).

As long as the values mentioned above are known, calculations are carried out automatically.

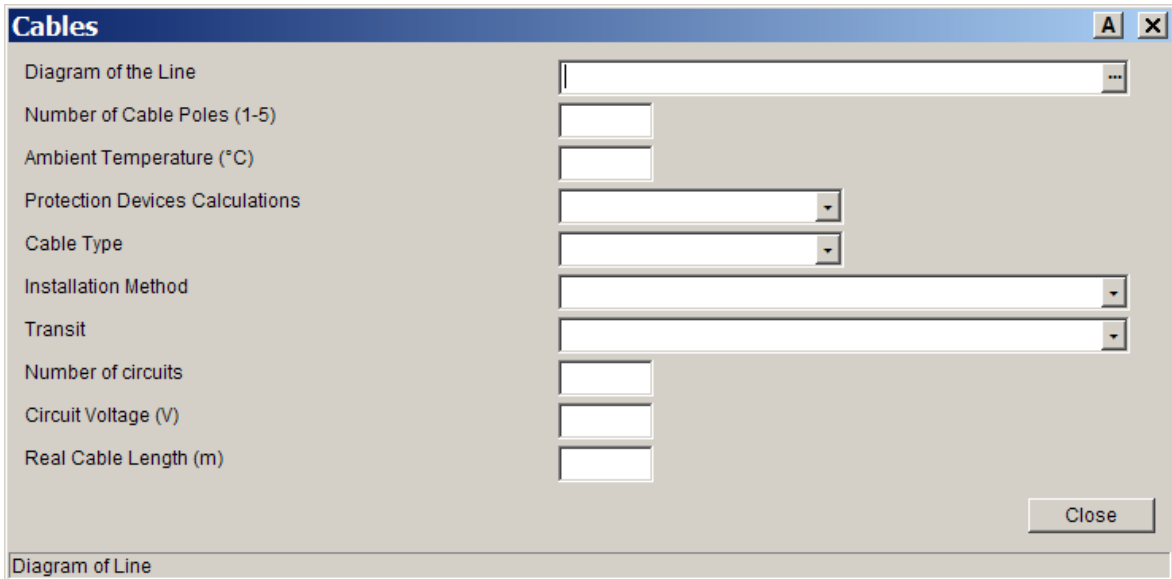
Calculation of cables and protection devices: The calculation of the cable protection devices can be performed based either on the current that flows through the line or on the cable cross-sectional area of the line cable. The selection of the calculation method can be assigned to each line separately, according to the designers demands.

For calculating the cable and the protection devices you should have in mind the parameters which are presented in detail below:

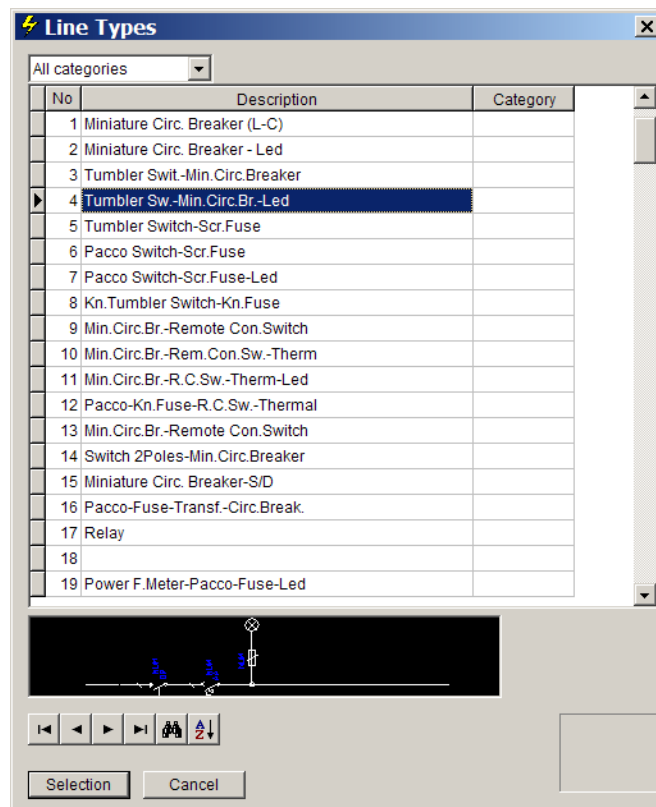
Parameter	Value
Circuit Voltage (V)	220.0
Real Cable Length (m)	26
Diagram of the Line (1 - 240)	26
Cable Type (1:NYA 2:NYM 3:NYN)	1
Number of Cable Poles (1-5)	3
Ambient Temperature (°C)	40
Cables Location (1:Soil 2:Air) (NYN Cables Only)	2
Max Permissible Conductors Temp. (°C)	70
Protection Devices Calculations (1:By the Current 2:By the Cable)	1

- The drawing line (completing this line is essential for calculating the protection devices)
- The cable type and its core number.
- The ambient temperature.
- The maximum allowed cable temperature
- The transition mean of the cable (air, soil)
- The arrangement and operation mode of the cable in its transition mean

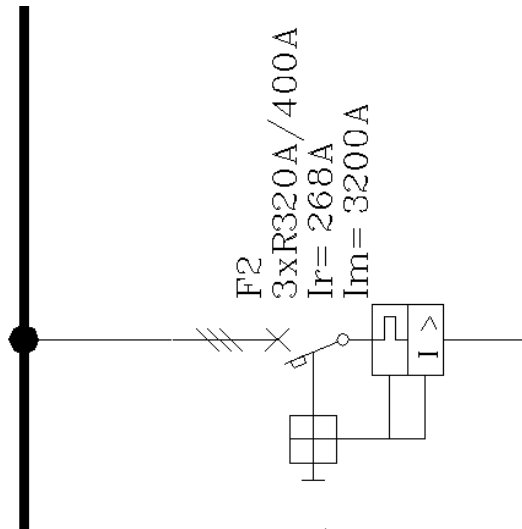
All the above parameters receive initial values from the typical data sheet and may be selectively updated, after pressing F5 (or by selecting the cables data from the list which is displayed a right click) so that the relevant window from the selected line is being activated. Then a window appears, summarizing all the data that must be filled.



As for the drawing line, you may view all the names of the library drawing lines, while “being” on a specific line, you can see, at the bottom, its drawing and its parameters which will get values during the calculations (e.g. AB, DP etc). In this window we can sort the data for an easier and faster seek of the desired line.



There are protection devices with a stable size (i.e. a screwing fuse) and other devices that give us the option to adjust their thermic or magnetic element, such as an automatic switch. In case that such a switch is selected, both adjustments (I_r and I_m) will be written down (see the following example where the indication "3xR320A/400A" refers to a switch of 400A, thermic element 320A adjusted to 268A (I_r) and magnetic element adjusted to 3200A (I_m)).



Core number of the cable: The user defines the number of the cable cores which supplies the load of the line. The usual number of cores used depending on the load type are the following:

- 1Ph-Lighting, 1Ph-Sockets → 3 cores
- 3Ph-Lighting, 3Ph-Sockets → 5 cores
- 3Ph-Motor → 4 cores


A crucial factor to the calculation is whether the load is ohmic or motion. If the load is a motor then it is calculated according to the following panel:

Motor		Direct on line start			Star-Delta switch start	
Power		Nominal current A	Slow Melting Fuse A	Cable Cross-sectional area mm ²	Slow Melting Fuse A	Cable Cross-sectional area mm ²
Kw	PS					
0,135	0,18	0,5	2	2,5	2	2,5
0,22	0,3	0,72	2	2,5	2	2,5
0,4	0,54	1,1	2	2,5	2	2,5
0,55	0,75	1,5	4	2,5	2	2,5
0,8	1,00	2,0	4	2,5	4	2,5
1,0	1,36	2,5	4	2,5	4	2,5
1,5	2,04	3,5	6	2,5	4	2,5
2,2	3	5,0	10	2,5	6	2,5
3	4,1	6,7	10	2,5	10	2,5
4	5,4	8,8	15	2,5	10	2,5
5,5	7,5	10,8	20	2,5	15	2,5
7,5	10,2	15,6	25	4	20	2,5
10	13,6	20	35	6	25	4
11	15	24	35	6	25	4
12	16,3	25	36	6	25	4
14	19	30	50	10	35	6
15	20,4	32	50	10	35	6
19	25,6	39	60	16	50	10
22	30	45	60	16	50	10
28	38	55	80	25	60	16
32	44	64	100	35	80	25
38	52	74	100	35	80	25
43	58	83	125	50	100	35
50	68	96	125	50	100	35
57	78	110	160	70	125	50
62	84	120	160	70	125	50
70	95	130	160	70	160	70
80	109	150	200	95	160	70
90	122	170	200	95	200	95
100	136	190	225	120	200	95
110	150	205	250-400A	120	250	95
132	180	245	315	120	250	120
160	220	290	400	150	315	150
200	270	360	500	185 or 2X70	400	185 or 2X70

If the designer ignores the fact that the load is a motion load and he wants his cable and device calculations to be performed based on the line current, all he has to do is to avoid inserting "no" in the "Motor" field in the load type window.

Voltage Drop in lines: For the calculation of the voltage drops, the following parameters are taken into account:

- The cable equivalent length which is calculated with the Moment Theory (directly from the drawing, in case "Net Recognition" has already been performed through FINE)
- The Resistance as well as the Reactance of the cable

 **Note:** In case data are typed (and not automatically identified through FINE), there are two types of line lengths: The effective length (for precise bills of cables) and the equivalent length (for voltage drop calculations). The equivalent length is the value provided in the second column of the calculation sheet while the effective length can be

inserted in the window mentioned above, which can be activated in each row by pressing **F5**. In drawing both are automatically updated. Regarding the line which supplies the initial panel A, in particular, provided there is a transformer and short circuit calculations are desired, the cable length between the transformer and the panel can be inserted in the second column.

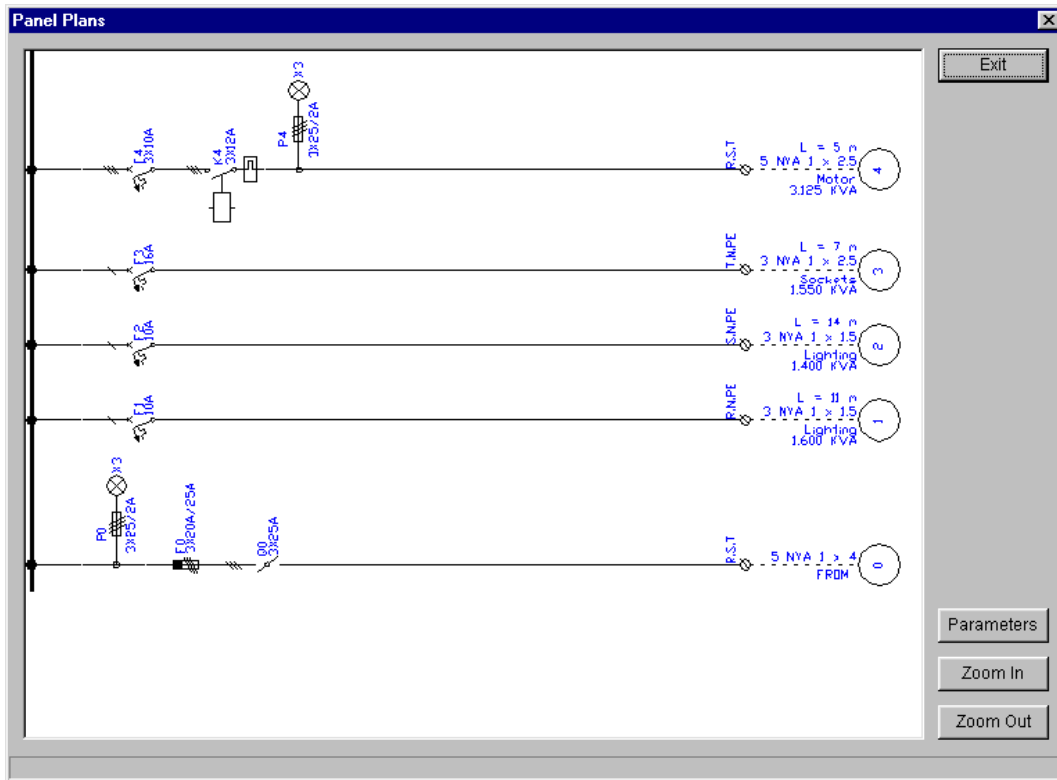
Apart from the data which are inserted directly by the user and those displayed in the calculation window, all intermediate data can appear on the screen as well (by pressing **<F9>**) to facilitate direct control.

Attention! In order to return to the previous mode, press **<F9>** in the first column of the sheet.

Net. Sec.	Line Length (m)	Load of the Line (KW)	Load Type	Cos phi	Cabl Type	No of Parallel Cable	Cr.Sect. Calc. (mm ²)	Des. Cr.Sect. (mm ²)	Permis Curren No.Cot	Corr. Factor	Permis Current (A)	Max Fuse (A)
A.P		13.52	Control Pane	0.966	NYA	6			35.00	0.820	28.70	25
A.B	16	6.761	Control Pane	0.966	NYA	4	4	4	27.00	0.820	22.14	20
A.C		6.761	Control Pane	0.966	NYA	4	4	4	27.00	0.820	22.14	20
B.P		6.761	Control Pane	0.966	NYA	4	4	4	27.00	0.820	22.14	20
B.1	11	1.6	Lighting1	1	NYA	1.5	1.5	1.5	16.00	0.820	13.12	10
B.2	14	1.4	Lighting1	1	NYA	1.5	1.5	1.5	16.00	0.820	13.12	10
B.3	7	1.550	Sockets1	1	NYA	2.5	2.5	2.5	21.00	0.820	17.22	16
B.4	5	2.5	Motor1	0.8	NYA	2.5	2.5	2.5	21.00	0.820	17.22	10
C.P		6.761	Control Pane	0.966	NYA	4	4	4	27.00	0.820	22.14	20
C.1	11	1.6	Lighting1	1	NYA	1.5	1.5	1.5	16.00	0.820	13.12	10
C.2	14	1.4	Lighting1	1	NYA	1.5	1.5	1.5	16.00	0.820	13.12	10
C.3	7	1.550	Sockets1	1	NYA	2.5	2.5	2.5	21.00	0.820	17.22	16
C.4	5	2.5	Motor1	0.8	NYA	2.5	2.5	2.5	21.00	0.820	17.22	10

More specifically, the following data appear for each line: length (equivalent length), load value and type, cos phi, cable type, calculated and desired cable cross section, current flow allowed through the selected cable at normal conditions, correction factor of the allowed current, allowed cable current at project conditions, selected fuse size and finally current flow through the line. View of all these data is extremely useful since it provides the user with advanced monitoring. Apparently, each modification made by the user is followed by automatic updating of the values in the calculation and result sheets.

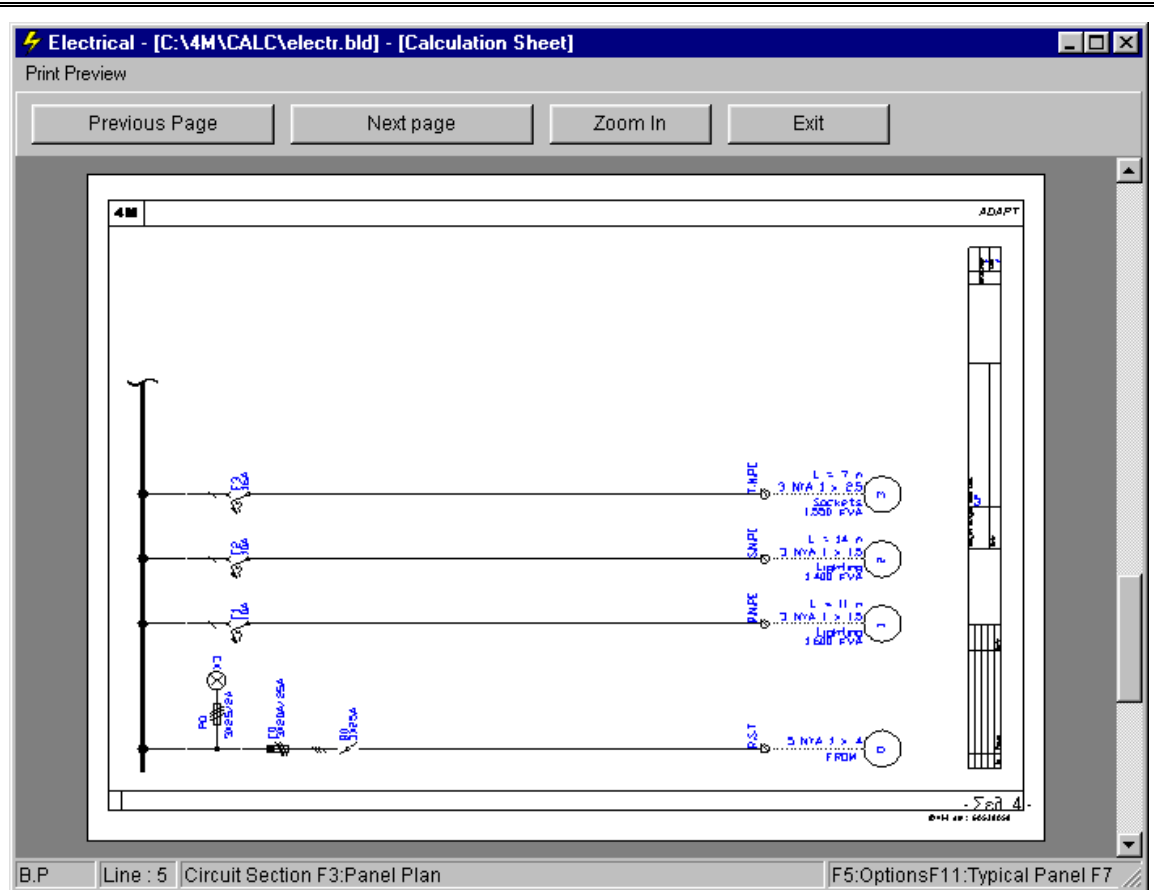
Automatic Drawing of Table Charts: As far as line drawing is concerned, press **<F3>** in the first column of the line you want to appear on the screen. In the same way, the whole panel can be displayed as long as you click a panel indication.



Furthermore, the user is enabled to modify a range of parameters concerning panel drawing. These parameters are listed below:

- Colours of drawing elements (bar, small circle, big circle, text, remaining circuit): For example, if you want to modify the colour of a selected text section, click the "Change Colour" key to select the new colour which is displayed on the window top right side.
- Spacing between panel lines. The default value is 100. Therefore if a lower value (e.g. 80) is inserted, the spacing is reduced accordingly while it increases for a higher value.
- "Label", showing the project properties, on the printing page or no legend at all, as shown in the following screen print preview.





Caution! The special case of high loads requiring extremely large cable cross sections (above 300 mm²) can be easily overcome. As soon as the user selects the desired cable cross section (e.g. 200 mm²), the programme automatically selects more than one such cables (e.g. 2 cables of 200 mm²). More specifically, to see that happen, select (by pressing **F12** in the column of the desired cross section) the particular cable cross section (of the same cable type) and then press **F9** within the calculation sheet. The required number of such cables is shown.

Note: The program allows the fast data entering in case of similar lines or panels. A similar (typical) line is repeated by filling its name in the first column (all data of the existing line appear, except the name which has to be typed). In order to insert a typical panel, you should type the panel name in the first column (e.g. B.P), select it and press **F11**. A list including the existing panels will appear, for you to select the desired one.

3.4.2 Panel Calculation

Based on the data entered in the line calculation sheets, all data are automatically calculated for each panel separately. More specifically, each panel corresponds to a page, which can be viewed by pressing **<F7>** in the first column (The appearing panel is the one corresponding to the row that the cursor is placed upon). To return to the calculation sheet, you should press **<F7>** again.

Load Type	Installed Power (kW)	Cos phi	Apparent Power (kVA)	Coincidence	Max Demand (kVA)
Lighting 1	3.00	1.00	3.00	1	3.00
Sockets 1	1.55	1.00	1.55	0.8	1.24
Motor 1	2.50	0.80	3.13	1	3.13
TOTAL	7.05	0.97	7.30		7.00

Phases Allocation	
Phase R (kVA)	2.64
Phase S (kVA)	2.44
Phase T (kVA)	2.59

Short-Circuits	
Short-Circuit Current (kA)	
Permissible Short Circuit Current (kA)	

Panel Symbolism	
Panel Name	B.P

Increments	
Due to Reserve (%)	
Due to Motors (A)	
Due to Lamp Starting (A)	
Total Current (A)	11.52

Recommend	
Main Switch (A)	25
Fuse or Circuit Breaker (A)	20
Supply Cable (mm ²)	4.0


Selected	
Main Switch (A)	12.01
Fuse or Circuit Breaker (A)	0.96
Supply Cable (mm ²)	4
Deegree of Protection of Panel IP	
Embedded to another Panel	1

In the top-right part of the window, a detailed name for the panel can be assigned (in addition to the symbolic name A.P, B.P etc mentioned earlier). More specifically:

In the top-left part of the window, panels are automatically calculated for each load type (that is, in case two or more lines refer to the same type of load, e.g. lighting, they are added):

- Installed Effective Power (kW)
- cos phi (in case more than one lines of the same load type carry different values, a weighted average is calculated)
- Installed Apparent Power (kVA)

Additionally, the maximum possible demand (in KVA) for each load type is calculated, using a pre-set (default) coincidence coefficient, equal to 1. This value can, of course, be modified by the user. Coincidence, after all, is the only field that can be modified by the user in the top part of the screen.

 **Note:** The user cannot modify the coincidence of a line, which load type as well as its coincidence are defined in the typical data sheet, since the remaining data are automatically calculated from the relevant values inserted in the line calculation sheet.

In the bottom-left part of the window, the following are calculated automatically:

- **Phases allocation** (R,S,T), that is, the loads in each phase (KVA), if we have to deal with a three-phase panel
- **Maximum Appeared Intensity of Current** (A)
- **Total Demand Factor**
- **Intensity for equal phases** (A)

- **Possible maximum Appeared Intensity of Current (A).** Note that the difference between this value of intensity and the previous one is a mean of evaluating the degree of equal allocation of the phases of the specific panel.
- **Short circuit current**
- **Permissible short circuit current**

The last two values will appear only if the option of calculating the short circuit current intensity is activated (in the network data sheet) and the type of the transformer that serves the network is selected.

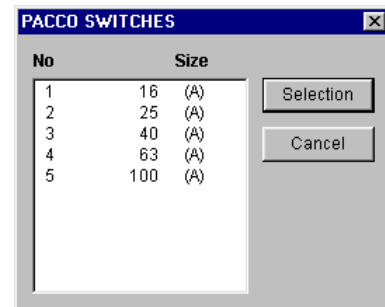
On the right side of the window, increments can (optionally) be given to the panel load, due to:

- **Reserve** (increment %)
- **Motor Starting** (in A)
- **Lamp Starting** (in A), when there are fluorescent lamps

From the above data, the final current of the panel (in A) is calculated right below.

Finally, the values of the following data automatically appear. They can also be selected by the user:

- **Main Fuse (A)**
- **Fuse or Automatic Switch**
- **Cross sectional area of supply cable (mm²)**



Degree of Panel Protection IP: The degree of panel protection represents its degree of protection against dust and water. It is specified by the letters "IP" and a two-digit number. The meaning of each digit, according to IEC 529, which is equivalent to the European standard EN60529:1992 is the following:

1st Characteristic Digit

Protection against contact and entry of solid objects.

- | | |
|---|---|
| 0 | No Protection |
| 1 | Protection against solid objects of diameter > 50 mm |
| 2 | Protection against solid objects of diameter > 12 mm |
| 3 | Protection against solid objects of diameter > 2.5 mm |
| 4 | Protection against solid objects of diameter > 1 mm |
| 5 | Protection against dust |
| 6 | Sealed against dust |

2nd Characteristic Digit



Protection against entry of liquids

- | | |
|---|---|
| 0 | No Protection |
| 1 | Protection against dripping water |
| 2 | Protection against dripping water in angles up to 15° |
| 3 | Protection of all sides against water |
| 4 | Protection of all sides against high-pressure water |
| 5 | Protection against blasting water |
| 6 | Protection against salt water. |
| 7 | Protection against dipping in water |
| 8 | Protection against sinking in water |

By pressing **F11**, the help window appears, indicating the most common degrees of protection. The user can select the one that is the most appropriate for the case in question.

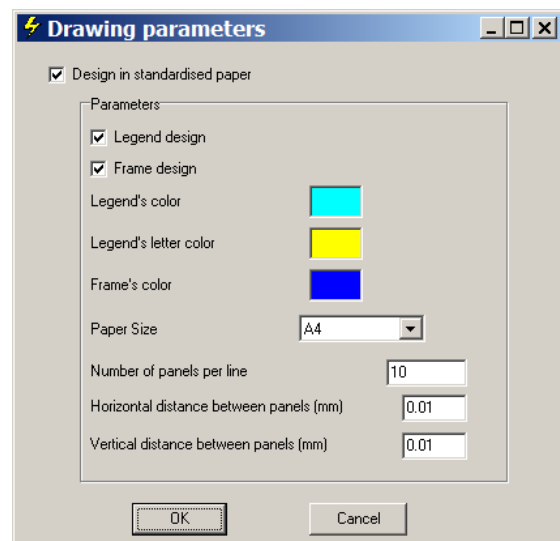
- **Embedded to Other Panel:** The indication “Embedded to Other Panel” (1:No, 2:Yes) exists, so that the cases where a panel is contained within another one (bar) can be dealt with. If the user specifies, at this point, that the panel is embedded, it will be automatically supposed that the panel does not actually exist, but it is a sub-bar of the panel supplying it. In a linear drawing, it will not be viewed independently, but together with the supplying panel.

The user can modify this data, always providing, however, safer values. Values are generally inserted by pressing **<F11>** and selecting them from the corresponding table that appears on the screen.

 **Note:** The user can see the panel line, the cable attributes and the panel attributes, by clicking on the toolbar  instead of using the keys **F3**, **F5** and **F7** accordingly.

3.4.3 Export to a DWG or DXF file

As long as the calculation sheet is activated, the user can select “Export to DWG” (or “Export to DXF”) through the “Edit” group of commands to generate a dwg (dxf) file including the panel drawings. Besides, the user has available a set of parameters to determine the form of the panel drawings generated (paper size, distances, density, colors and others).



3.4.4 Substation

As far as the selection of the substation (if required) is concerned, the values that appear on screen should be properly filled. Then the required load is automatically updated, whereas the rest of the values are automatically updated after selecting “Transformer Type” from the libraries.

3.4.5 Fan

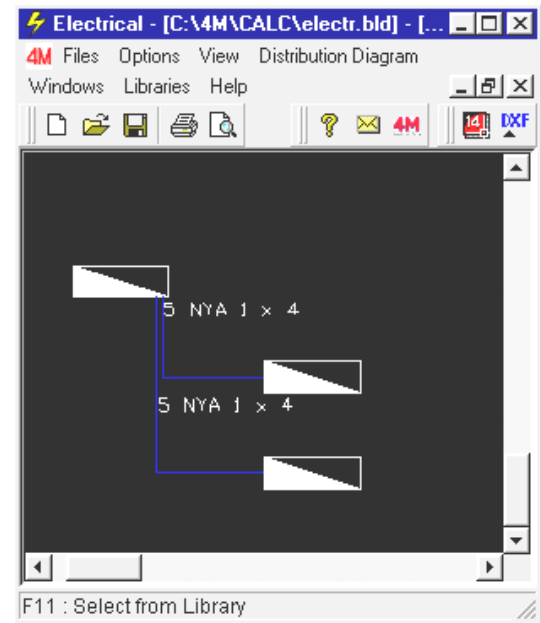
In order to select the fan for the ventilation of the substation, if required, the following window needs to be filled. By pressing **F11** in the row “Fan Selected”, the catalog of the library fans appears, which can be used to select the fan that satisfies your requirements.

3.4.6 Block Diagram

This command refers to the block diagram (distribution diagram) of the panels, showing the names specified in the panel calculation sheet. If a substation is determined, then the symbol of the transformer appears, as well. By activating the window, the option "Diagram Distribution" also appears in the menu, where the user can specify the printing scale and export the drawing to DWG and DXF files.

3.4.7 Voltage Drops

The total voltage drops appear in detail, for each route of the electrical network (from the main panel to each final consumption point). The value of the voltage drop is provided in Volts and in % percentage.



3.4.8 Bill of Materials

This window shows a detailed list of the materials used in the project along with their quantities, the total costs and the grand total. This list is editable by the user within a spreadsheet-like environment.

3.4.9 Technical Reports, Assumptions, Cover Page

The above three windows support the generation of some important project reports. The user can choose among different prototypes (templates). In particular, by selecting "Technical Description" for example, the corresponding window appears, updated with the project's results. When this window is activated, an additional option in the main menu also appears named "Technical Description". Choosing "Prototype Selection" from this menu the prototype management window will appear on our screen along with the list of the available prototypes for the application we work with. Selecting the corresponding prototype (by using the "Load" key), the relevant text appears inside the Technical description window, completely updated. Any desired modification in the technical description of the project, either by changing the position of a word-parameter or processing the text the way we want to, can be done by pressing the "Edit Prototype" icon.

3.5 Libraries

The libraries consist of the following groups:

Type of Cables

No	Description	Type	Insulation	Description by VDE	Nominal Voltage	Armouring Type	Exterior Cloak Type
1	H07V-U	Insulated Cable	PVC	NYA			
2	H07V-R	Insulated Cable	PVC	NYA			
3	H07V-K	Insulated Cable	PVC	NYA			
4	A05VV-U	Cable	PVC	NYM			
5	A05VV-R	Cable	PVC	NYM			
6	J1VV-U	Cable	PVC	NYN			
7	J1VV-R	Cable	PVC	NYN			
8	J1VV-S	Cable	PVC	NYN			
9	H05V	Insulated Cable	PVC	NYA			
10	H05VV-F	Cable	PVC				
11	H03VV-F	Cable	PVC				
12	H03VH-H	Cable	PVC				
13	H05RR-F	Cable	PVC				
14	H07RN-F	Cable	PVC				
15	XLPE/PVC	Cable	XLPE EPR				
16	NYCY	Cable	PVC				

Correspondences

New Type	Old Type
H07V-K	NYAF
H07V-U	NYA (re)
H07V-R	NYA (rm)
A05VV-U	NYM (re)
A05VV-R	NYM (rm)
A05VV-F	NYMHY
A03VV-F	NYLHY (rd)
H03VH-H	NYFAZ
H05RR-F	NMH
H07RN-F	NSHou
J1VV-U	NYN (re)
J1VV-R	NYN (rm)
J1VV-S	NYN (sm)
A05VVH3-U	NYIFY

Symbols for Cables

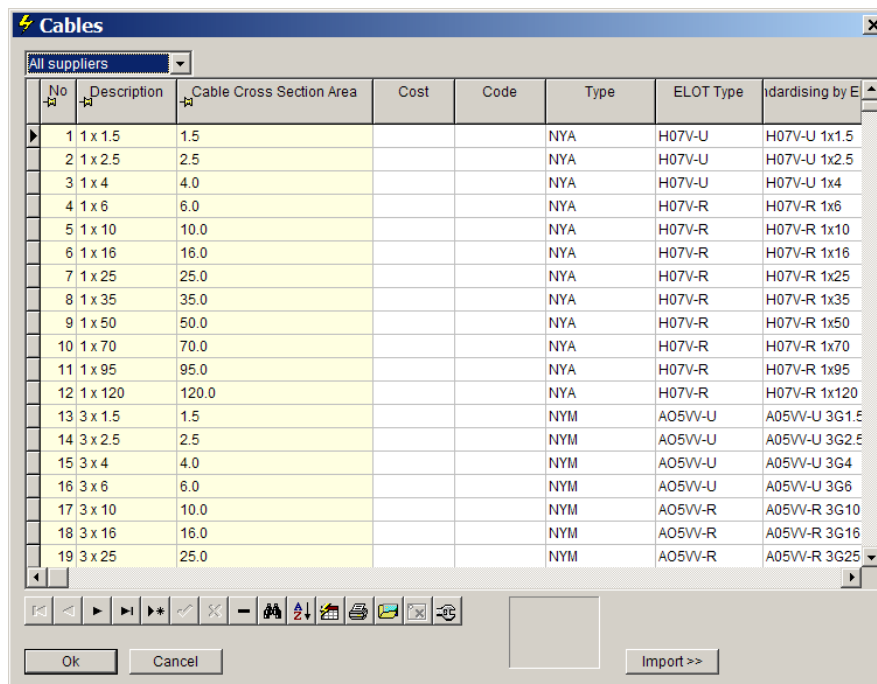
General Specification	
Harmonized regulation	H
Recognized national type	A
Cable according to IEC	J
Voltage u/U	
300/300 V	03
300/500 V	05
450/750 V	07
600/1000V	1
Insulation	
PVC (polyvinyl chloride)	V
Natural Rubber	R
Silicon Rubber	R
Screening	
Cooper tapes	Z4

Outer sheath	
PVC	V
Natural rubber	R
Neoprene	N
Fibre-glass tissue	J
Tissue	T
Tissue with a flame retardant material	T2
Manufacturing peculiarities	
Wide separated line	H
Wide non separated line	H2
Strings for filling the gaps	D5
Conductor	
Single core	U
Multicore	R
Thin & multicore, lines of permanent installations	K
Thin & multicore, very flexible	F
Very thin & multicore, high flexible	H
Very thin & multicore, naked conductor	Y
Sectors	S
Number of conductors	
Totally	...
without a safety conductor	X
with a safety conductor	G
Conductor section	
Color	...

Some Examples:

1. Insulated cable PVC 1,5 mm² single core conductor 450/750 V black colour (blk): **H07V-U 1,5 blk.**
2. Flexible cable (rubber) for 3-poles, ground cable 450/750 V.2,5 mm²: **H07RN-F3G 2,5**
3. PVC, flexible cable, bipolar, 075 mm²: **H05VV-F2X 0,75**

Cables: This library contains various types of cables along with their attributes (type, description, manufacturer, section, number of poles etc).



Load Types: This library contains the various types of electrical loads.

The user can create his/her own load types or modify the existing ones. The fields in the load types are the following:

- **Description:** Specify the load description. This description is printed in the calculations and the panel drawing.
- **Minimum Cable Cross Sectional Area:** Determine the minimum cable cross sectional area that will be allowed to be used for the specific load type, e.g. 2.5 mm² for sockets.

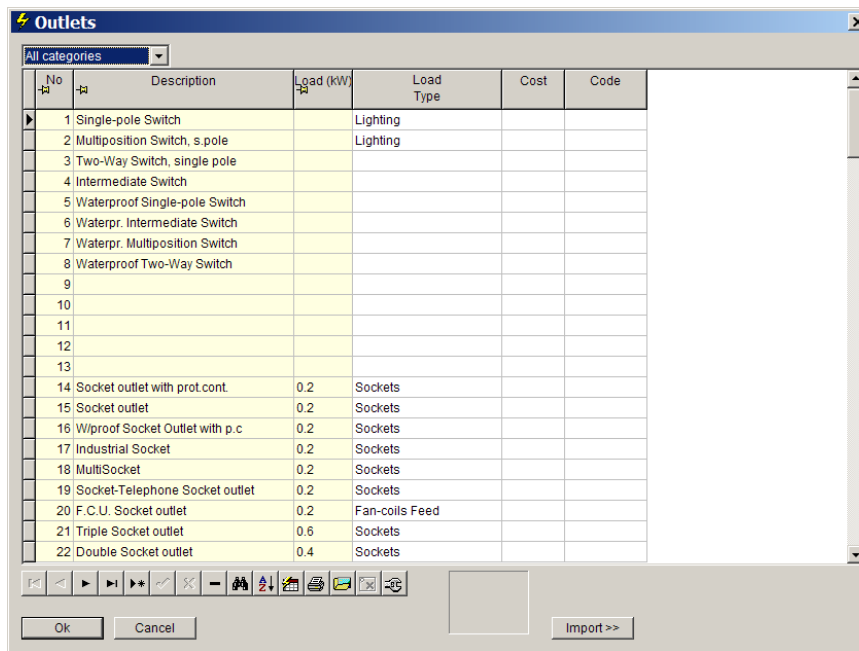
No	Description	Cross Sec	Cos phi	Minimum Fuse	Diagram of the Line	Motor	Cable Type	Type by NELOT	Phase Pole	Phase Pole
1	Lighting	1.5	1	10	Miniature Circ. Breaker (L-C)	No	NYA	H07V-U	Three Pole	Quintel Cable
2	Sockets	2.5	1	16	Miniature Circ. Breaker (L-C)	No	NYA	H07V-U	Three Pole	Quintel Cable
3	Heater	4.0	1	20	Switch 2Poles-Min.Circ.Breaker	No	NYA	H07V-U	Three Pole	Quintel Cable
4	One-Phase Cuisine	6.0	1	25	Switch 2Poles-Min.Circ.Breaker	No	NYA	H07V-U	Three Pole	Quintel Cable
5	Motor	2.5	0.8	16	Min.Circ.Br.-R.C.Sw.-Therm-Let	No	NYA	H07V-U	Three Pole	Four Pole
6	Panel	4.0		20	Supply panel	No	YYY	J1V-R	Three Pole	Quintel Cable
7	Tree-Phase Cuisine	6	1	25	Tumbler Sw.-Min.Circ.Br.-Led	No	NYA	H07V-U	Three Pole	Quintel Cable
8	Standby Line	1.5	1	10	Miniature Circ. Breaker (L-C)	No	NYA	H07V-U	Three Pole	Quintel Cable
9	Surge Diverter	2.5	1	10	Miniature Circ. Breaker (L-C)	No	NYA	H07V-U	Three Pole	Quintel Cable
10	TV Amplifier	2.5	1	10	Miniature Circ. Breaker (L-C)	No	NYA	H07V-U	Three Pole	Quintel Cable
11	Electric Radiator	2.5	1	16	Miniature Circ. Breaker (L-C)	No	NYA	H07V-U	Three Pole	Quintel Cable
12	Centrifugal Fan	2.5	0.85	10	Tumbler Swit.-Min.Circ.Breaker	No	NYA	H07V-U	Three Pole	Four Pole
13	Heat - pump	2.5	0.87	16	Min.Circ.Br.-R.C.Sw.-Therm-Let	Star-Delta Sw	NYA	H07V-U	Three Pole	Four Pole
14	Axial Fan	2.5	0.87	16	Tumbler Swit.-Min.Circ.Breaker	No	NYA	H07V-U	Three Pole	Four Pole
15	Water cooler	2.5	0.87	16	Miniature Circ. Breaker (L-C)	No	NYA	H07V-U	Three Pole	Quintel Cable
16	Speaker Amplifier	2.5	0.87	16	Miniature Circ. Breaker (L-C)	No	NYA	H07V-U	Three Pole	Quintel Cable
17	Security Lighting Feed	1.5	1	10	Miniature Circ. Breaker (L-C)	No	NYA	H07V-U	Three Pole	Quintel Cable
18	Garage Door	2.5	0.87	16	Miniature Circ. Breaker (L-C)	No	NYA	H07V-U	Three Pole	Four Pole
19	Dishwasher	2.5	0.88	16	Miniature Circ. Breaker (L-C)	No	NYA	H07V-U	Three Pole	Quintel Cable
20	Blender	2.5	0.88	16	Miniature Circ. Breaker (L-C)	No	NYA	H07V-U	Three Pole	Four Pole
21	Freezer	2.5	0.88	16	Miniature Circ. Breaker (L-C)	No	NYA	H07V-U	Three Pole	Quintel Cable
22	Cold Store	2.5	0.88	16	Miniature Circ. Breaker (L-C)	No	NYA	H07V-U	Three Pole	Quintel Cable
23						No	NYA	H07V-U	Three Pole	Quintel Cable
24	Kitchen Furnace	2.5	1	16	Miniature Circ. Breaker (L-C)	No	NYA	H07V-U	Three Pole	Quintel Cable

- **Cos phi:** Determine the cos phi of the load. For an ohmic load, value 1 is assigned. For a non-ohmic load, it is cos phi < 1.
- **Minimum Fuse:** Specify the minimum fuse value in A that will enable the application to secure the load.
- **Drawing Line:** Specify the line number, from "line types" of the program library, which will be used for drawing the start point of the supply line of the load. The term "Drawing Line" (drawing line for protection devices) refers to the combination of materials that must be placed in the Electrical Panel, in order to protect the supply cable of the load and to control the load.
- **Motor:** Specify if the load is a motor and its starting method (1:direct on line, 2 by Star/Delta switch). This greatly affects the way that the cables and the protection devices will be calculated. The way the protection devices and the cables are calculated for thermal loads is completely different than for motion loads.
- **Cable Type:** Specify the cable type according to HD384.
- **One-phase poles**
- **Three-Phase poles**

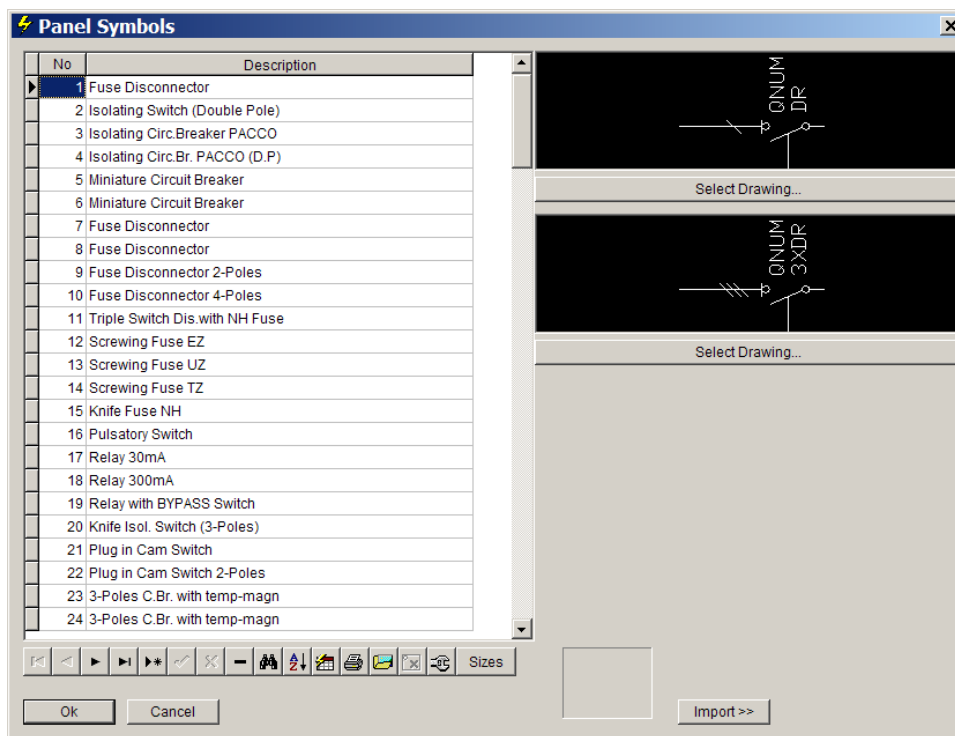
Transformers: It includes transformer types available in the market, along with a complete list of their features.

Fans: It includes fan types circulating in the market, along with a complete list of their features (supply, static pressure, electrical data, cost etc).

Outlets: It includes outlets as well as devices, along with their features (load size in kW, load type 1-160 and cost).

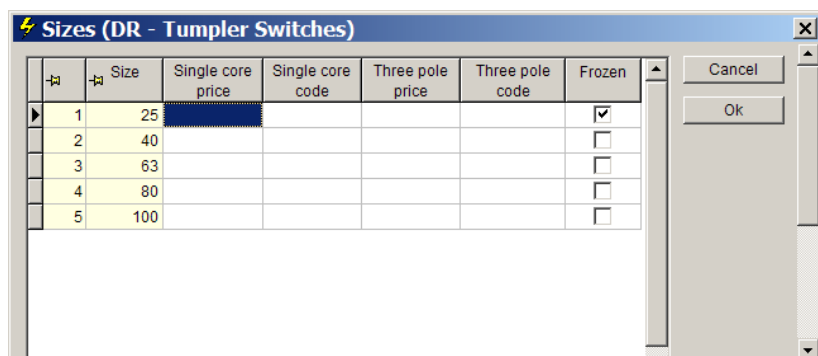


Panel Symbols: Here the user can edit either the symbol drawings or their attributes.



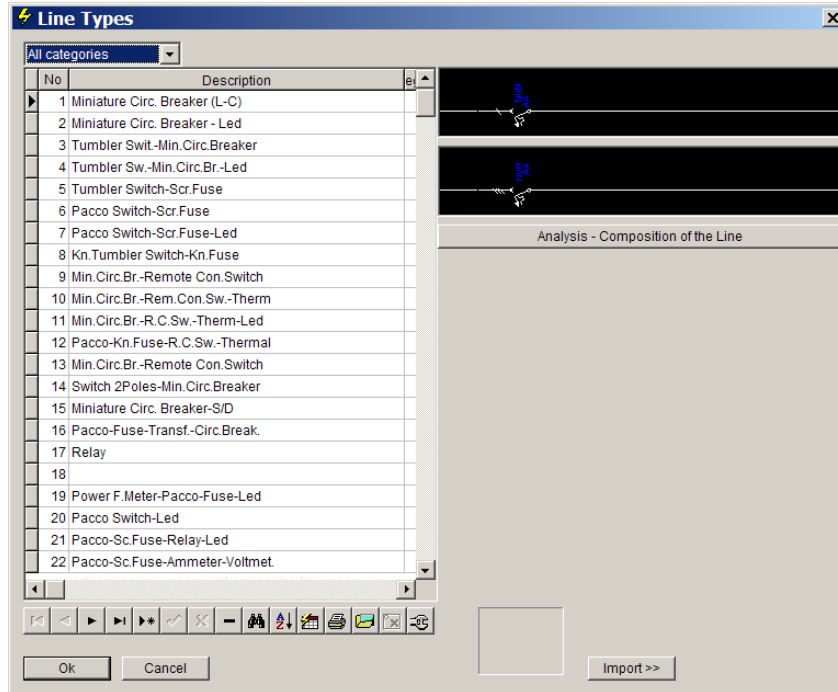
By selecting a certain symbol and by pressing the “Sizes” button, the user can edit the respected values.

The two fields related to the drawings are referred to the one-phase or three-phases drawings respectively. Using the “drawing selection” the

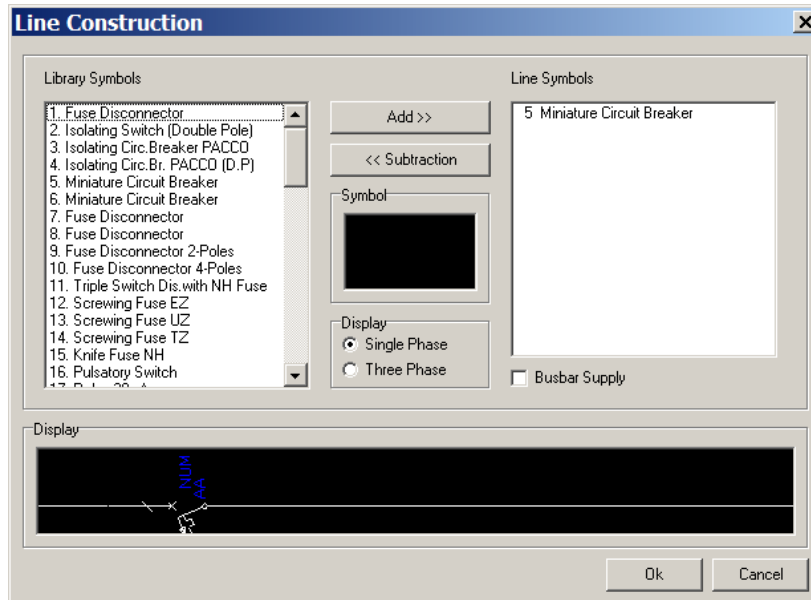


user can insert its own drawing and assign it with the symbol on the left. The symbols drawings in a DWG form are contained in the folder **DBJ** which is located at the libraries folder 4M\LIBS\DATAF of the program.

Line types: In this library, the user can create new lines, as well as to edit the existing ones.



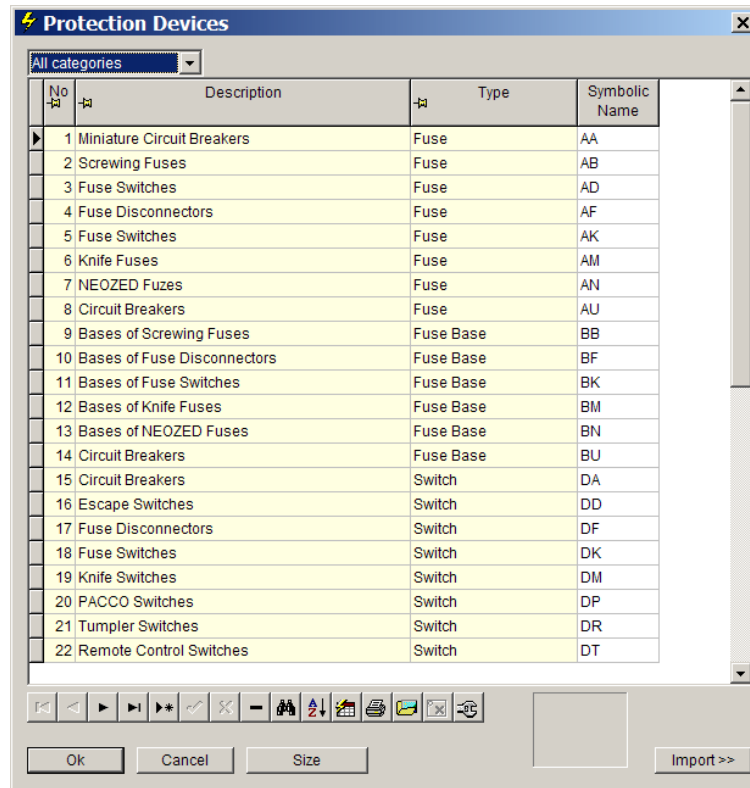
Composing or Analyzing a line: In order to create (compose) a new line, we have first to select an empty line (i.e. the 18th). Then, after filling the name of this line in the description field, the user has to press the respective button, so that the line creation dialog appears on screen.



Elements and protection devices can be easily added or removed by using the options “Add” or “Remove” respectively. The one-phase or three-phases drawing of any line is shown at the bottom of this dialog. The protection devices exist as DWG drawings within the folder **DBJ** which is located at the libraries folder 4M\LIBS\DATAF of the program.

Protection Devices

For each group, a new window containing the detailed sizes appears by pressing the button “Sizes”.



All the above libraries are open to the user for editing.

3.6 Help

This option includes a number of options that aim to support the user in learning each application-program following various ways. The most important of them is the **.Contents and Index** option according to the common windows standards.

