



# View & Plot (pro)

21. April 2023

## Version 2023

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**CAD**

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# 1 Introduction

## 1.1 About *CADdy++ View & Plot (pro)*

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*CADdy++ View & Plot (pro)* can be used with model files (\*.MOD) that have been created with *CADdy++ basic, economy or professional* to open them as read only files, to be looked through and printed.

Using *CADdy++ View & Plot pro* also the file formats CADdy PIC, DXF, DWG and PC-Draft can be imported, displayed and printed.

In addition in *CADdy++ View & Plot pro* the user can get information's about distance between points and objects from the drawing or the 3D model.

## 1.2 Installation

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Install CADdy++ View & Plot / View & Plot pro as described in the following chapter.

If the installation also includes *CADdy++ Model Management*, you must only start the program after you have also completed an ODBC link to the database named CADDYMV.MDB: it will be searched for the model records and drawing records.

## 1.3 *CADdy++ 64-Bit*

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*CADdy++* is, besides the 32-bit version, in all performance levels also available as a 64-bit version.

For the *CADdy++ 64-Bit-Version* a 64-bit operating system is necessary (Windows 64-bit), the *CADdy++ 32-Bit-Version* can be installed on 32-bit OS as well as on 64-bit OS.

When using a 32-Bit-Version of Windows, you **must** install the *CADdy++ 32-Bit-Version*.

If you are using Windows 64-bit and you want to work on big models, we recommend to us the 64-bit Version of our software. For 32-Bit-operating systems a maximum size of 3 Gigabyte RAM is addressable. The new *CADdy++ 64-Bit* technology allows extremely big models when using a 64-bit Windows platform on an appropriate hardware.

### **Please note:**

In *CADdy++* connections to different databases are required, realised via the ODBC interface from Windows. From CADdy++ - Version 2012 onwards, automatic database connections are made

to the installed SQLite databases when the program is started, if there is no permanent database connection defined.

**Please be aware**, that in the case of manually defined Access databases a 64-bit Office must be installed when working with the *CADdy++* 64-Bit Version.

DXF- or DWG- files can be embedded in the *CADdy++* 64-Bit version. The import and export of DXF / DWG files is not possible. For this tasks use the 32-bit version, which can be installed parallel in another directory.

In the 64-bit version of *CADdy++* Version 2011 a new, state-of-the-art library for pixel graphics has been implemented. The old commands to fix, polygon fill and vectorise images are not supported any more.

## Install program

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To install *CADdy++ View&Plot (pro)* start a computer with Windows Vista®, Windows 7®, Windows 8® or Windows 10® installed.

### Installation

**Please consider the remarks in the chapter before "*CADdy++* 64-Bit" for the installation of the *CADdy++* 64-Bit Version.**

1. Start the installation program „SETUP *CADdy++* <name of the performance level>.exe“ from the CD-ROM.
2. In the following dialogue window select the language for the installation program.
3. Click the **Next** button to start the installation or the **Cancel** button to abort.
4. After reading the license agreement and marking the button to accept them, click the **Next** button to proceed.
5. Enter your name, the company name and the serial number of your license in the dialogue fields.  
When installing *CADdy++ basic* also the key to register the full version can be entered here. Enter the data you have received with your delivery. If the fields for the serial number and the key are left free, a timely limited test version can be started after the installation is complete.  
Click the **Next** button.

6. Confirm the predefined destination folder for the program directory by clicking the **Next** button or define a new folder using the **Browse** button. The predefined destination folder is added as default to a selected directory. If the destination folder doesn't exist, please confirm to create the folder.
7. If the installation is to be done in the folder of a previous version, answer the question to proceed. When choosing **No** a new destination folder can be defined.
8. In the component selection all the modules from your license are activated by default.  
Confirm the selection by clicking the **Next** button.
9. Confirm the start folder for the program link with **Next**, or define a different one with **Browse**. The default start folder will also be used when defining a new folder by default.
10. Please select, if a program link should be added to the quick launch bar or to the desktop.
11. Finally you will get an overview about all of your settings. Click the **Install** button to start the installation or the **Back** button to change your settings.

## Silent Installation

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The *CADdy++* programs for Installation and Uninstall supports the unattended installation (silent installation).

Using an unattended installation the complete Setup or Uninstall of *CADdy++* is done without any user entries.

The time for the installation will reduce significantly because there is no waiting time for user entries and so several installations can be made consecutively. If *CADdy++* should be installed on several PC's, time can be reduced using this method, because the configuration can be made once and then be used on multiple PC's.

The installation can be preconfigured with command line parameters. The common syntax is as follows:

```
installationfile.exe <parameter1> <parameter2> <parameterN>
```

### General course of Installation

1. Start the installation program „SETUP *CADdy++* <name of performance level>.exe“ from a command line with the following additional parameters:

```
"SETUP CADdy++ XXX.exe" /SAVEINF=c:/temp/MyLog.log
```

All entries from the installation are stored in a LOG file.

2. To start the silent installation with the Log file use the parameters:

```
"SETUP CADdy++ XXX.exe" /SP- /VERYSILENT  
/SUPPRESSMSGBOXES /LOADINF=c:/temp/MyLog.log
```

Then the installation will run unattended with no user requests. The data stored in the LOG file can be edited manually before the installation.

**Please be aware:**

During an unattended installation with the parameters described above, the files in the standard directories changed by a user, as the standard templates or the 2D Standard Parts, are not overwritten. If the installation is started from a command line without the parameter /SUPPRESSMSGBOXES, the dialogues will appear, but then the installation will not run unattended any more.

**Parameters for the unattended Installation**

/SP-

Deactivates the question from the start of the installation, if the program should be installed.

/SILENT, /VERYSILENT

The parameter /SILENT deactivates the installation assistant but the progress bar is displayed. The parameter /VERYSILENT deactivates both dialogues.

All other dialogues are displayed, for example to show error messages during the installation.

If a restart is necessary after the installation and the parameter '/NORESTART' is not used, with the parameter /SILENT the question concerning the restart is displayed. Using the parameter /VERYSILENT a restart is done without request.

/SUPPRESSMSGBOXES

Suppresses the request dialogues during the installation and has only an effect in combination with '/SILENT' and '/VERYSILENT'.

Request dialogues with a selection are answered as follows:

- Yes, when asked if a new file should be kept.
- No, when overwriting existing files.
- Cancel in the request Cancel / Retry.



- Yes (= Next) in the situations  
not enough space available / folder already exists / folder does not exist already / No Uninstall warning / Question to Cancel the Setup / Confirm Uninstall.
- Yes (= Restart) in the situations  
Done - Restart / Uninstall needs Restart.

`/NOCANCEL`

Avoids the cancellation during installation. Useful in combination with the parameters `'/SILENT'` or `'/VERYSILENT'`.

`/NORESTART`

Prevents a Restart, even if necessary.

`/LANG=language`

Defines the language used during installation. For `language` please insert the internal name in the form `[language]` (GER,ENG,ITA,SPA).

If a valid `/LANG` parameter is used, the language dialogue is suppressed.

`/LOADINF="filename"`

Loads the settings defined in the specified file (complete with path) after checking the parameters from the command line. The file can be created using the parameter `/SAVEINF =`, as described below.

If the filename contains a blank, it must be entered in the form `"filename"`.

`/SAVEINF="filename"`

Saves the installation settings in the specified file (complete with path). If the filename contains a blank, it must be entered in the form `"filename"`.

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## Create link with Model Management database

---

If you have also selected *CADdy++ Model Management* as part of the installation, you must set up a link to the database in which the records are saved.

The system must be informed about both their location and the database driver that is in use.

The link between the data source, the driver and *CADdy++ Model Management* is completed by the ODBC interface.

Complete this step in the following way:

1. First, find the drive/the folder containing the database **CADDYMV.MBD**.
2. Then activate the **Start Menu** and execute the commands **Settings, System Control**.
3. Activate the **ODBC** Administrator. This can be found in the folder **Administration** from Windows 2000 onwards.
4. Specify a new data source by clicking **Add**.
5. Select the driver in this dialog box **Microsoft Access Driver (\*.mbd)**.
6. Click the **Complete** command button to allocate a database source to the driver you have just selected.
7. Type the entry **CADdy MV** in the text box **data source name** (ensuring that there is a blank between the 'y' at the end of CADdy and Model Management). This is the name used internally by Material Management for the database, so you must make sure you type the name correctly.
8. Click **Select** to choose the folder containing the database file named **CADdyMV.MDB**.
9. Mark this file and exit all the dialog boxes by clicking **OK**.

The link we have described above must be created on each computer separately so that its user will be able to access the Model Management database individually.

## Support for SQLite Database

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The Office 2010 64-Bit-Version is not compatible with other 32-Bit Office programs. This results in problems when using MS-Access databases under Windows 64-Bit. For this reason the **CADdy++** internal databases will be delivered in the SQLite format from now on. The existing MS-Access databases are delivered in parallel, but they will not be used as the standard.

In **CADdy++** connections to different databases are necessary, which are realised via the ODBC interface from Microsoft.

Database connections are always realised in ODBC using the so called ODBC data source. This data sources can be predefined in the Windows System Control or they are automatically created when **CADdy++** is started and deleted when the program is left. The automatic creation of data sources was limited to Access databases up to now.

**Please be aware**, that from *CADdy++* - Version 2012 upwards the automatic connection to databases is done to the SQLite databases, if no permanent database connection was created.

To simplify the management for the access to databases, from *CADdy++* 2012 onwards the creation of data sources is controlled by the file **ODBCDataSources.ini**. This file can be found in the directory **LW:\CADdy...\Program\Bin** after the installation

**Please be aware**, that this file is a program file and will be overwritten with update installations. To save user settings in this file, copy the file **ODBCDataSources.ini** into the *CADdy++* USER directory. When starting the program the INI file will preferably be found in the USER-, and then in the ...Program\Bin directory.


### **Usage of existing databases**

If you have used a predefined database connection in previous *CADdy++* versions, for example to the file Bauteile.mdb, this connection will also be used after the installation of *CADdy++* Version 2012 as default.

### **Change of existing Access databases to SQLite**

To change existing Access databases to SQLite on the *CADdy++* Installation CD from Version 2012 a Converter<sup>®</sup> can be found, which can be installed on any PC.

Follow these steps:

1. Installation of the Converter: Start the installation program „**CADdy++\_MDB2SQLite.exe**“ from the *CADdy++* Installation CD in the directory **MDB2SQLite** and follow the instructions.
2. In the Windows Explorer open the install directory of the converter and start the program **MDB2SQLiteConverter.exe**.
3. In the upper line enter the drive, directory and the name of the MDB file to be converted.  
This can be done by entering the name or a file selection can be made by clicking the  icon.
4. In the lower line enter the directory and the name of the DB3 file.
5. Click the **Convert** button.
6. In the case that an automatic database connection should be created when starting *CADdy++*, please read the following chapter.

Otherwise specify a predefined database connection via an ODBC data source to the DB3 file created before.

**Please note:**

To delete the directories and files created when installing the converter, please use the Windows Explorer.

## **Settings for the creation of ODBC data sources**

---

To define your own settings for the creation of ODBC data sources, copy the file ***ODBCDataSources.ini*** from the directory drive:\CADdy...\Program\Bin into the CADdy++ USER directory.

### **General structure of the INI file**

[ODBCDataSources]

General parameter, valid for all connections.

[FileTypeXYZ]

A section, starting with the name 'FileType' containing parameters for the file type XYZ

[ODBC Datenquellen]

Parameter for a concrete ODBC data source, where the name (DSN=**DataSourceName**) is the same then the name of the section. CADdy adds a ,\_Intern' to this name, to differentiate it from user defined database connections.

### **Section [ODBCDataSources] - general parameter**

Name	Value	Meaning
AutoSearch		A list of file extensions, divided by semicolon, searched consecutively.
ConnectOnStartup	1	All data sources listed in the INI file are created when the program is started and deleted again when the program is left.
	0	The data sources are created upon request and also deleted afterwards.
IgnoreExtension	0	If a database file is given in the program with file extension (e.g. in Standard parts FSC file), first this file is searched and after that the file extensions defined in AutoSearch.

- 1 The given file extension will be ignored and a search will be made only with the file extensions defined in AutoSearch.

### **Section [FileType...] - the list of file types**

Every section, starting with the name 'FileType', describes a file type with the according file extension and the parameters to create an ODBC data source.

Name	Value	Meaning
Extension	string	File extension including leading point.
DefaultDescription	string	Description of the connection (displayed in the ODBC Manager).
DefaultConnectionString	string	By semicolon separated list of ODBC connection parameters for this file type. This string may contain variables in the form \${...}, see below.
Driver32	string	Name of the 32bit ODBC driver (for 32-bit programs)
Driver64	string	Name of the 64bit ODBC driver (for 64-bit programs)

### **Section [...] - ODBC Data sources**

Every further section defines the parameters for a concrete ODBC data source. The values ,ConnectOnStartup', ,IgnoreExtension' and ,AutoSearch' are optional and can be defined here to overwrite the global settings from the section [ODBCDataSources] for this specific ODBC data source. Also ,DBFileName', ,Description' and ,Driver32' / ,Driver64' in this section are optional.

Name	Value	Meaning
DBFileName	string	Name of the database file (with or without file extension). This parameter is optional – if he is set, he overwrites the name given from the program. If no file is linked to this data source, the parameter must be set empty.
Description	string	Description of the data source (displayed in the ODBC Manager)
ConnectionString	string	By semicolon separated list of ODBC connection parameters.

		This string may contain variables in the form \${...}, see below.
ConnectOnStartup	0 / 1	Overwrites the value for this ODBC data source
IgnoreExtension	0 / 1	Overwrites the value for this ODBC data source; file extensions searched.
AutoSearch		List of file extensions divided by semicolon
Driver32	string	Name of the 32bit ODBC driver (for 32-bit programs)
Driver64	string	Name of the 64bit ODBC driver (for 64-bit programs)

### **Predefined variables for ConnectionStrings**

In the ConnectionString variables (,DefaultConnectionString' and ,ConnectionString') the following variables can be used:

Name	Value
\${DefaultConnectionString}	The value of the ,DefaultConnectionString' from the File type section
\${DefaultDescription}	The value of the ,DefaultDescription' from the File type section
\${DSN}	Name of the data source (DataSourceName)
\${DBFileName}	The complete file name including extension of the database file
\${Description}	The value of the ,Description' from the data source section
\${DIR_HOME}	User directory
\${DIR_EXE}	Program directory

### **Predefined variables for DBFileName**

In the file name ,DBFileName' the variables \${DIR\_HOME} and \${DIR\_EXE} can be used.


## **2 Screen**

### **2.1 Main window**

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The main *CADdy++* window contains the following components:


## Header

 MODEL.MOD - CADdy++ Mechanical Design - [2D Drawing Scale 1.0 : 1.0 Current Drawing Dimension 420.0 mm / 297.0 mm]

Whenever the program is started, this line will contain the name of the model MODELL.MOD, the description of the program and the name "3D Model" or the description of the current 2D drawing with its dimensions.

When you open a model, this information will be replaced by the details on the model that has been opened.

## Menu bar

 File Process Information View Windows ?

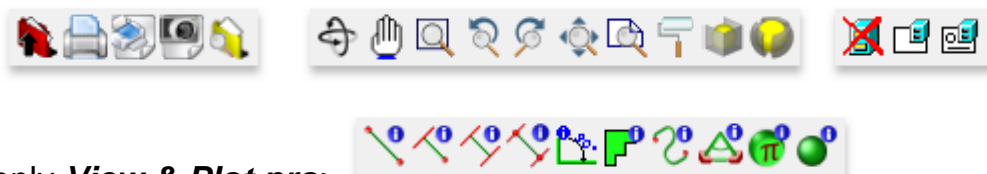
The menu bar contains headings of the pull-down menus, i.e. menus which are "pulled down" by clicking on the heading, therefore opening the menus. You can use the lines in this menu to call up the commands of the program.

This is done by moving the cursor to a menu heading. Confirm by pressing the left mouse button (= clicking) to start the command.

The commands in the menu bar contain a number of underlined letters e.g. View, Drawings.

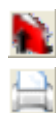
In such a case you can use the key combination **Alt A** to open the **View** menu, the key combination **Alt A Z** to call up the command **View, Drawings** directly.

## Toolbar



only **View & Plot pro:**

















The symbol bar contains, amongst other things, commands for opening and printing model data and for changing the display. It is shown by using the menu command **View, Display symbol bar** below the menu bar. You will find these commands:













Open model (with read only)



Print current drawing/3D model

	Print 3D Solid
	Create Pixel File from 3D View
	Model Explorer
	Rotate dynamically using middle mouse button, also zoom dynamically, set viewpoint
	Vary View
	Zoom window
	Display previous view
	Display next view
	Display all 3D objects
	Display entire WP (original) in top view
	Redraw
	Start Hide Action (only 3D)
	Start Hide Others Action (only 3D)
	EDM system (only if Model Management is installed)
	Administrative Settings
	Display record

Only **View & Plot pro**:

	Distance point to point
	Distance point to line
	Distance line to line
	Determine Distance
	Angle
	2D face calculation
	Continuous line length
	Information about Distance and Angle of Solid Faces
	Information about solid calculation ( Volume, centre of gravity, weight)
	Calculate a surface area

Start any command by clicking its symbol.



## Prompt line

Zoom Window

Define first corner of window!

In this line at the bottom of the main window, messages are displayed which show the command selected, e.g. zooming a window.

Next to this, is a message stating which action now needs to be carried out for the respective command, e.g. define the first corner point of a window.

## Working with the Status dialog box

The Status dialog box is for displaying numbers such as point coordinates, lengths, sizes of angles.

### Activate Status dialog box and select line

For the input of numeric values or formulae, the cursor must be in the Status dialog box. There are two options of doing so:

- **Using the mouse**

Using the mouse, move the cursor into the required text box.

By simply clicking on the text box, it then becomes the input text box.


By double-clicking, the text box is activated. At the same time, the existing entry is highlighted and can be written over directly.



- **Using the keyboard**

Press the spacebar. This results in the change from the drawing area to the last active text box in the Status dialog box and vice versa.

The required text box within the Status dialog box can be reached using the cursor control keys  $\uparrow$ ,  $\downarrow$  or using **Tab**, **Ctrl-Tab**.

The existing entry is highlighted and can be written over directly.

The Status dialog box can be docked on to the program window to the left, right, top and bottom. In order to do so, move the dialog box over the border of the program window, until it visually changes its form and then let it fall. The docked on Status dialog box can be released again by clicking on the strip  and keeping the mouse button pressed to loosen the dialog box from the program window.

Minimise the Status dialog box using the  icon, close it using the  icon. Once you have minimised the Status dialog box in the locked on state, you can open it again by double-clicking next to the



strip in the program window frame.

If you wish to move the Status dialog box on the border of the program window without it being docked onto it, keep the Ctrl key pressed before releasing the Status dialog box on the required position.

The size of the Status dialog box can be modified by moving the cursor to the edge of the Status dialog box until a black double arrow appears. At this moment, press the mouse key and drag the Status dialog box to the required size.

The vertical division between the names in the Status dialog box and the entry text boxes can be modified by moving the middle vertical line.

Setting options for the Status dialog box, e.g. the number of decimal places for the display, as well as the background colour for text boxes that must be edited or even if you wish to define a transparency degree for the Status dialog box (from Windows 2000), can be found in the menu bar under **File, Options** on the **User Interface** option card.

---

## 2.2 3D view window

A model completed in *CADdy++* may contain several 2D drawings and a 3D model. The 3D model will be displayed inside a 3D view window that is shown inside the main window.

---

## 2.3 2D drawing window

A model completed in *CADdy++* may contain several 2D drawings and a 3D model. Any 2D drawing will be displayed inside its own 2D window that is shown inside the main window.

---

## 2.4 EDM system toolbar

If you have installed *CADdy++ View & Plot* and *CADdy++ Model Management*, you can use following commands:



Deactivate ODMA



Activate ODMA



Administrative settings




Display record


If *CADdy++ Model Management* is not installed, you cannot use the symbols of the toolbox to execute any commands.

## 2.5 The Model Explorer

Open the Model Explorer using the menu **View, Model Explorer**.

The Model Explorer offers three option cards:

 **Model Explorer** – Shows all objects in the 2D and 3D area in a structured tree and the content of selected areas in a detailed view. It allows the management and structuring of drawings and the 3D model.

 **Layer Explorer** – Manages the display properties of 2D objects. Further informations can be found in the chapter **2D commands, Control display of 2D objects**.

Change to the option card **Model**:




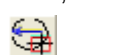
The entire structure of the opened model is displayed in the **Tree structure** on the left side of the Model Explorer. In the **List view** on the right side, the objects and/or groups of the selected folder are displayed in the tree structure. In the bottom half of the Model Explorer, you will find the **News area** in which error messages or notes concerning design problems can be displayed (example: after activating a snap point using the keyboard without previously having activated a command). You can activate or deactivate the display of news using the **Settings, Options** menu on the **User Interface** option card under **use model explorer to display message**.

The individual areas of the Model Explorer can be adapted individually to the required size by clicking on the grey strip that separates the areas and keep the mouse button pressed to drag the area to the required size.

## Overview of the icons in the Model Explorer

The Model Explorer displays different icons related the the current option card. Please find summary of the icons with a short explanation of the commands:

### Option card: **Model Explorer**

	Scan groups on / off
	New group will (not) be current group
	(Do not) activate selected component
	Localise object in model tree



Highlight active part / component



Only delete messages



Delete all texts

### Option card: **Layer Explorer**



Scan layer on / off



New layer (un)marked



Localise object in model tree



Current layer for Geometry



Current layer for Text



Current layer for Construction\_Aids



Current layer for Hatches



Current layer for Dimensions



Current layer for Centre\_Lines



Current layer for Hidden\_Lines



Current layer for Hidden\_Edges



Current layer for Winding\_Lines



Current layer for Tangential Edges (for example in model views)



List view / Tree view active







Only delete messages



Delete all texts

## Working with the Model Explorer

The **Model Explorer** can be *docked* onto the program window. Once you have opened the dialog box, move the **Model Explorer** to the border of the program window until it visibly modifies its shape and then let it fall. The docked on **Model Explorer** can be released again by clicking on the strip  and keeping the mouse button pressed, drag the dialog box into the centre of the program window. In docked on status, minimise the **Model Explorer** using the  icon and close it using the  icon. Once, in the docked on state, you have minimised the **Model Explorer**, you can open it again by double-clicking next to the strip  in the program window frame or by dragging it to the strip by keeping the mouse button pressed. If you wish to move the **Model Explorer** to the border of the program

window without it being docked on, keep the **Ctrl key** pressed until you have positioned it on the required location.




### **Scan groups on /**

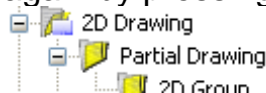


### **Scan groups off**

If scanning is switched on, you can scan through the entire model structure with a mouse click or by using the arrow keys on the keyboard.

Click on a structure branch in the left half of the Model Explorer and then confirm the arrow key on the keyboard.


Closed node points  **2D Drawing** can be opened whilst scanning by pressing the right arrow key on the keyboard, in order to allow you to scan in the subordinate structures. Opened nodes can be closed again by pressing the left arrow key.




### **Determine new group as current group /**



### **Do not determine new group as current group**

If this command is active , a new group is always generated one level under the active group or partial drawing and can be directly renamed.

If the command is not active , the previously active group or partial drawing remains active. Groups that are later generated in the same procedure are generated on the same hierarchical level as the first new group.



### **Localise object in model tree**

Once you have started the command using the displayed icon, identify an object in the active graphic window. The structure tree (left) will be opened up to the folder in which this object is found. The selected object will be suppressed in grey in the list view (right) of the Model Explorer.



### **Activate tagged components /**



### **Do not activate tagged components**

If this command is active, each folder you have selected by mouse click in the tree structure of the Model Explorer, is defined as the active working area.

If you identify the *3D Model* folder or the folder of a *2D Drawing*, the previously active workplane or the previously active partial drawing becomes the active working area.

If, however you identify a workplane or a partial drawing/group directly, this folder becomes the active working area.



### **Erase news**

If you have activated the display of news using the **Settings, Options** menu on the **Model Explorer** option card under **Output error messages/notes using the Model Explorer**, this command can be used to erase the content of the news area in the Model Explorer.










Use this command to delete all texts. Single texts can also be deleted using standard text commands.

### **Display**

The colour display of the respectively active areas can be determined using the **Settings, Options** menu on the **Model Explorer** option card.

In the view of the Model Explorer the folders can be differentiated between via colours:

-  (yellow) = 2D group
-  (yellow) = 2D-Standard Part with BOM entry
-  (green) = 2D component
-  (purple) = 2D component group
-  (yellow) = 3D group
-  (green) = 3D component
-  (purple) = 3D component group

## **Lowlight non active Groups**

---

Using this command, objects in the current group in 2D are marked in colour and in 3D are highlighted by different transparencies.

To highlight the geometry in active groups, open the **Model Explorer** using the menu command **View**. In the Model Explorer the following buttons can be found:



If the button is displayed as shown, the command is **deactivated**. All objects in active and non active groups are displayed according to the standard settings. When the button is selected, the button display is changed as described below:



If the button is displayed as shown, the command is **activated**. In this case only the objects in the active group are displayed according to the standard settings. All other objects are lowlighted. When changing the active group, e.g. in the Model Explorer, the

display of the objects in active and non active groups will change accordingly.

## Context menu in Model Explorer

---

Open the **Model Explorer** using the menu **View** and switch to the option card **Model Explorer**. The following describes the individual options of the context menu. The context menu can be opened using the right mouse button in the Model Explorer. Depending on whether you are currently in the tree structure or list view, different options are offered to you in the context menu.

### Current group (F12)

---

The area, in which solids are generated (3D solids), as well as workplanes and groups in the 3D area or partial drawings and groups in 2D drawings, can be activated as the current target context for new objects, at the same time, as the current work area. The following objects are actually created in this active area:

**3D Solids**

**Workplanes**

**2D / 3D Groups**

**2D / 3D Parts and components**

**2D Partial drawings**

In the tree structure in the left of the Model Explorer or in the list view in the right of the model Explorer, identify the *3D Solid* node point, a workplane, a partial drawing or a group and select the **Current Group** option from the context menu. Then press the **F12** key to define the respective area as the active context.

### Current 2D area (Shift+F12)

---

Workplanes (WP) or partial drawings (PD) can be activated as the current work area using this option. Opposite of the **Current Group (F12)** option, the group previously activated as the target context within this workplane/partial drawing, remains active. Only the work area is changed.

### Current 2D/3D context (Ctrl+F12)

---

Alternatively to the key combination **Ctrl+TAB** or the **Window** menu, you can use this command to swap between opened 2D drawings and the 3D window using the Model Explorer. Within the structure of the

selected drawing or the 3D window, the previous work area (3D, workplane, partial drawing or group) remains active.

**Please note:**

A new view of closed windows can be gained by opening the context menu on the closed drawing / the closed 3D window and selecting the **New, New View** option.

## **Selection**

---

Using the **Selection** option, another menu is opened with which you can select numerous selection possibilities. You will need to differentiate between the objects (lines, circles, text, etc.) in a group and the groups (including the content of the group).

### **Select (Ctrl+S) /Deselect (Ctrl+D) geometry objects**

This command can be used to select or deselect the objects in the selected structure folder (3D solid, workplane, partial drawing, group, part or component) in the *tree structure* or the objects in Groups, Parts or Components selected in the *list view*.

Subgroups or objects contained within them, are not taken into consideration with this command.

If you keep the Ctrl key pressed when identifying folders in the tree structure by mouse click, the *Select/Deselect geometry object* is also carried out.

### **Select (Shift+S)/Deselect (Shift+D) all geometry objects**

This command selects/deselects all objects within the selected folder (3D solid, workplane, partial drawing, Groups, Parts or Components) including all objects in the subgroups.

If you keep the Shift key pressed when identifying folders in the tree structure by mouse click, the **Select/Deselect geometry object** is also carried out.

### **Select object (Alt+S) / Deselect object (Alt+D)**

In the tree structure, select or deselect a group and its subgroups in the objects selected in the list view and /or groups and their subgroups.

Proceed as follows:

#### **In the structure tree:**

...the selection is equivalent to the group selection.

- Keep the **Alt key** pressed and click on the groups(s) that you wish to select.

or



- Tag the group you wish to select/deselect. Then use the key combination **Alt+S** to select or **Alt+D** to deselect.

or

- Tag the group you wish to select/deselect. Then select the **Selection / Select Object** or **Deselect Object** command from the context menu.

Repeat the procedure for other groups you wish to select.

#### **In the list view:**

In the list view the selection contains groups if groups are selected and objects if objects are selected. The difference is displayed using different colours.

- Tag the objects and groups you wish to select/deselect. Then use the key combination **Alt+S** to select or **Alt+D** to deselect.

or

- Tag the required objects and groups you wish to select/deselect. Then select the **Selection / Select Object** or **Deselect Object** command from the context menu.

#### **Select all contents (Ctrl+Alt+S) / Deselect all contents (Ctrl+Alt+D)**

This option can be used to select the objects and/or groups contained in the list view of the *tree structure*. Objects tagged in the *list view* are selected as objects, in tagged groups their objects and subgroups are selected.

#### **In the structure tree:**

- Keep the **Ctrl** and **Alt** keys pressed and click on the required structure folder whose content is to be selected. All folders below the 3D space and the drawings in the structure can be selected.

or

- Tag the structure folder *3D Solid*, workplane(s), partial drawing(s) or group(s) whose content you wish to select/deselect. Then choose the key combination **Ctrl+Alt+S** to select or the key combination **Ctrl+Alt+D** to deselect.

or

- Tag the structure folder *3D Solid*, workplane(s), partial drawing(s) or group(s) whose content you wish to select/deselect. Then choose the **Selection / Select All**

**Content** command or **Deselect All Content** command from the context menu.

**In the list view:**

- Tag the groups whose content you wish to select/deselect. Then choose the key combination **Ctrl+Alt+S** to select or the key combination **Ctrl+Alt+D** to deselect.

or

- Tag the groups whose content you wish to select/deselect. Then choose the **Selection / Select All Content** command or **Deselect All Content** command from the context menu.

**Paste selected objects (Shift+Alt+S)**

This command can be used to move the objects previously selected from the structure folder in the tree structure or the groups selected in the list view.

The command correspond to the command from the **Process** menu, **Accept Selection In Group**

**Please note:**

Objects and groups can be moved between the structured using Drag&Drop:

- Firstly tag the objects or groups you wish to move in the structure.
- Click again on the objects and, whilst keeping the mouse button pressed, drag them to the required structure folder in which you wish to insert the objects/folder.

**Insert copies of selected objects (Shift+Alt+C)**

This command can be used to copy objects previously selected in the drawing / in the 3D window into the selected structure folder of the tree structure or the selected group in the list view. The absolute position of the copies is taken over from the originals.

Please note:

Objects and groups can also be copied between the structures using Drag&Drop:

- Firstly tag the objects or groups that are to be copied into the structure.
- Click on the object again und keep the mouse button and the Ctrl key pressed to drag it to the required structure folder into which the object/folder is to be inserted. The copy procedure is recognised by the plus sign displayed on the mouse pointer.

## Find and select

The options in the context menu **Selection**, **Find and select** allow you to search for groups in the structure tree and to select the groups or their contents at the same time.

First select an option for the type of selection. Then the dialogue for the specification of the search opens and you can start the search. Decide whether you want to select the result of the search individually or make all selections in one step.

You have the following options for search and selection:

### **Select / deselect geometry objects**

With this option, only the geometry objects contained in the searched group can be selected / deselected.

### **Select / deselect all geometry objects**

The geometry objects contained in the searched group and in all subgroups are selected / deselected.

### **Select / deselect group and its subgroups**

The searched group and its subgroups can be selected / deselected.

### **Select / deselect content of the group**

The geometry objects contained in the searched group and the contained groups (including subgroups) can be selected.

After selecting an option, a dialogue opens in which you determine the search criteria. Enter all or part of the search word in the dialogue. Start the search via the **Find next** button. After each further click on this button, the next search result is displayed. Click on **Select** to select the individual search result or on **Select All** if you want to select all objects with the same search result in the structure tree.

You can refine the search options by activating the options for whole words only and/or upper/lower case. The procedure corresponds to the same search without selection and is described in the chapter *Context menu in Model Explorer*> [Find](#).

---

## Visibility/Pickability


### - **Visible / Invisible (O)**

This command can be used to suppress the objects of a group and their subgroups. Groups that cannot be seen are automatically

made inactive and are identified by the  icon.

**Please note:**

If you wish to make a previously suppressed group visible and active again, the easiest way is to use the option **Active / Inactive** so that the group is active and visible at the same time.



- **Active / Inactive (P)**  
Objects in workplanes, partial drawings or in groups that you have deactivated with this command, remain visible but cannot be selected or processed. Inactive workplanes, partial drawings and groups are tagged with the  icon.
- **Toggle visibility incl. child groups (Ctrl+Alt +O)**  
This command can be used to suppress or hide the objects of workplanes, partial drawings and groups and their subgroups. In this case, in opposition to the **visible / hidden** option, also the groups and subgroups deeper in the structure also receive the status visible / hidden.
- **Toggle pickability incl. child groups (Ctrl+Alt+P)**  
Objects in workplanes, partial drawings and in groups which you deactivate using this option, remain visible but cannot be worked with or picked. In this case, in opposition to the **active / inactive** option, all subgroups also receive the status active / inactive.
- **Branch visible (Ctrl+O)**  
All workplanes, partial drawings and groups and subgroups below the node get the status visible.
- **Branch invisible (Ctrl+Shift+O)**  
Workplanes, partial drawings and all groups and subgroups below the node get the status invisible. Objects in workplanes, partial drawings and groups are hidden.
- **Branch active (Ctrl+P)**  
Workplanes, partial drawings and all groups and subgroups below the node get the status active. All objects in groups can be accessed. Previously hidden objects are visible again.
- **Branch inactive (Ctrl+Shift+P)**  
Workplanes, partial drawings and all groups and subgroups below the node get the status inactive.

## **Use Workplanes as Cutting Planes**

---

Existing workplanes can be switched to cutting planes now. If the cutting plane interferes a solid the inside of the solid is viewed.

In addition three standard cutting planes can be called each of them in the centre of all solids with different directions. A maximum of 6 cutting planes can be active at the same time, additional cutting planes will be displayed as non active cutting planes.

Cutting planes can be modified and transformed with the same commands as the workplanes, e.g. move along Z direction to “walk” through the model with the command  **Move WP along Z** or switch the cutting direction with the command  **Invert WP**.

A standard cutting plane can be activated as follows:

1. Open the **Model Explorer** using the menu command **View** and select the tab **Model Explorer** or **Assembly Explorer**. (from **professional**).
2. Start the context menu and select from the option **Cutting Plane** the standard cutting plane.

To switch an existing workplane to a cutting plane do the following:

1. Open the **Model Explorer** using the menu command **View** and select the tab **Model Explorer**.
2. Place the cursor on the workplane and open the context menu. Select the option **Cutting Plane, Add and Activate.(Shift+Q)**.

To deactivate a cutting plane, so that the solid is not displayed in a sectioned manner any more, open the context menu on the cutting plane and select the option **Cutting Plane, Activate/Deactivate**.

To switch from a cutting plane to a workplane the command **Cutting Plane, Remove** is used.

## Properties (Shift+F2)

---

### Only in the context menu in the list view (right)

Double click on the required object or tag it in the list view and then from the context menu, select the **Properties** option. Depending on the selected object, the **2D Object editor** dialog box is opened or a respective other dialog box (text editor, 2D standard part, etc.). This command corresponds to the **Information / Edit Object** command from the toolbar.



## New

---

Dependent on the selected folder or object, use this command to define a new **group** (2D/3D area), a new **partial drawing** (only 2D area) or a new **2D drawing** (only 2D area).

### **Set up new group (Ctrl+N)**

If in the tree structure in the 3D area you tag a workplane, or in a 2D drawing a partial drawing or a group, a new group will be set up under this folder once the key combination **Ctrl+N** has been selected or **New / Group** is selected from the context menu.

Depending on the setting you have made in the Model Explorer using the icon  **Determine new group as current group** /  **Do not determine new group as current group**, further new groups are structured or set up on the same structure level.

### **New partial drawing (Ctrl+N)**

In the structure tree, tag a 2D drawing and select the key combination **Ctrl+N** or select **New / Partial Drawing** from the context menu.

In order to be able to edit the settings of a partial drawing, tag a partial drawing in the tree structure and select the **New / Drawing Settings** option from the context menu.

### **2D drawing (Ctrl+N)**

In the tree structure, tag the topmost node point (model path) and select the key combination **Ctrl+N** or the option **New / 2D Drawing** from the context menu.

The Drawing Manager can be opened using the Model Explorer by tagging a 2D drawing in the tree structure and selecting the **New / Drawing Manager** option from the context menu.

### **New view(Shift+Ctrl+N)**

If you wish to open a new 2D or 3D view window, for a new 2D view window, tag a drawing in the tree structure and then select the **New, New View** option in the context menu.

The same command can be used to set up a new 3D view window if the context menu is opened on the *3D space* structure folder.

### **Edit attributes (Shift+A)**

---

If a group (such as parts or components) has been allocated attributes, this command can be used to display or edit the attributes. Open the context menu by pressing the right mouse button whilst on the respective group in the tree structure or list view and select the command **Edit Attributes (Shift+A)**.

In order to be able to display or edit the attributes of an object (such as the a standard sheet or another symbol), firstly tag the object in the list view and then select the required command from the context menu or using the keyboard commands

## Rename (F2)

---

Tag the folder you wish to rename in the tree structure or the list view. Press the **F2** key or select the **Rename** command from the context menu. Enter the new name and confirm the entry with the Enter key. Objects such as lines, circles and text cannot be renamed.

### Please note:

By changing the name of a part or component also the part/component ident number is changed and a new part/component is created!

## Focus objects (Shift+F3)

---

Only in the context menu in the list view (right)

By selecting this command in the context menu, objects tagged in the list view or the content of tagged folders will be highlighted in colour and focused in the respective drawing or the 3D model according to the menu **Settings, Options** on the **User Interface, Model Explorer** option card.

This command has been extended:

### Ctrl key

When pressing the **Ctrl key** during the command the current settings in **Options** are inverted.

### Shift key – only in 3D objects

When pressing the **Shift key** during the command, all other 3D objects are faded out, to focus on inside solids.



To fade in the solids again select the icon **Start Hide Action** at the upper 3D window.

Both keys can also be used in combination.

### Please note:

The extended keys are only valid when starting the command from the context menu. They are not valid when starting the command using the **Shift+F3** key combination.

## Find (F3)

---

Enter the word to be found entirely or part of it in the dialog box that appears by selecting the **F3** key or using the **Find** command in the context menu. Start the search using the **Search** command button. After every further click on this command button, the next search results are displayed.

The search criteria can be given in more detail by activating the options *whole words only* and/or *match case*.

The search is limited to the word start. Wildcards are not permitted.

---

## Replace (F4)

---

In order to replace descriptions of folders with new descriptions, select the **F4** key or select the **Replace** command from the context menu. In the **Search for:** text box, enter the text to be found. In the **Replace by** text box, enter the new text. Click on **Search**, **Replace** or **Replace All**. As opposed to folders, the object names of lines, circles, text, etc, cannot be replaced by other descriptions.

The search is limited to the word start. Wildcards are not permitted.

---

## Refresh (F5)

---

To update the view in the Model Explorer, press the **F5** key or select the **Refresh** option in the context menu.

---

## Tag all (Ctrl+a) (Ctrl+A)

---

In order to tag all geometry objects and folders in the list view of the Model Explorer, press the key combination **Ctrl+a** or **Ctrl+A**.

This command is active in the tree representation (left) as well as in the list view (right).

---

## Select geometry objects (Shift+Ctrl+a) (Shift+Ctrl+A)

---

In order to only tag the geometry objects in the list view of the Model Explorer, press the key combination **Shift+Ctrl+a** or **Shift+Ctrl+A**.

This command is active in the tree representation (left) as well as in the list view (right).

---

## Cut (Ctrl+X) / Paste (Ctrl+V)

---

This command can be used to move groups or objects in the structure and between partial drawings and workplanes.

1. In the Model Explorer, tag the group(s) or object(s) you wish to move.
2. Select the key combination **Ctrl+X** or select the **Cut** option in the context menu.
3. In the tree structure, tag the node point under the group(s) or object(s) is to be positioned.



4. Press the key combination **Ctrl+V** or use the **Paste** option in the context menu.

**Please note:**

Groups and/or objects can also be moved in the structure and between different partial drawings and/or workplanes using

***Drag&Drop.***

Collect the required elements by keeping the Ctrl key pressed.

Whilst keeping the mouse button pressed, drag the object and/or groups to the required position within the tree structure.

## **Copy (Ctrl+C)**

---

2D and 3D objects, groups, parts and components can be copied using this command.

Tag the desired element and choose the command **Copy** or press **Ctrl+C** keys. Select the destination folder and choose **Paste** in the context menu or press **Ctrl+V** keys. The copied 2D/3D objects will keep their absolute position and will maybe on top of each other.

2D and 3D objects, groups, parts and components can also be copied in the structure tree of the Model Explorer by Drag&Drop with the pressed **Ctrl-key**.

## **Delete (Del.)**

---

The following objects can be removed from the model using the Model Explorer:

- 2D- and 3D-Groups
- 2D and 3D Parts and Components
- 3D workplanes
- 2D objects
- 3D objects
- Partial drawings

In the tree structure, tag the respective node point or, in the list view the respective objects and/or groups to be erased. Then press the **Del.** Key or select the **Erase** command from the context menu. The selected objects including the objects contained within them are removed from the model.

**Please note:**

The last partial drawing and the partial drawing for standard sheet area within a drawing, cannot be erased.

## Disassemble (Ctrl+Del.)

---

The **Disassemble** command can be used to remove groups from the structure. The content of the group (objects and subgroups) remain and, within the structure, move up by one hierarchical level.

If the command Disassemble is used pointing on a part or component only the part attributes are deleted and the part or component is changed to a group.

- Tag a group, a part or a component in the structure tree or collect them with the pressed **Ctrl-key** in the list view.
- Press the key combination **Ctrl+Del.** or select the **Disassemble** command from the context menu.

## Format, Sort

---

Using the **Format, Sort** option opens another menu, which contains many different sort criteria. Different sort criterias can be defined for the structure tree and for the list view.

- **Sort on/off (F6)**  
This option can be used to define whether one of the following sorts are to be used or whether the objects and groups are to be listed in the order in which they were generated in the Model Explorer.
- **Toggle sort direction (F7)**  
The elements in the structure tree or the list view are sorted ascending or descending.
- **Sort case sensitive (F8)**  
The sorting is case sensitive.  
Condition: The sorting is active (F6)
- **Sort logical (Ctrl+F8)**  
The sorting follows logic criteria. An example for the logic sorting are numbers. Without a logic criteria when sorting 2 digit numbers a 0 must be in front of all 1 digits. The logic criteria will sort first 1 digits and then 2 digits.
- **Sort groups separatly (F9) (only in list view)**  
The sorting differs between groups and objects. If this option is not active both are listed on the same level.  
Condition: The sorting is active (F6)

### **Format, Open / close branch (F11)**

---

This command can be used to open or close the selected branch in the tree structure including all the subordinate structures.

### **Format, Open all / close all (Shift+F11)**

---

This command can be used to open or close all branches of the entire model area right up to the lowest structure.

### **Format, View side by side (Ctrl+L)**

---

With this command the position of the list view can be changed to be below or beside the structure tree.

### **Format, View among each other (Shift+Ctrl+L)**

---

With this command the position of the message window can be changed to be below or beside the structure tree.

## **3 Working with *CADdy++ View & Plot (pro)***

### **3.1 Starting the program**

---

Once the installation program is complete and the computer has been restarted, the programs you have installed, can be started in the following ways:

1. Open the Windows Start menu.
2. Select ***Programs***
3. Select the designation of the program group you specified at installation for *CADdy++*.
4. In the next menu click the line *CADdy++ View & Plot*.
5. If you have installed *CADdy++ Model Management* as an additional module to this program, you are now shown the Login-Dialog window, in which you can enter your ***User name***.  
If the name you have entered has not been defined in the database, the internal name *Standard* will be used.

## 3.2 Display/remove symbol bar

---

Some of the commands in the menus can be reached more quickly if the symbol bar is displayed.

The symbol bar can be displayed by selecting the menu command **View, Display symbol bar**.


Start the program again to display the symbol bar.

If you do not want to display the symbol bar, select the menu command **View, Remove symbol bar**.

## 3.3 Activate/Deactivate ODMA

---

If you have installed *CADdy++ Model Management*, but want to open models with data that is not saved in the database for *CADdy++ Model Management*, you can deactivate the link to the database.

1.  Click the icon to start the **Activate/Deactivate ODMA** command.
2. Reply to the security prompt asking whether the current status of the model you are working on is to be saved or not.
3. If you have previously opened a model with data saved in *CADdy++ Model Management*, after you have acknowledged the message, a copy of the file will be made under the name **MODELL.MOD** in order to avoid possible loss of data. Confirm the message by clicking **OK**.



You will then see from the icon that the ODMA interface is no longer active.

You can activate the interface again by clicking the icon.

## 3.4 Open model

---


In *CADdy++ View & Plot* the models will be opened in the status that they had when saved. This means, for example, that only those partial drawings and layers are visible and available for printing that were visible at the time the models were saved.

If 2D or 3D windows were closed during the save procedure, you can still open them using the **Model Explorer** or using the command **View, New 3D Window** or **New 2D Window**.

## Open model without CADdy++ Model Management

---

When you are working without *CADdy++ Model Management* or the link to this module has been deactivated, use the following procedure to open a model:

1. Select the **File, Open model** menu command, or click the symbol  in the symbol bar.
2. You will see the name of the folder containing models in the **Find in** text box of the dialog that has just opened. The name is that specified using the **File, Folder** menu.  
You may perhaps have to select a different folder.
3. Tag the file you want in the list of MOD files, or type the name of the file in the **file name** text box.
4. Click the **Open** command button.

### Please note:

*CADdy++* models and templates are now protected against simultaneous access from different workstations. An unintentional parallel processing of the files is therefore no longer possible.

When opening a Model or a Template in *CADdy++*, an owner file with the same name is created in the same directory. If another computer tries to open this file, an error message appears, in which the current user is displayed.

## Options

---

Using the menu **File, Options** the following settings can be made:

### display thick lines

If this check box is active, 2D line objects which have been allocated a line width, will be displayed in this line width - in relation to the set 2D drawing dimensions. It is independent of the selected scale.

If this check box is not active, all lines are displayed in the width of a pixel.

Independent of this display, for print output, lines are always displayed which are allocated themselves or using layers.

### scale line pattern with factor (Model specific setting)

This factor influences the pattern length of the Windows standard

line types in the program: the larger the value, the longer the lengths of the lines as well as the distances between them.

## Determine Settings Grafic Windows

---

### background colour

This can be used to specify the background colour of all 2D and 3D windows.

1. Click on the button next to this entry.
2. In the colour selection dialog box, determine the required background colour and confirm with **OK**.

Depending on the background colour, the defined character colours are modified automatically so that the drawn objects are always visible.

**Please note:** You can swap between the background colours of black and white using the menu command **Window, Black Background** or **White Background**.

**Activate colour gradient in 3D windows:** With this option the colour gradient for 3 windows can be activated or deactivated.

**Gradient:** With this value the transition between the choosen colours can be defined.

**Colours:** The gradient will be shown between the two defined colours.

**Effect:** Starting from the first colour the direction can be defined here.

**Accept for all 3D windows:** When the option **Activate colour gradient in 3D windows** is active the actual settings can be adjusted to all 3D windows.

**Define colour gradient for each grafic area:** Activate this option if you wish to take over the settings of colour gradient from one grafic area to another grafic window or parts of it.

**Get parameters from current grafic area:** When choosing this option the colour gradient is taken over from the current 3D window.

**Apply parameters to current grafic area:** When choosing this option the parameters are passed only to the current grafic window.

In divided 3D windows the parameters are only passed to the current area.

**Apply parameters to current graphic window:** When choosing this option the parameters are passed only to the current graphic window.

In divided 3D windows the parameters are passed to all the graphic areas.

**Please note:**

Please be aware that the graphic performance can be lower with activated colour gradients.

---

## Determine save and recover settings

### Backup automatically

If this setting is active, the current status of the model or template is automatically set up in a backup file after a determined number of **actions** or **minutes**.

The backup file has the name of the currently processed model or template which has the character # proceeding.

E.g. a file with the name #MODEL.MOD will be set up for the current model MODEL.MOD. If the number is >1 the save files will be named according the key: **#<model name>-#nn#.mod**.

The folder for the backup files (\*.MOD) can be specified by using the **Settings, Folder** menu command.

### Generate drawing preview with automatic backup

If this setting is active, preview data is generated for each drawing contained in the model within the automatic file backup. If this option is deactivated, this will lead to speed advantages.

### Create backup (\*.bak) when overwriting models

If this setting is active, during a save procedure the previously saved file is backed up with the same file name in the same directory with the extension \*.bak.

This option affects models, templates and backup copied which have been saved at least twice. If this option is active, they will only be created if the *number of auto save files* = 1. The creation of \*.bak files in the work directory is not changed by this setting.

A \*.bak file cannot be loaded from the Explorer using Drag&Drop. However, you can select the file types \*.bak using the **File, Open Model** command. Then execute the **File, Save As** command, in order to reallocate the extension \*.MOD to the model.

### Number of Auto Save Files

With the setting *number of auto save files* it can be defined, how many auto save copies (\*.MOD) should be stored to a model in the current [folder](#) for the *auto save files*. If the number is >1 the

save files will be named according the key: **#<model name>-#nn#.mod**.

The file with the lowest number is the oldest one, the file with the highest number the newest one. In the range of the numbers for auto save files, the files will be updated according to the settings „backup after ?? actions“ or „backup after ?? minutes“.

### General options for file save

#### ***Background colour for drawing preview***

Using the colour command button, you can define the background colour for the preview window in the **File, Open Model** dialog box. This setting is saved in the Ui\*.SYS file after you quit the program.

#### **Number of Recent Files menu entries**

The list of the recent files is shown in the menu command **File**. Click on an entry to load the model.

### Create owner file

When opening a model or a template in CADdy++, an owner file with the same name and the file extension \*.lock is created in the same directory, which blocks the loading of the same model from another computer.

Models and templates are thus protected against simultaneous access from different workstations. Unintentional parallel editing of files is prevented.

If you disable this option, no owner file will be created. However, the opening of a file will still be blocked if an owner file has been created by another computer.

### Determine folder settings



---

In the menu command **File, Folder** it can be defined, from which folder the CADdy++ files should be loaded by default.

1. For every file type the list contains two fields:  
one with the file type name and one with the default folder.  
To define a new folder, double click the folder field or open the context menu with the right mouse button and choose the option **Select a new folder (Ctrl+N)**.  
How to change several folders in one step is described below.
2. In the following dialogue window select a new folder for the file type and confirm with **OK**.
3. To store the folder list, the desired folders must be activated.  
**Only those folders are saved, which are active ☒ .**



To activate all folders use the keys **Ctrl+A** (or context menu: **Select all**) and then the **space bar**.


4. Select the button **Save directory definition file** , to store the settings in the current file or select the button **Save directory definition file as...** , to store the settings with a different name. Define the name and the directory for the parameter file and click the button **Save**.
5. If changes were made, they can be confirmed with the button **OK** or **Apply**. Click the **Cancel** button, if changes should not be used.

### Edit the Folder List

Besides the above described possibility to create new folders, the entries in the list can be edited as follows. Please note, that changes can only be made, if the path and folder already exists.

- A directory can directly be changed in the column folder, by marking the field and select **Edit (F2)** from the context menu.
- To exchange a list of folders activate ☒ the file types to be exchanged first. Then select **Replace marked items (Strg+F4)** in the context menu. Enter the old partial folder name in the field **Search** und the new partial folder name in the field **Replace** always starting with the volume name . Confirm the **Replace** button for every folder or select **Replace all** to exchange all the marked folders in one step.
- If all folders should be exchanged by a new name first activate all file types. Then start the option **Select all (CTRL+A)** from the context menu and press the space bar. Select the option **Replace (F4)** and enter the old directory name in the field **Search** and the new directory name in the field **Replace** always starting with the volume name . Confirm the **Replace** button for every folder or select **Replace all** to exchange all the marked folders in one step.

### Load a folder list

1. Click the **Open directory definition MPS-file** button .
2. Select the desired MPS file.
3. Click the **Open** button.
4. Confirm the new folders by clicking the **OK** or **Apply** button.

### **Please note:**

When leaving the program the current settings for folders are stored in the files UIMVP.SYS.

# Open model using CADdy++ Model Management

## General procedure

---

If the ODMA link to the database of the Model Management is active, you can select and open models only by using *CADdy++ Model Management*.

Use the following procedure:

1. Select the **File, Open model** menu command.
2. Open the **Find** card.
3. Specify the **Object type** to define whether the data records of **3D models** or of **2D drawings** included in them are to be found. Then activate one of the three options for the **Drawing format**.
4. Define the search criteria for selecting records on this card, then start the procedure by clicking **Start search**.  
Further information on this procedure is given in the following section.
5. If entries matching the search criteria are found, acknowledge the message displayed by clicking **Yes**.
6. The models that have been located will be displayed in the **Found** tab card.  
Tag the model that you want to open and then click **OK**.

## Select record

---

You can use this option card to find records in the *CADdy++ Model Management* database to the models or 2D drawings you are looking for.

### Object type

After you have executed the **File, Open model** command to indicate whether the options are to be used to search for the records on **2D objects/drawings** or on **3D objects/models**.

After you have activated the **3D object/model** option, the program will restrict its search to those records that refer to files with MOD as the extension.

After you have activated the **2D object/drawing** option, the program will restrict its search to records that refer to 2D drawings.

In this case you can specify a more precise criterion so that the search

will be just for **CADdy drawing formats**:

By using the setting **New drawing format \*.MOD** you will restrict the search to 2D drawings in model files; the setting **Old drawing format \*.PIC** will restrict the search to “old” CADdy PIC files.

Finally, with the option **Both formats** you will find both MOD and PIC format files, as long as the criteria have been specified correctly.

### **Drawing data**

You can type suitable search criteria in the input boxes that are active. Wildcards are also permitted anywhere.

The list fields for the date to be entered will always contain the system date, but there is a calendar for editing it. Use the **Date specification** to specify whether the search is to be **Before** or **After** the date that has been set. You can use a similar method to specify the **Time period**.

Use the list field **Free search** to select another search field. The criterion used here can be typed in the box to the right of the field. A special feature is that indicated by the search criterion **Converted from CADdy PIC**. Use this setting to display the records to find all the records that have been converted from PIC to MOD.

If search criteria have already been specified, you can select and activate them by clicking **<<Previous search criteria** and **Next search criteria >>**.

### **Start search**

Click the **Start search** command button to start the search.

After you have done so, you will see a box informing you how many records have been found.

If you confirm the message by clicking **Yes**, the **Found** dialog box will appear with the complete list of records.

But if you click **No** instead, you can expand or completely change the search criteria. Afterwards start the search again.

### **Select record**

You will see a list of all the records found as a result of the most recent search in the option card headed **Found**.

The list is arranged according to the first field.

By clicking the column headings you can sort according to the contents of other fields, either upwards or downwards.

Select a specific record by clicking it.

Then you can do any of the following:

- You can display all the data by selecting the **Data** option card. If you have searched for 2D drawings, you can display their **Modification history**.
- You can load the model that goes with the record. Confirm this step by clicking **OK** or by double clicking the record you have selected.

## Layer Manager in the Model Explorer

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In the model explorer, to be opened using the menu command **View, Model Explorer**, there is the option card **Layer Explorer**.

Choosing this option card the following icons appear:



### Layer Scan On / Off

Using the displayed icon the layer scan is turned on and off. In activated mode only the objects on the tagged layer are displayed.

**Important:** The scan mode is only valid for the tree structure not for the list view.



### New Layer tagged / not tagged

If generating a new layer this new layer is tagged when this icon is active. Generating another new layer with the shortcuts **Ctrl-N** or **Ctrl-n** will sort this layer inside the first new layer and automatically offers the command to **Rename**.

If this icon is not activated new layers are on the same level without offering the command to **Rename**.



### Localize Object in Model Tree

Start the command to determine for 2D and 3D objects, to which layer the object is related. Identify an object in the current graphic window and the layer, related to the identified object, is marked (blue background) in the Structure Tree.

The other command buttons are used to define **Layer as...** for special objects.



### Tree view active / List view active

If the **Tree view** is active then in the detail view only those layers are displayed which are one step beyond the selected layer.

If the **List view** is active all layers beyond the selected layer are displayed in the detail view.



### Clear message box

If the option **Use Model Explorer to display messages** is

activated in **Settings, Options, User Interface** with this command the content of the message box can be deleted.



Use this command to delete all texts. Single texts can also be deleted using standard text commands.

## Context menu in the Layer Explorer





---

The layer context menu is used to process layers. It is opened by placing the cursor somewhere in the left side of the Layer Explorer and clicking the right mouse button.

### Layer structure

The left section of the dialog box contains the structure of the layers as a hierarchy.

Below the basic layer with the description *Standard* there are the descriptions of the separate layers or, where applicable, branches of the layers. The symbols alongside the layer descriptions denote the object type for which these layers are used.

The layer branches are denoted by the symbols   placed alongside. You can open a branch by clicking  and close it by clicking . If the layer is tagged, you can use the two keys + or – instead.

### Commands in the context menu

Using the context menu in the Layer Explorer various settings can be made and also layer can be defined or erased. The following commands can be used in the context menu.

## **New: Generate layer (Strg+n)**

---

Before layers can be used for drawing 2D objects, they must firstly be generated and defined their attributes.

Layers with their own names and attributes should be defined in a template. If saved in a template, all models generated on the basis of this template are available.

When starting *CADdy++*, one of the templates STANDARD.TPL, PROTO2D.TPL or PROTO3D.TPL is loaded. Some standard layers with their own display attributes have already been defined in this template. These can be used as stated, modified slightly to meet your requirements or changed completely.

The layer structure in *CADdy++* can be set up as a directory tree with different hierarchical levels.

For later processing of layers and the attributes allocated to them, this

structure should be formed simply. The hierarchy of this structure should be kept with as few levels as possible.

1. Select the menu command **View, Model Explorer**.  
Choose the option card **Layer Explorer**. below which you want to generate a new layer. Then click the right mouse button.


If the layer structure is to remain simple, select the basic layer.

2. Select the command **New..., New Layer (Ctrl+n)** in the context menu.

If layers are to be generated for dimensions, for which dimension text, dimension lines and dimension delimiters are to have different display attributes, select the *Dimension* option. Otherwise, select the command **New dimensioning layer (Shift+Ctrl+n)**.

If you do not need to make this distinction, you must select the command **New Layer** as well for dimensions.

The new layer is added to the end of the layer list. Independent of the selected option, the new layer is added to the end of the list. It receives the name **New\_layer** from the program. If a number of layers are generated one after the other, further layers receive the names **New\_layer0**, **New\_layer1**, etc.

When the command  **Tag new layer** is active, the description of the layer can be entered immediately.

If the **New Dimension Layer** option has been selected, four layers for dimension text, dimension lines, dimension delimiters and tolerance entries are generated on the next hierarchical level.

## Erase: Erase layer

---

When you want to erase a layer or all the objects in a layer, or want to complete both these actions, use the following method:

1. Select the menu command **View, Model Explorer**.  
Choose the option card **Layer Explorer**
2. Move the cursor to the layer structure and then tag the layer to be dealt with. Then click the right mouse button.
3. Next, choose one of the following commands:

### **Erase, Erase layer only (Del.)**

This action will delete the layer. The objects linked to this layer will now be linked to the higher up in the layer structure hierarchy.

### **Erase, Erase geometry only (Ctrl+Del.)**

This action will delete all the objects that are linked with the layer.

**Erase, Erase layer and geometry (Shift+Ctrl+Del.)**

This action will delete the layer and all the objects that are linked with it.

**Delete, Delete empty layers (Alt+Del.)**

All layers are deleted, which do not contain any object. To delete all empty layers in the model place the cursor on the base layer **Standard** and choose the command **Delete, Delete all empty layers** from the context menu.

## **Properties: Define display attributes of layer (Shift+F2)**

---

Objects (2D objects and 3D polylines) linked with a layer are displayed with the colour, line type and line width which have been defined for the current layer.

This does not apply to objects with display attributes specified as related to objects.

Line type and line width are ignored with the Truetype fonts.

Display attributes can be specified for one layer or several layers or also changed by using the following method:

1. Select the menu command **View, Model Explorer**.  
Choose the option card **Layer Explorer**
2. **Processing single layers:**  
Tag the layer and click the right mouse button. Select the command **Properties (Shift+F2)** in the context menu or double click the layer in the list view.

**Processing several layers**

When you want to allocate the identical attributes to several layers, press down *Ctrl* or *Shift* and tag them in the right section of the **Layer Explorer**.

Click the right mouse button and select **Attributes** in the context menu.

3. Select the **colour** and **line type**, and enter a value for the **width** in the box for the line width.

When you select a user-defined line type, click the line in the line types list for user-defined lines, and then select the line type in the dialog box.

4. Confirm the modifications by clicking on the **Accept** command button.

If there is already a link between the layer(s) and 2D objects with

layer-related attributes, they will be re-drawn using the display attributes that have been specified.

5. They attributes for other layers can be specified by repeating Step 2 to Step 4.

**Please note:**

If the display attributes are to be modified - independent of the layer, use the menu command **Process, Object Display**.

The display of the Windows system line types can be influenced, using the menu command **Settings, Options** and entering a factor in the **scale line pattern with factor**, by which the pattern length is to be enlarged or reduced. The defined value is stored separately for the current graphic window.

Use the menu command **Settings, Options** to determine whether lines are to be displayed with the allocated dimensions in relation to the dimensions of the drawing sheet or with a standard line width.

In the 3D-Window user-defined line types are displayed as a continuous line.

## **Process: Rename layer (F2)**

---

Layers can be given any clear name.

1. Select the menu command **View, Model Explorer**.  
Choose the option card **Layer Explorer**.
2. In the list of previously defined layers, click on the name which is to be modified.
3. Enter the key **F2** or or select the option **Process, Rename** in the context menu. A text box will appear so that you can type the name.  
Complete the procedure by pressing *Enter*.

**Please note:**

If drawings are to be converted in to DXF or DWG format, the layer names should only contain the characters A-Z, 0-9, -, \_. It must also be noted that the DXF/DWG format does not support the CADDy++ differentiation between upper case and lower case characters.

## **Process: Search layer (F3)**

---

A layer can be searched in the Layer Explorer.



1. Select the menu command **View, Model Explorer**.  
Choose the option card **Layer Explorer**.
2. Click in the Layer Explorer and enter the key **F3** or select **Process, Search** in the context menu.
3. Enter the search string in the dialog box. The search is started clicking the command button **Search**. Every click on the command button will show the next search result.  
The search options can be optimized using the options *only complete words* and/or *Capital/small letters*. Wildcards are not allowed.

---

### Process: Replace (F4)

---

The descriptions of layers can be replaced.

1. Select the menu command **View, Model Explorer**.  
Choose the option card **Layer Explorer**.
2. Click in the Layer Explorer and enter the key **F4** or select the command **Process, Replace** in the context menu.
3. Enter in the text box **Search for:** the text to be found. Enter in the text box **Replace by:** the new text. Click on the command button **Search, Replace** or **Replace all**.  
The search is restricted to the beginning of the word. Wildcards are not allowed.

---

### Process: Update (F5)

---

The display of the Layer Explorer can be updated.

1. Select the menu command **View, Model Explorer**.  
Choose the option card **Layer Explorer**.
2. Click in the Layer Explorer and enter the key **F5** or select the command **Process, Update** in the context menu.

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### Cut and Paste: Tag all (Strg+A)

---

All layers in the Layer Explorer can be tagged in the list view for example to move the structure, to edit or delete all layers.

- by defining a rectangle over all layers with the pressed mouse button.
- by the key combination **Crtl+A** in the list view.
- by selecting the option **Cut and Paste, Tag all**.

## Cut and Paste: Cut (Ctrl+X) / Copy (Ctrl+C) / Paste (Ctrl+V)

---

To move or copy layers in the Layer Explorer follow these steps:

1. Select the menu command **View, Model Explorer**.  
Choose the option card **Layer Explorer**.
2. Tag in the list view the layer(s) to be moved or copied.
3. To move the layers enter **Ctrl+X** or select **Cut** in the context menu.  
To copy the layers enter **Ctrl+C** or select **Copy** in the context menu.
4. Tag the layer in the Layer Explorer where the layers should be pasted.
5. Enter **Ctrl+V** or **Paste** in the context menu.

### Please note:

Layers can also be moved / copied using the standard Windows Drag&Drop command.

## Cut and Paste: Move geometry between layers (Shift+Ctrl+V)

---

To move geometries from one layer to another follow these steps:

1. Select the menu command **View, Model Explorer**.  
Choose the option card **Layer Explorer**.
2. Tag in the list view the layer(s), which contains the elements to be moved.
3. enter **Ctrl+X** or select **Cut and Paste, Cut** in the context menu.
4. Tag the layer the elements should be inserted to.
5. Enter **Shift+Ctrl+V** or select the option **Cut&Paste** or **Geometry, Insert Geometry**.

The objects gets the display properties from the new layer, except when they have object related display properties.

If the new layer has the property not visible, the objects are not displayed when inserted.

## Geometry: Move selected geometry to layer

---

Selected objects can be moved to a layer. Hereby selected elements on different layers are moved to one layer.

1. Select 2D or 3D objects.

2. Select the menu command **View, Model Explorer**.  
Choose the option card **Layer Explorer**.
3. Tag the layer the selected objects should be moved to.
4. Enter **Strg+M** or select **Geometry, Insert Geometry** in the context menu.

## Geometry: Erase geometry on a layer (Strg+Entf)

---

To erase the complete geometry on a layer follow these steps:

1. Select the menu command **View, Model Explorer**.  
Choose the option card **Layer Explorer**.
2. Tag the layer on which the objects should be deleted.
3. Enter **Ctrl+Del** or select the command **Geometry, Only Erase Geometrie**.

## Geometry: Select using layer (Shift+Strg+A)

---



To select all objects on a layer follow the next steps:

1. Choose from the menu **View, Model Explorer**.  
Change in the **Model Explorer** to the tab **Layer Explorer**.
2. Select the layer assigned to the objects to be selected.
3. Press **Shift+Strg+A** or choose from the context menu on the selected layer **Geometry, Selected layer geometry**.

## Display of the list view in the Layer Explorer

---

It can be defined whether in the list view all layers should be displayed or only assigned layers or only empty layers.

1. Select the menu command **View, Model Explorer**.  
Choose the option card **Layer Explorer**.
2. Select the layer in the structure tree which is on top of the layers to be displayed as empty or used layers.
3.  /  Select between Tree view and List view. If the **Tree view** is active then in the detail view only those layers are displayed which are one step beyond the selected layer.  
If the **List view** is active all layers beyond the selected layer are displayed in the detail view.

### Note:

To switch the display between all *empty layers* and all *used*

layers select the base layer Standard and choose the **List view**.

4. Using the context menu in the list view select the option **Display, ...**
5. Select between the following possibilities:



**Display all layers (B)**

All layers are displayed which are below the tagged layer in the Layer Explorer.



**Display assigned layers (Ctrl+B)**

All layers assigned to geometry are displayed which are below the tagged layer in the Layer Explorer.



**Display empty layers (Shift+B)**

All empty layers are displayed which are below the tagged layer in the Layer Explorer.

## Layer:Sort Layer Explorer

---

Using the **Format, Sort** option in the Layer Explorer opens another menu, which contains many different sort criteria.

Open the context menu in the layer structure tree or the detail view. Choose the option **Format, Sort** and then:

- **Sorting On/Off (F6)**  
Turns the sorting on or off.
- **Toggle sort direction(F7)**  
After pressing the function key **F7** or after selecting this command the structure tree or the detail view is sorted ascending or descending if the sort is turned on.
- **Sort case sensitive (F8)**  
After pressing the function key **F8** or after selecting this command the sorting is case sensitive or not if the sort is turned on.
- **Sort logical (Strg+F8)**  
With this command the sorting follows logical criterias. An example for a logical sorting is the sorting of single and double digits numbers. Usually a zero must be added in front of the single digit numbers to get the correct result. With the logical sort this numbers are sorted correctly.

## Settings for displaying the Layer Explorer

---

Using the context menu **Format** different settings can be made for the display of the Layer Explorer. The options are depending if the context menu is called in the layer tree or in the list view.

- **Open / close branch (F11)**  
This command can be used to open or close the selected branch in the tree structure including all the subordinate structures.
- **Open all / close all (Shift+F11)**  
This command can be used to open or close all branches of the entire layer area right up to the lowest structure.
- **Tile horizontal / vertical (Strg+L)**  
This command defines if the structure tree and the list view are tiled horizontally or vertically.
- **Toggle message splitter (Shift+Strg+L)**  
This option defines the position of the message box to be beside or below the layer structure tree.
- **Extended text display (Strg+S)**  
This command can be used to define if the column headlines are shortened or completely written.
- **Display of columns in the list view**  
This command is used to hide or display columns in the list view. Click in the context menu on the column to be displayed or hidden. The setting is stored when leaving the program.  
  
The sequence of the columns can be changed by clicking on a column and move it to the new position with the pressed mouse button.  
The width of a column can be changed by clicking between two columns and moving the cursor with the pressed mouse button.

## Define visibility of layers

---

Layers can have the attribute visible or hidden. This will also have a bearing both on the 2D objects as well as 3D polylines.

Objects and solids which are linked to layers which have the attribute as hidden, are not displayed on the screen and are not printed.

In one way, parts of the drawing which are not required at the current time can be suppressed, e.g. hatchings, and when the drawing is redrawn, the process is faster.


In another way, the parts of the drawing to be printed can be controlled using visible or hidden.

You can specify or change the visibility of any layer(s) by using the following method:

1. Select the menu command **View, Model Explorer**.  
Choose the option card **Layer Explorer**.

## 2. Processing single layers

Double click the icon  on the layer to be hidden.

When double clicking the icon  the objects on the layer are visible again – but not active also.

**or**

Double click the name of the layer in the list view that you want to deal with to open the dialog box for processing attributes.

You can also click the right mouse button to get the context menu and then select the command **Attributes**.

## Processing several layers

When you want to make several layers visible or hidden, press Ctrl or Shift and tag them in the right-hand section of the dialog box.

Click the right mouse button on one of the tagged layers and select **Attributes** in the context menu.

3. Deactivate or activate the **visible** setting.
4. Confirm the modification by clicking on the **Accept** command button.  
If the setting is deactivated, the objects of this layer will be erased from the screen.  
If the setting is activated, the objects are displayed.

Layers can also be faded in or out using the context menu in the Layer Manager..

By clicking the right mouse button on a layer with the command **Visibility and Pickability** the following options can be selected.

### Context menu

in structure tree	in the list view	key command
visible / invisible	visible / invisible	O
active / inactive	active / inactive	P
Toggle visibility incl. child groups		Strg+Alt+O

Toggle pickability incl. child groups		Strg+Alt+P
Tree visible from here	visible	Strg+O
Tree invisible from here	invisible	Shift+Strg+O
Tree active from here	active	Strg+P
Tree inactive from here	inactive	Shift+Strg+P

**Please note:**

Layers for which the visible setting has been deactivated, will be allocated the inactive attribute (**active** setting deactivated).

If the layer is made **visible** again, the inactive setting is retained.

If a layer which has previously been defined as hidden is to be visible and active at the same time, the **active** check box should be activated.

Using the menu **File, Read/Write Parameters** the properties for existing layers can be read from an MPS file. Precondition for that is that the layers have identical names.

## Define access options of layers

---

Layers can be allocated the *active* or *inactive* attribute. This will also have a bearing both on the 2D objects as well as the solids.



2D objects or solids which are linked to layers which have the inactive attribute, cannot be selected using the cursor.

In this way, it is possible to limit the selection of objects. In the same way, objects can be protected against modifications.

The access options relating to any layer(s) can be specified or changed by using the following method:

1. Select the menu command **View, Model Explorer**.  
Choose the option card **Layer Explorer**.

2. **Processing single layers**

Double click the icon  on the layer to be deactivated. When double clicking the icon  the layer is active again.

**or**

Double click the name of the layer that you want to deal with to open the dialog box for processing attributes.

You can also click the right mouse button to get the context menu and then select the command **Attributes**.

### Processing several layers

When you want to allocate several layers the identical access options, press Ctrl or Shift and tag them in the right-hand section of the dialog box.

Click the right mouse button and select **Attributes** in the context menu.

3. Deactivate or activate the **active** setting.
4. Confirm the modifications by clicking on the **Accept** command button.

Layers which were previously hidden, will also be allocated the **visible** attribute when allocating the **active** setting.

Layers can also be activated or deactivated using the context menu in the Layer Manager..

By clicking the right mouse button on a layer with the command **Visibility and Pickability** the following options can be selected.

#### Context menu

in structure tree	in the list view	key command
visible / invisible	visible / invisible	O
active / inactive	active / inactive	P
Toggle visibility incl. child groups		Strg+Alt+O
Toggle pickability incl. child groups		Strg+Alt+P
Tree visible from here	visible	Strg+O
Tree invisible from here	invisible	Shift+Strg+O
Tree active from here	active	Strg+P
Tree inactive from here	inactive	Shift+Strg+P

#### Note:

Using the menu **File, Read/Write Parameters** the properties for existing layers can be read from an MPS file. Precondition for that is that the layers have identical names.

### Determine layers for special objects

All 2D objects which have been generated using the CAD Menu, **Draw 2D** menu, are linked with the current drawing layer.

In order to link these geometry elements with different layers (e.g. because they are to have different line widths and line types when



plotting the drawing), one of the previously defined layers must be determined as the *current layer*.

For other 2D objects such as text, dimensions, hatchings, etc. such differentiation is not required as such or at all. One layer can be defined for such objects, with which the objects are automatically linked when generated.

Therefore, hatch lines are not linked to the current layer when generated, but are linked to the hatch layer, text is linked to the text layer, etc.

1. Select the menu command **View, Model Explorer**.  
Choose the option card **Layer Explorer**.
2. Tag the layer that is to be reserved for special objects.
3. Select from the the grey command buttons in the top which special objects should be assigned to this layer.



**Current drawing layer**



**Texts**



**Construction aids (green symbol)**



**Hatching**



**Dimensions**



**Centre lines**



**Hidden lines (2D), i.e. standard parts and disassembled model views**



**Hidden edges (3D), i.e. model views**



**Standard parts thread lines**



**Transition edges in model views**

**Please note:**



A layer, chosen for multiple object types, is marked with the displayed icon.

If the layer, chosen for dimensioning, does not consist of 4 sub layers also this layer is marked with the displayed icon.



All layers, not chosen for any object type, are marked with the displayed icon.


If e.g. text is to be linked to various layers in order to plot in various colours or line widths, generate the required layers and allocate

them with the required names, e.g. *text\_05* and *text\_07* etc.  
Before you begin to add the text, choose one of the layers in the **Layer Explorer** dialog box and then specify it as the **Text Layer**.



## Display layer content

---

If the 2D objects are to be displayed which are linked to individual layers, therefore by "paging" through the drawing layers - so to speak, proceed as follows:

1. Select the menu command **View, Model Explorer**.  
Choose the option card **Layer Explorer**.
2.  Click the command button **Scan layer off**.
3. In the layer structure click the name of the layer containing 2D objects you want to see, or use the cursor control keys.

The only objects that will be displayed are those linked to the layer that is currently tagged.

Closed node points   Dimensions can be opened using the Arrow right key during the scan. Opened node points can be closed using the arrow left key.

The command **Layer Scan** stays active, also when closing the Model Explorer.

### Please note:

You can use **Display, In Use** when in the right-hand section of the dialog box to display and check just the layers linked to objects.

## Move layers

---

Layers can be set up structured in the Layer Manager. The dimensioning layer for example, is automatically generated with sublayers. If you later wish to modify the structure of the layer tree, you can move layers in the structure. Proceed as follows:

1. Select the menu command **View, Model Explorer**.  
Choose the option card **Layer Explorer**.
2. Position the cursor in the left half of the Layer Manager on the required layer in order to move a layer or, in the right half of the layer structure, tag a number of layers that you wish to move in the structure and confirm the right mouse button on one of the tagged layers.
3. Click on one of the tagged layers and keep the mouse button pressed.

4. Move the layer(s) in the structure tree on the layer, where it should be inserted.

## 3.5 Import 2D data

from **view & plot pro**



Using the menu command **File** files can be in *CADdy* PIC or *CADdy* INF/LAY format and in DXF/DWG format.

### Import *CADdy* PIC files




from **view & plot pro**


Select the **File, *CADdy* PIC File** command in order to load a *CADdy* PIC file.

When this command is being executed, a new model will be created, i.e. the current model will be erased first.

1. In the ***CADdy* Settings** option card of the dialog box which then appears, determine the drive, folder and names of the *CADdy* drawing to be loaded in the **drawing file** text box.  
This can be entered in the text box or be searched for using the  command button and then selecting the file.
2. If an info file (\*.INF) and layer file (\*.LAY) already exists with this name in the same folder, their drive, folder and names are entered automatically in the **layer file** and **info file** text boxes.  
Otherwise, enter the file names or determine them if necessary, using the  command button.

If no info file is entered, the object specific display attributes of the *CADdy* objects are taken over.

3. If the *CADdy* drawing to be loaded contains symbols, determine the folder in the ***CADdy* symbol folder** text box, in which the symbol files are to be searched for.  
Use the command button  in order to select a folder and add to the list.  
If a folder is to be removed from the list, tag the respective one and then activate the  command button.
4. In the ***CADdy* font folder** text box, enter the folder in which the *CADdy* fonts, determined with the info file, can be found.  
This can be entered in the text box or be searched for using the command button  and then a folder selected.

5. In the **CADdy plotter configuration file** text box, enter the folder and name of the \*.PLN file where the pen definition is stored. This can be entered in the text box or be searched for using the command button  and then a folder selected
6. **Only** in case, **no** INF-file is loaded for the import, it can be defined in the option **Accept Display Attributes**, if the **colour**, **linetype** and **linewidth** attributes of the objects are taken over from the object- or from the layer- information. If the option is active, the object information is used. The colour button can be used to define, which colour should be used for the **CADdy** standard colour.
7. The **Open Protocol File** command button can be used to open the import protocol of the last import in order to be able to view any useful information or errors that may have occurred.
8. In the area of the **CADdy objects** determine the objects to be converted, which are then to be taken over in **CADdy++** as object oriented.

**Fill areas** are not imported if the option is deactivated. If the option is activated, the fill areas are displayed by polylines.

**Colour areas** are not imported if the option is deactivated.

**Images** (pixel drawings) are not imported if the option is deactivated.



**Dimensionings** are taken over as geometry if the option is deactivated, not as associative dimensioning. As such, however a 1:1 illustration of the PIC file guarantees dimensioning. "Old dimensions" in CADdy drawings are generally taken over as geometry. Every non associative dimension lies in a separate group.

**Components** are imported as pure geometric data without special group structure. All component information will be lost.

**Standard parts** are imported as pure geometric data if the option is deactivated (individual elements such as lines and circles) and can no longer be edited as 2D standard parts.

**Shafts** are imported as pure geometric data if the option is deactivated (individual elements such as lines and circles).

9. Change to the **CADdy++ Settings** option card in order to make the necessary determinations for archiving symbols, the scale definition and units.

10. In the **target library** text box, determine the drive, folder and name of the symbol library in which the symbols contained in the *CADdy* drawing are to be archived. This can be entered in the text box or be searched for using the command button  and then selecting a file.
11. Within the symbol library, determine the folder in which the symbols are to be archived.  
This can be entered in the text box or be searched for using the command button  and then selecting a folder.
12. In the **Accept symbol** block, select one of the following options:
  - accept all symbols**  
All symbols in the target library are taken over which are contained in the *CADdy* drawing.
  - check all symbols**  
This checks whether symbols of the same name exist in the target folder of the target library. If this is the case, confirm whether the symbol is to be overwritten in the target folder.  
New symbols are taken over without a prompt.
  - only accept new symbols**  
Only the symbols from the *CADdy* drawing which do not exist in the target library are taken over.
13. If the PIC file contains text, in which an apostrophe appears, the text that follows this character is not displayed in *CADdy* and not printed.  
If this text or text parts however are to be imported, activate the option **accept hidden text**. They are retained during a work session.
14. Under the heading **accept scale**, activate one of the following options.
 

☒ accept scale from partial drawings

The scales available in the *CADdy* drawing file are interpreted and taken over as partial drawing orientated.

☐ accept scale from *CADdy* Info file

This option is necessary for older *CADdy* drawing files, since the scale information for the drawing is saved in the *CADdy* INFO file. Warning! With this setting, it is imperative that the respective *CADdy* INFO file is available.

☐ define scale manual

This option is necessary for older *CADdy* drawing files, if the scale is controlled using an INFO file, but no INFO file exists. Within this option it is necessary to enter the scale required for the conversion of the *CADdy* drawing file.

**15. Accept layers**

**Accept layer number and name:** If this option is active, the numbers and the names of the layers are imported. The name of the layer in **CADdy++** will start with the number of the layer.

**Generate 3 digits number and accept name:** If this option is active, all the numbers of the layers will have three digits.

**Accept all layers:** If this option is active, all the 512 layers from **CADdy** are imported, not only the used ones as usually.

**16.** Determine the unit to be used for the drawing to be converted.  
Example: drawing in unit mm, architecture drawing in cm; dm or m

**17.** If the CADdy-PIC file contains components, in the register **Component** the definitions to take over the component informations can be defined.

See description in chapter **Transfer component informations from PIC files to CADdy++**

**18.** Start the import of *CADdy* drawings using **OK**.  
During the import procedure, this can be cancelled at any time by pressing any key.

**19.** The drawing boundaries of the PIC drawing and the current *CADdy++* drawing, cannot be compared to each other when importing. Open the dialog box **Settings, 2D Drawing**, deactivate and activate the drawing boundaries one after another. The drawing boundaries of the imported PIC file are then to scale.

**Please note:**


*CADdy* PIC files can also be imported by Drag&Drop. Info- and layer-Infos are not considered then. A PIC file imported that way is generated in a new window if it is dropped in the program window. If it is dropped in a drawing window it will be added to the elements in this window.

---

**Import from CADdy INF and LAY files**

from **view & plot pro**

The settings saved in a *CADdy* INF file and a *CADdy* LAY file can be loaded by using the **File, CADdy PIC File** command.

1. The option card **CADdy Settings** includes **drawing file**: leave this blank.
2. The entries in the text boxes for the **layers file** and the **info file** must contain the names of the info file (\*.INF) and the layers file (\*.LAY) that are to be loaded; an alternative is to click the  icon to locate them.

**Please note:**

Info files can be imported using Drag&Drop.



Since font and dimension sizes in *CADdy++* could be related to the paper and are therefore scale independent, these values should be defined in the *CADdy* INF file for the scale of 1:1.

Only the occupied layers of a *CADdy* drawing are taken over into *CADdy++*.

## Transfer component informations from PIC files to *CADdy++*

from **view & plot pro**

To transfer component informations from PIC files to *CADdy++*, please follow the Steps below:

1. In the *CADdy* directory text box, enter the folder, where the file **CADdyST.SYS** is stored. This can be entered in the text box or be searched for using the command button  and then a folder selected.
2. Activate the command button **Read CADdyST.sys**, to read the component informations from this file.  
The values are entered in the fields below.
3. Define in the left field below the line **Definition of the primary attribute** one or more attributes, where the primary attribute in *CADdy++* should be built of. The primary attribute, which is defined in the **component** menu in *CADdy++* is shown in the right field.  
By activating the command button  the attributes are added in the above field. The definition of the primary attribute can be supplemented by arbitrary characters.

**Example:**

- Arbitrary characters in the definition of primary attributes: "BT-" and "-"
- Use the *CADdy* attributes: "\$(**name**)" and "\$(**ID.No.**)"

- is combined as “**BT-\$(name)-\$(ID.No.)**”
- the result after import would be “**BT-Hexagon Nut-12129**”

4. The attribute names in the right fields can be changed or the **CADdy** attribute names can be used.  
If you delete names in the right fields, those attributes are not converted to **CADdy++**.

## Import CADdy files: Technical information

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from *view & plot pro*

When *CADdy* files are being imported the program will take the following objects/structures into consideration:

### PIC files

- Drawing sizes
- Partial drawings will be taken over with their names and scales
- Drawings will be loaded with the colours and line widths and line types allocated as attributes.  
During this procedure the line types will be altered so that they approximate those utilised in Windows.  
User-defined line types will not be supported.
- Dimensions specified in versions as of *CADdy* Version 14.0 will be altered to editable dimensions, and the dimensions in older versions will be taken over as lines and text.
- Hatching will be taken over as individual lines.
- Clusters will be taken over as groups that will be given CLUSTERnn as their names.
- Shafts processed in the Shaft Editor can be processed.
- Component information will be taken over as attribute text.
- Standard parts which are available in *CADdy++*, are taken over as standard parts according to their display in *CADdy*.  
When editing standard parts whose representation type is not supported in *CADdy++*, their display will be modified.  
Standard parts which are not available in *CADdy++*, are taken over as line objects.
- Line widths (pen thickness) defined using layers are not taken over.
- Fillings and colour areas are not taken over.



- Walls from architecture are taken over as parallel lines.
- Only CADdy layers are taken over. They are inserted on the layer structure under the layer with the description *Standard*, whereby unnamed layers retain their number as the description, all others retain the descriptions entered in the LAY file.

If the layer *Standard* is renamed, a layer is set up below this layer with the description *Standard*, under which all CADdy layers are inserted.

No individual layers are set up for dimensions. Instead they are linked with the current layer for dimensioning. If the dimensioning in CADdy++ has a dimensioning structure, the partial elements of the dimension are linked with these layers.

- Dimensions from CADdy are taken over in their display 1:1 into CADdy++. If the drawing is later scaled in CADdy, the dimensions are taken over into CADdy++ as edited dimensions and are held in the prefix field so that the dimension does not appear as underlined.

### **INF and LAY files**

If CADdy INF files and, possibly, CADdy LAY files are being imported, the following data will be taken over:


- Below the layer with the description *Standard*, the loaded layers are set up with the original description or with an automatically numbered description. Layers whose description already exists, are not set up.

If the layer *Standard* is renamed, a layer is set up below this layer with the description *Standard*, under which all CADdy layers are inserted.

Colour and line types of layers are taken over.

### **Please note:**

In CADdy++ the object-related display attributes take precedence over attributes specified for layers.

If you want to ensure that all the objects are displayed using the layer attributes, you will have to select all the objects first. This is done by activating the **Process, Object Display** command and then deactivating the setting for *object specific* by clicking .

Since font and dimension sizes in CADdy++ could be related to the paper and are therefore scale independent, these values should be defined in the CADdy INF file for the scale of 1:1.

CADdy drawings that are to be plotted to a specific scale can be scaled by means of a scale factor, i.e. the drawing dimensions and the sizes of all the elements in these drawings can be enlarged or reduced by this factor. This setting will also effect the line widths allocated to the elements as attributes.

### **Symbols**

If you want to be able to transfer all A-symbols that you have set up in CADdy over to CADdy++, there is the option of loading all the symbols which lie in a directory into one CADdy PIC file using a Plus program <SybinPic.vab>.

Finally import the CADdy file into CADdy++ to save all symbols in the currently set symbol library and they are then directly available with their descriptions.

#### **Please note:**

So that a symbol library remains clear in its overview in CADdy++, it is advisable to only take over a certain amount of symbols in one step. Therefore, structure your CADdy symbols first of all and combine them into specific topics before loading them into the CADdy drawing.

### **Import 2D DXF/DWG files**

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from ***view & plot pro***

With the version CADdy++ 2019 a new DWG/DXF converter has been integrated. This converter can be selected directly from the menu command ***File, Import, DXF/DWG File ....***

#### **Import with the current converter**

from ***view & plot pro***

In order to load files in DXF or DWG format, select the menu command ***File, Import, DXF/DWG File (new) ....***

When executing this command, the DXF/DWG file is imported into the active 2D window of the current model. If necessary, set up a new model or a new drawing within the model if you do not wish to import the DXF/DWG file into the opened active 2D window.


If the DWG-file contains 3D SAT data, a top view is generated during the import and placed as a symbol in the active 2D drawing.

In addition, the SAT bodies can also be imported from the DWG file and imported into the 3D area. To do this, activate the ***Import SAT bodies*** switch in the ***DXF / DWG Import*** dialog. If bodies are found in blocks

and are used more than once, the structure is adopted and displayed as groups.

For the case that you have accidentally imported the drawing in an incorrect file or an incorrect drawing, you can undo the step just made using the **Undo** command button.

1. In the **Source file** text box, determine the drive, folder and name of the file to be imported.

This can be entered in the text box or be searched for using the  command button.

2. **Unit / Factor**

If the file to be imported contains information about the unit used during creation, this information is determined automatically. Otherwise, select the unit in which the DXF/DWG file is to be imported from the list or enter a factor in the form of a number. The unit or factor determines the magnification or reduction factor for the imported elements. (For hall plans or floor plans, for example, the unit Meter usually makes sense when importing).

3. **Check and adjust**

This option is enabled by default and corrects errors in the DWG or DXF file during the import process.  
For files with large amounts of data, this option may delay the import process. Therefore, you can optionally deactivate this option.

4. **Take page size**

The sheet format is read from the file to be imported and adapted in *CADdy++*. However, if you want to keep the current drawing dimensions, deactivate this option.

5. **Move to origin**

All objects are combined in an imaginary box, which is moved to the coordinate origin.

6. **Adjust view**

The view is adapted to the imported objects so that they are displayed full window.

7. **2D objects**

2D objects are imported into the active drawing.

8. **3D top view**

If the DWG file contains 3D-SAT data, a top view is created during import and placed as a symbol in the active 2D drawing.

### 9. 3D objects

Falls die DWG-Datei Körper im SAT-Format enthält, werden die Körper in das 3D-Fenster importiert.

### 10. Exclude Tab

Use this tab to exclude objects from the import. The import process can be accelerated by activating the relevant option.

#### Frozen layers

For example, layers are frozen to improve performance in CAD software and reduce recovery times for complex drawings.

#### Hidden layers

Objects on these layers are not visible.

#### Invisible elements

These are individual object types that have been hidden.

### 11. Start the DXF/DWG file import with *OK*.

#### Please note:

Symbols are created from dimensions during the import process in *CADdy++*. This ensures that the representation of the dimensions is transferred 1:1 from the original drawing to the model.

Blocks are converted into symbols during the import process in *CADdy++*. After the import, these are only located in the model.


DXF and DWG files can be imported using Drag&Drop. A file imported in this way is generated in a new window of the current model, if the file is dropped in the program window. If the file is dragged into an open drawing, the DXF/DWG file is imported in the drawing.

### Import with the old converter


from *view & plot pro*

In order to load files in DXF or DWG format, select the menu command ***File, Import, DXF/DWG File (old) ....***

1. In the ***source file*** text box, determine the drive, folder and name of the file to be imported.

This can be entered in the text box or be searched for using the  command button.

2. Objects which should not be imported into the *CADdy++* model can be ignored during the import. This can accelerate the import drastically.

3. If objects should be imported, which are not placed in the x-/y-plane activate the option ***Ignore Extrusions***.
4. ***Check & Adjust***  
This option is active by default and corrects errors in the DXF / DWG file during the import.  
If the files to import are very big the option can reduce the import time. Therefore this option can be deactivated.
5. Activate the ***Settings*** option card and activate the default ***accept dimensions as symbols*** check box, if dimensions are to be taken over in the model as symbols and saved in the determined target library.
6. Select from the ***unit*** text box the unit for the import of the DXF/DWG file or define a scale factor. The unit or the scale factor defines the zomm factor for the imported elements. (for architectural drawings very often the unit *meter* is used)
7. Activate the ***convert AutoCAD R12 fonts*** check box if the fonts which are found in the determined AutoCAD font files are to be converted. Only original AutoCAD V12.0 fonts are supported with the exception of so-called Big Fonts.
8. In the ***ACAD fonts folder*** text box, enter the folder which contains the AutoCAD font files.  
  
This can be entered in the text box or be searched for using the  command button.
9. If all the blocks should be disassembled before importing select the option ***Disassemble Block References***.  
This option can accelerate the import drastically, especially with nested blocks.
10. If the file to be imported contains external references to other available files, activate the option ***Disassemble External References***.  
The referenced files must be stored in the original directories or in the same directory then the file to be imported.
11. Under the heading ***Accept symbol***, select one of the following options:  
  
**only accept symbols in model**  
All blocks are taken over in the model as symbols, which are contained in the selected DXF/DWG file. No additional symbol library is set up.

### **accept all symbols**

All blocks which are found in the selected DXF/DWG file are taken over as symbols in the model and defined symbol library.


### **check all symbols**

This checks whether symbols of the same name exist in the target folder of the target library. If this is the case, confirm whether the symbol is to be overwritten in the target folder. New symbols are taken over without a query.


### **Only accept new symbols**

Only symbols for which no symbol of the same name exists in the target library are taken over.

12. In the **target library** text box, determine the drive, folder and name of the symbol library in which the blocks contained in the file and, if necessary, dimensioning as symbols are to be archived.

This can be entered in the text box or be searched for using the  command button.

13. Within the symbol library, determine the folder in which the symbols are to be archived.

This can be entered in the text box or be searched for using the  command button.

14. When you have selected the source file, the fonts used in the DWG/DXF files are given. If you wish to change the font allocation change to the **Font allocation** option card.

To do this, highlight the font which you wish to change, under the CADdy heading.



Click on the icon shown to open the font selection. Select the font and then confirm that it is to be allocated to an AutoCAD font by clicking on the **Allocate** button.

15. Start the DXF/DWG file import with **OK**.

### **Please note:**

DXF and DWG files can also be imported by Drag&Drop.

A DXF or DWG file imported that way is generated in a new window if it is dropped in the program window. If it is dropped in a drawing window it will be added to the elements in this window.

### **Technical information**

The information given immediately below will be taken into account during import procedures from 2D DWG-and 2D DXF files:

### **Object classes dealt with**

ARC	circular arcs
ATTDEF	Attribute definitions
ATTRIB	Attribute text
BLOCK	<b>Symbol</b>
CIRCLE	full circles
DIMENSION	Dimensions
ELLIPSE	Ellipses
HATCH	Group with name "h_Hatchname". All hatching patterns are supported.
IMAGE	<b>BMP, JPG, PCT, PCX, PNG, TGA, TIF</b>
INSERT	Invisible object class
LEADER	Individual geometry element
LINE	lines
LWPOLYLINE	Lines
MLINE	Group with name "Mline."
MTEXT	Text
POINT	Points
POLYLINE	Polylines
PROXY	Symbols
SEQEND	Hidden object class
SOLID	Colour face
SPLINE	<i>CADdy++</i> Spline
TEXT	Text
TRACE	Lines
VERTEX	Invisible object class

### Layer attributes

Name	Layer name
Line type	Layer line type see "Line type allocation for DXF/DWG import"
Colour	Layer colour see "Colour allocation for DXF/DWG import"
Display: On	Active: Yes
Display: Off	Active: No
Freeze: On	Active: No
	Visible: No

Freeze: Off	Active: Yes
	Visible: Yes
Lock: Yes	Active: No
Lock: No	Active: Yes

### Element properties

Line type	Element line type see "Line type allocation for DXF/DWG import"
Colour	Element colour see "Colour allocation for DXF/DWG import"
Width	Element width see "Linewidth allocation for DXF/DWG import"

### Line type allocation for DWG/DXF import

CADDy line type	ACAD line type
————	CONTINUOUS
-----	DASHED
.....	DOT
- . - . -	DASHDOT
— · — · —	DIVIDE
-----	HIDDEN
-----	BORDER
.....	DOT2
— — — —	CENTRE
— · — · —	PHANTOM

### Colour allocation for DWG/DXF import

CADDy++ converts all colours according to the ACI standard.

The DXF/DWG format describes a spectrum of 255 different colours, numbered from 1 to 255.

In order to allocate a colour to a number, use the "AutoCAD Colour Index" (ACI standard). This contains the RGB values of the different colour numbers.

### Special characters

Special characters which are contained in text are retained up to the following which have to be converted specially.

%%d = °

%%c = Ø

%%p = ±



%%% = %

%%o = Text overlined

%%u = Text underlined

### Important Informations:

Partial drawings will be converted without recognizing the individual scale to get an identical appearance in the DXF/DWG format.

The DXF-/DWG format only supports specific digits for BLOCK and LAYER names. Please use only the following digits:

Characters (A-Z)

Numbers (0-9)

(-)

(\_)

(\$)

Please recognize that *CADdy++ will support upper and lower case characters.*

The DXF-/DWG format does only support upper case characters. Make sure your layers and symbols are named only with upper case characters.

In the DXF/DWG format the length for layer and symbol names is limited to 31 digits.

## Import of 2D PC draft files

### *ab view & plot pro*

This converter is not supplied as standard. Enquiries should be directed to DataSolid GmbH.

### Not supported:

Auxiliary infinite lines, line widths, grid data

### Layer attributes

Description	Layer name
Line type	Layer line type see "Line type allocation for DXF/DWG export"
Colour	Layer colour see "Colour allocation for DXF/DWG export"
Active: Yes	Lock: Yes
Active: No	Lock: No
Visible: Yes	Display: Yes
Visible: No	Display: Non

## 3.6 View model

---

A model may contain several 2D drawings and a 3D model that are displayed in separate view windows.

When a model is worked on with *CADdy++*, the procedure is always completed in one of these windows, the current window. You can avoid the need to change windows by closing all of them except one.

When the model is saved, this status will be saved, i.e. when a model is opened, sometimes only one window will be displayed.

You can ascertain which windows go with the model and can be displayed by following this procedure:

- Open the **Window** menu  
At the foot of the menu you will be able to read a description of each of the windows that is open.
- If the line with the description **3D model** is not displayed, you must select the **View, New 3D view window** menu command to open the window.
- Open the command **Model Explorer** in the **View** menu. On the tab **Model** the 3D space and all drawings in the model are listed in the structured view. New windows are opened when clicking on the 3D space or a 2D drawing with the right mouse button and selecting the command **New, View** in the appearing context menu.

## Window commands

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The following sections contain descriptions of the commands for opening windows that are closed, for selecting the current window and for positioning windows that are open.

### Open 2D drawing window

---

If a model contains 2D drawings with windows that were closed when the model was saved, these windows can be opened in the following way:

1. Select the menu command **View, Model-Explorer**.
2. Mark the drawing in the structure to open a new view.
3. On the marked drawing press the right mouse button to open the context menu and choose the command **New, View** or mark the drawing and use the key combination **Shift+Ctrl+N**.

## Open 3D view window

---

If a model contains 3D data for which the window was closed when saving the model, you can open the window by using the **View, New 3D view window** menu command.

Alternatively, activate or open the 3D window via the menu **Window, Open/activate 3D Window** (Alt-W-3).

## Select current window

---

To select one of the opened view windows as the current window, use the following possibilities:

### Full screen display

- First open the **Window** menu and then click the line containing the name of the window that you want.  
After you have done this, windows which have been reduced to a symbol will be restored.
- You may also use the key combination *Ctrl Tab* or the menu command **Window, Next Window** by scanning the opened windows, until the desired window is displayed. The reduced window symbol will not be restored here.

### Reduced display

In this case you may select one of the visible windows as the current window; by clicking on this window.

### Symbol display



You can use either of the system icons to restore or to maximise.

## Modify window size

---

To modify the size of the reduced window which is now restored in the main window, proceed as follows:

1. Move the cursor onto the frame of the window. The cursor will now be displayed as a double arrow.
2. Move the cursor – while keeping the left mouse button pressed – in the direction of one of the arrows.
3. When you have the desired window size, release the mouse button.

## Modify position of window

---

To change the position of the reduced window which is now restored in the main window, proceed as following:

1. Place the cursor on the header of the window.
2. Move the cursor, together with the window while pressing the left mouse button.
3. When you have the desired position, release the mouse button.

## Tile all windows

---

To display all opened view windows of a model, select the menu command **Window, Tile** or the menu command **Window, Cascade**.

## Drawings and partial drawings

---

If the model you are working on is to refer to several **2D Drawings** rather than just one, you can do so by using the **Model Explorer** to create them.

Several **view windows** can be opened for each drawing. A different view can be displayed in each window, i.e. a different view of the drawing. You can also change from one window to another while you are processing the drawing.

Any number of partial drawings can be worked on when you are dealing with a drawing.

These are work sections with a name and a scale that can be allocated authorised user access and display attributes of their own.

The advantage of partial drawings is that you will be able to create a standard sheet, for example, that will contain a text box, separate sections of an assembly drawing or detailed drawings to different scales, each separated from the others for easier processing.

## Use a number of drawing

---

In order to select one of the opened 2D drawing windows as the active window, activate the required area

- using the **Model Explorer**,
- use the list at the end of the **Window** menu
- or change windows by pressing **Ctrl Tab**.

## Process partial drawing attributes

To process the attributes relating to a partial drawing, select in the **Model Explorer** the context menu command **New, Settings 2D drawing ...** on a partial drawing

The following editing options for partial drawings are available in the dialog box:

### Settings for displaying partial drawings

If the model contains several partial drawings, it may be useful to be able to distinguish the objects of the currently selected partial drawing from the objects that are in the currently inactive partial drawings.

For this purpose it is possible to display the objects of the non-active partial drawings in different colors - depending on the access rights. Objects of the current partial drawing are always displayed in the colors assigned to them.

You can switch the different display on and off using the **Show / Hide status of partial drawing** button.



These two icons are for specifying the colours indicating access rights to all the partial drawings that are not active.



**display status** (grey symbol)


This display mode must be active to display in accordance with the access rights that have been specified.



**suppress status** (coloured symbol)

The objects in all the partial drawings will be displayed in accordance with the attributes that are specified in the layers or with reference to the objects themselves.

### **status**

This can be used to change the status, i.e. the access rights to the objects in a partial drawing that is not active and is tagged with the  symbol. If the symbol is yellow it shows that the partial drawing is active, otherwise its grey.

The display will be in accordance with the status defined for the access rights, if the display mode **Display Status** is active.



**read only**

Objects in the partial drawing can be identified with the cursor to define points.

**without access**


Objects in the partial drawing cannot be identified with the cursor.


**do not display**

Objects in the partial drawing will not be displayed.

### **Change Status in the Model Explorer**

The status of partial drawings (*visible / invisible* or *active / inactive*) can also be changed in the context menu of the **Model Explorer**.

If a partial drawing or a workplane is set to invisible also all the objects inside are invisible. In a workplane also the frame of the workplane is invisible. Partial drawings and workplanes with the status invisible and with that implicitly with the status inactive are marked with the symbol .

When deactivating a partial drawing or workplane, the contained objects are not pickable. This status is marked with the symbol .

The status of a partial drawing or a workplane can be changed in the **Model Explorer** as follows:

1. Select the command **Model Explorer** in the menu **View**.
2. Tag the partial drawing or workplane to be changed.
3. Start the context menu by pressing the right mouse button placed on the partial drawing or workplane and select the desired option in the sub menu **Visibility / Pickability**.


For this, also the listed key combinations can be used.

in the structure tree	in the detailed view	key command
Toggle visibility	Toggle visibility	O
Toggle pickability	Toggle pickability	P
Toggle visibility and apply to subtree		Strg+Alt+O
Toggle pickability and apply to subtree		Strg+Alt+P
Enable visibility of tree	Enable visibility	Strg+O
Disable visibility of tree	Disable visibility	Shift+Strg+O
Enable pickability of tree	Enable pickability	Strg+P
Disable pickability of tree	Inaktiv	Shift+Strg+P

## Activate partial drawing

A partial drawing can be activated in a number of different ways:

### Select in the 2D symbol bar

Open the list  to the very left of the 2D toolbar and select the required partial drawing.


The scale of the active partial drawing is displayed in the header of the 2D drawing window or – if in full screen display – in the header of the main window.


### Select via object



Click the icon in the symbol bar and then identify an object from those in the partial drawing.

### Selection using the *Model Explorer*

For the setting  **Activate selected components** in the *Model Explorer*, a partial drawing automatically becomes the active partial drawing (and also the target area for new objects) if this is identified in the tree structure.

For the settings  **Do not activate selected components** you have the following options available to activate a partial drawing:

Tag the required partial drawing and press the key(s)

- **F12** or select the option **Declare to current 2D/3D context** from the context menu if the partial drawing is also to be activated as the target area for new objects.

or

- **Shift+F12** (context menu: **Declare to current work area**) if you only wish to activate the partial drawing. A previously activated group within this partial drawing however remains as the active target area for new objects.

## Control model display

In the lower section of the **View** menu you can activate the commands for controlling the view inside the current window.

Some of the commands are also available in the symbol bar.

### Enlarge a window display



This command can also be loaded using this icon in the top toolbar.

In order to enlarge the display of objects to fill the entire size of a rectangular section (window), follow this procedure:

1. Select the menu command **View, Zoom Window**.  
The cursor contains a magnifying glass as an additional symbol.
2. Using the cursor determine the first corner point of the window which is to contain the objects to be enlarged.
3. Determine the diagonally opposite corner point of the window.  
The objects which lie within the window are enlarged so that they fill the current window.
4. Redefine a window or abort using the right mouse button.

## Vary View (w)

---



After clicking on the displayed icon or using the hotkey **w** (or **W**) the command **Vary view** is started.

This command can be used to **Move view** and/or **increase / decrease** and also to **rotate** if the 3D window is active. The condition to move a window is, that the screen window must be decreased or increased before, so that the screen window can also be moved by the scroll bars.

By double clicking in the 2D or 3D window the view can be centred. Double clicking in a 3D window also defines the viewpoint, which is used as centre point for rotation. The centre of gravity is used as centre point for rotation when double clicking on a surface. Double clicking on an edge uses the nearest endpoint, midpoint, centre or quadrant point as centre point for rotation.



### Pan view

Start the command and click into the active view. Keep the mouse button pressed and move the view with the mouse. The view can be moved until it reaches the border of the screen or until the scroll bars are in their end positions.



### Decrease / Increase view

To decrease / increase the view in the active window choose the following:

- In case the mouse is equipped with a wheel this can be used to decrease / increase the view.



- Alternatively the view can be decreased / increased by using the **CTRL-key** and moving the mouse pointer up and down the screen.
- Also the view can be decreased / increased by clicking the middle mouse button until the icon appears and then move the mouse pointer up and down the screen.



### **Rotate view (only in the 3D window)**

To rotate the view in the 3D window choose one of the following:

- Use the **Shift-key** and press the left mouse button.
- Also the view can be rotated by clicking the middle mouse button until the icon appears and then press the left mouse button.



### **Rotate View around Z Axis (only in the 3D window)**

To rotate the view around the Z axis of the current work plane please follow one of this steps:

- After starting the command keep pressed the left mouse button and click the key combination **Ctrl-Shift** and move the cursor left or right.
- After starting the command click the middle mouse button as often as the displayed icon is attached to the cursor icon. Keep the left mouse button pressed and move the cursor to the left or right to rotate the view around the Z axis of the current work plane.

## **Enlarge display**

In order to display the content of the current window in an enlarged form, follow this procedure:

1. Select the menu command **View, Zoom +**.  
A magnifying glass symbol is added to the cursor.
2. Using the cursor, determine the fixpoint.  
This point is moved to the centre of the window. All objects are displayed enlarged by a factor of 1.2.
3. Redefine a fixpoint for further enlargements or abort using the right mouse button.

### **Please note:**

The command can also be loaded by pressing the **+** key.

## Reduce display

---

In order to display the content of the current window in a reduced form, follow this procedure:

1. Select the menu command **View, Zoom -**.  
A magnifying glass symbol is added to the cursor.
2. Using the cursor, determine the fixpoint.  
This point is moved to the centre of the window. All objects are displayed reduced by a factor of 1.2.
3. Redefine a fixpoint for further reductions or abort using the right mouse button.

### Please note:

The command can also be loaded by pressing the - key

## Centre view

---

In order to move the display of objects to the centre of the current window, follow this procedure:

1. Select the menu command **View, Centre View**.
2. Using the cursor, define a point which is to be moved to the centre of the current window.
3. Redefine a point or abort using the right mouse button.

### Please note:

The command can also be loaded by pressing the \* key.

## Display all objects

---



This command can also be loaded using this icon in the top toolbar.

With the help of the **View, Show all** menu command you can find and display all objects, i.e. even those that are outside the defined drawing boundaries.

Select the menu command **View, Show All**.

## Display original

---



This command can also be loaded using this icon in the top toolbar.

The 2D drawing can be displayed to fill the size using the menu command **View, Show Original**.

This is for loading the menu command **View, Original Workplane** inside the 3D view window. In this way you will be able to display the current workplane in a top view in the size you have specified.

## Display last view

---



This command can also be loaded using this icon in the top toolbar.

For each 2D drawing window and each 3D view window, the last selected view status is always saved.

This command can be used to update the previously defined view status step-by-step.

## Display next view

---



This command can also be loaded using this icon in the top toolbar.

For each 2D drawing window and each 3D view window, the last selected view status is always saved.

If the command **Last View** has previously been used, this command can be used to update the next view status step-by-step.

## Redraw

---



This command can also be loaded using this icon in the top toolbar.

If the drawing in the current window contains inaccuracies due to processing or erase commands, the drawing can be regenerated using the menu command **View, Redraw**.

Current settings such as selected windows, perspectives and display attributes of objects are not modified by doing so.

### Please note:

The command can also be loaded by pressing **F5**.

## Start Hide Action

---



The **Start Hide Action** command is started via the button shown above. It is part of the 3D selection mode and serves to temporarily hide solids in the 3D area.

After you have started the command, identify the solids that you would like to hide successively. The status invisible remains also when leaving the command but is not stored with the model or in the UNDO list.

If solids have been hidden this way or by one of the other options described above in the 3D selection mode, this is made apparent by a modified display of the command button **Start Hide Action**:



No solids are hidden.



Solids have been hidden.

Pressing the command button shows all of the previously hidden solids once again.

Hold the **Shift key** and press the button if you would like to retrospectively hide further solids.

**Please note:**

The options in 3D selection mode, including the command **Start Hide Action**, serve to temporarily hide interfering solids in 3D commands. Use the **Groups** options in the **Model Explorer** to switch solids, 3D parts or 3D components to invisible; also permanently, even after a model has been saved and closed.

## Start Hide Others Action

---



The solid identified using the command **Start Hide Others Action** remains visible. All other solids in the current view are faded out. The status *invisible* remains valid when leaving the command. The solids are faded in when clicking the command on the right side.

If solids are faded out, the icon **Start Hide Others Action** is changed:



No solids are faded out.



Solids are faded out.

By clicking the lower icon all solids can be faded in again.

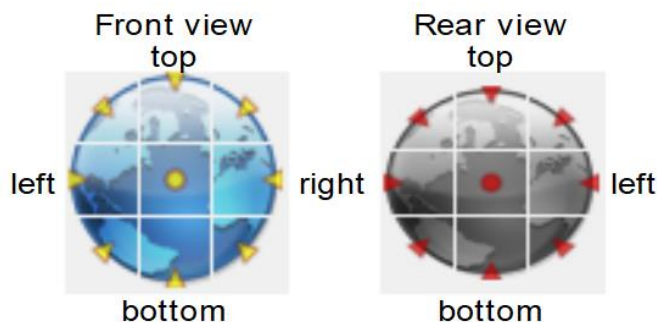
## Select 3D standard view

---

In the 3D view window, one of 16 standard views can be selected directly.

First use the menu command **View, Camera** to open the toolbox that can then be used to select these views. Position this toolbox at any position on the screen.

The toolbox contains the following symbols for the selection of views:



These icons can be used to select the main view directions named in the illustration and also to select the combined view directions such as from front, left and top.

For views, in which the important information is in the front view, a dimetric projection can be selected:



Dimetric left



Dimetric right

#### Please note:

The centre for each view is the focal point. This is positioned in the middle of the view window.

When defining the focal point using the command **Select Special 3D View**, ensure that the previously selected section of the model remains in the area of the view window when changing the view.

## Navigate View using Camera Cube

The camera cube is a 3D navigation tool that allows you to switch between standard and isometric views.

Select the menu command **View, Camera Cube** to open the dialogue window.

By clicking on the cube, you rotate to one of the predefined views. A click on the home button switches to the home screen. Use the arrow symbols to rotate the view. You can also open the selection dialog for dimetric views.

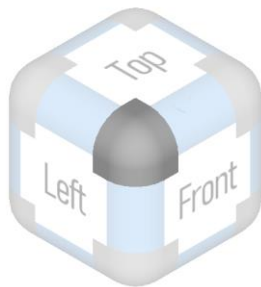
The settings for the animated view can be defined using the menu command **Settings, Options** in the tab **User Interface** in the section **Animated view in 3D window**. The settings also affects the command



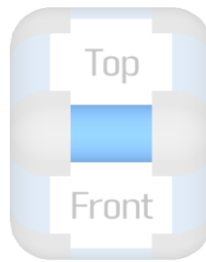
**View on Face / WP.**

- Activate / Deactivate animation
- Number of steps to define, how many steps in the defined time interval are made until the end view has been reached. Enter a value between 1 and 100.

- Time interval per step (ms) defines the interval between two steps. Enter a value between 1 and 1000 (ms).



**Corner**



**Edge**



**Face**

Click on one of the highlighted areas to switch to the default view.  
Another click on the same area rotates the view 180° to the rear area.



The home button switches to the standard perspective *front left top* (F6).



Click on one of the arrow buttons to rotate the view by 90°. If you hold down the Ctrl key at the same time, the view is rotated by 45°.



Opens the dialog ***Dimetric Views***.

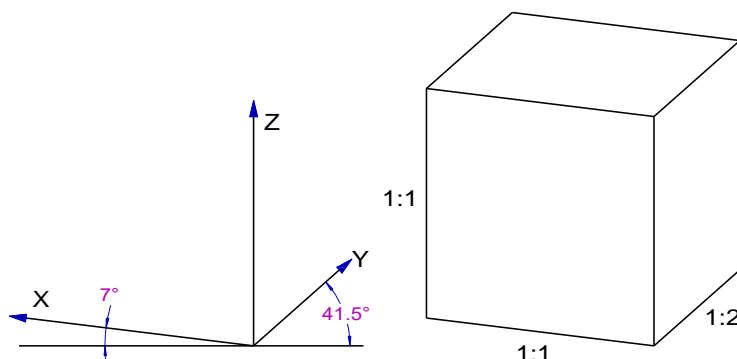
## Dimetric Views

---

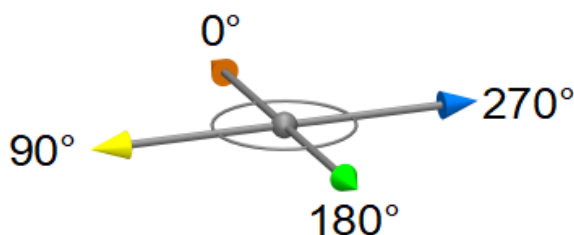
To display the model in a dimetric view in the 3D window, select the menu command **View, Dimetric Views**.

In the dimetric view, one edge of the component is drawn at an angle of 7°, the other at an angle of 42° to the horizontal.

Standardised dimetry:



The predefined dimetric views rotate the view of the model in the direction of the angle relative to the global 3D coordinate system with the angle aligned to the right or left.



Example for the view **Dimetric 0° left**.

#### Please note:

See the chapter Custom Views in the 3D Window for information on how to save additional dimetric or arbitrary views.

For the representation of the view in *CADdy++* the mathematically exact definition of the dimetry is used. The widths and heights of a box with the edge length 10 are shortened to  $\frac{2}{3}\sqrt{2} \approx 9,428$  and the depths are shortened to  $\frac{1}{3}\sqrt{2} \approx 4,714$ . The lines of the front view are displayed at an angle to the horizontal of  $\arcsin\frac{1}{8} \approx 7,18^\circ$  and the side view of  $\arcsin\frac{\sqrt{7}}{4} \approx 41,4096^\circ$ .

### Display workplane in top view



This command can only be loaded in the 3D window by using the top toolbar or with the *View, Workplane Original* command.

When loading the menu command **View, Workplane Original** the top view of the current workplane in the original is displayed inside the 3D view window. The display will be made according to the specified dimensions.

### Select special 3D view



This command can be activated using this icon in the top toolbar.

This command can be used to select a special view for the 3D view window, by dynamically rotating and/or zooming the model 3D space including all objects.

#### Dynamic rotate with defined focal point

This command can be used to dynamically rotate the view about a defined point: **the focal point**.

The rotation axis for a dynamic rotation runs, as long as it has not been otherwise defined using the focal point, simply through a point in the screen plane. Its position in the model space is more or less determined by a calculation procedure.

Particularly when working in an enlarged display – this can lead to objects that you wish to view from another direction being rotated out of the visible area of the view window.

For this reason, in the **Dynamic Rotate** command, you should define a focal point on an existing solid about which you wish to rotate the view.

Proceed as follows:

1. Define the focal point on a surface by identifying a solid or defining a point in the current work plane.  
The model 3D space is moved so that the define point lies in the centre of the 3D view window and the rotation axes run through this spatial point.  
  
If you do not wish to define any focal point, abort focal point definition using the right mouse button and then proceed as follows.
2. Using the cursor, define a point as near to the centre of the screen as possible.
3. Starting at this point, if the cursor is moved to the right or left, the model 3D space is moved in the same direction about the vertical screen axis which lay in the middle of the 3D view window when the command was activated for the first time.  
If the cursor is moved upwards or downwards, the rotation takes place in the same direction about the horizontal screen axis.
4. Determine the selected view, by confirming with the left mouse button.  
The right mouse button can be used to abort. The previous view is then redisplayed.
5. If the view does not fulfill your requirements, follow the procedure from step 1 again.

### **Dynamic rotation around the Z axis**

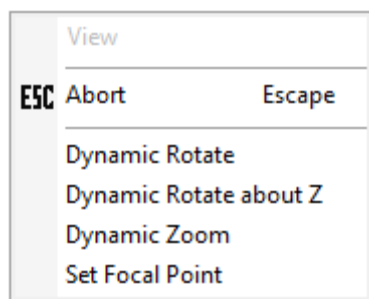
This command allows the rotation of the view around a defined focal point with a rigid Z axis of the current work plane.

Please follow this steps:

1. Activate the desired Work plane, to define the rigid Z axis.



2. Start the command **Dynamic Rotate with Focal Point** by clicking the displayed icon.
3. Define the focal point by clicking a surface (=rotation point of the view).  
The 3D space is moved in the way, that the defined point is in the centre of the 3D space.  
  
If no focal point should be defined, cancel the focal point definition by clicking the right mouse button and continue as described below.
4. Press the middle mouse button to start the context menu:

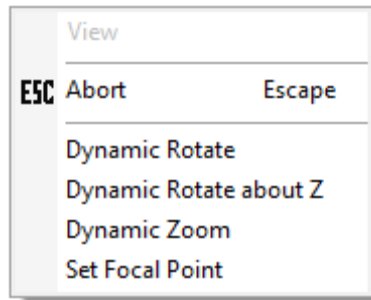


5. Select the command **Dynamic Rotate about Z**.
6. Define a point by cursor in the centre of the screen.
7. If the cursor is moved to the left or the right, the 3D space is rotated around the Z axis of the current work plane.
8. Set the desired view by clicking the left mouse button.  
Using the right mouse button instead, the command is cancelled and the previous view will be displayed again.
9. If the view is not correct, you can proceed with step 4 again and maybe select another option from the context menu.

### **Dynamic zoom**

1. Define the focal point on a surface by identifying a solid.  
The model 3D space is moved so that the define point lies in the centre of the 3D view window and the rotation axes run through this spatial point.  
  
If you do not wish to define any focal point, abort focal point definition using the right mouse button and then proceed as follows.

2. After starting the command, press the middle mouse button. This loads the following context menu:



3. Select the command **Zoom Dynamically**.
4. Using the cursor, determine a point in the screen plane. This is the "handle" on which the model 3D space is "held and moved".
5. "Move the model to the back" by moving the cursor upwards starting from the reference point or "drag forwards" by moving the cursor downwards.
6. Determine the selected view by confirming the left mouse button. The right mouse button can be used to abort the command. The previous view is redisplayed.
7. If the view does not fulfill your requirements, follow the procedure from step 1 again.

**Please note:**

**The following is valid for rotation:**

If a focal point has been determined, this is kept as the reference point for the rotation axes. If a section of a model has been selected which is not close to this focal point, a new focal point should be determined.

The further away the "handle" from the middle of the 3D view window, the larger the possible rotation angle.

**The following is valid for zooming:**

The nearer the "handle" to the lower edge of the 3D view window, the smaller the model 3D space can be displayed.

The nearer the "handle" to the top of the 3D view window, the larger the model 3D space can be displayed.

---

## Rotate view on solid face / workplane



Start the the **View, View On Face/WP** command.

This command can be used to change between a solid face or workplane in the top view.

Settings for this command can be made using the menu command **Settings, Options** on the **User Interface** option card:

- Activate /deactivate animation
- Number of steps defines how many steps are to be traced back in the animation during the set time interval, until the view is reached. Enter a value between 1 and 100 and confirm the entry with ENTER or use the arrow command buttons next to the entry.
- Time interval per step (ms) defines the time interval between 2 steps. Enter a value between 1 and 1000 (ms) and confirm the entry with ENTER or use the arrow command buttons next to the entry.

After you have started the command, identify a solid face, a 3D axis or the frame of a workplane. The 3D view is rotated in the top view of the selected solid face, a 3D axis or WP.

## 3.7 Information

from **view & plot pro**

The commands in the toolbox **Information** can be used to display the dimensions of 2D and 3D objects.

### Information - point-point distance

from **view & plot pro**



This command can also be activated using this icon.

This command determines the distance between two points and displays the value in the Status dialog box.

1. Define both points.  
The distance will be indicated temporarily by a line.

### Information - point-line distance

from **view & plot pro**



This command can also be activated using this icon.

This command determines the minimum distance between a point and a 2D line object (line, circle, ellipse, spline) and displays the value in the Status dialog box.

1. Define the point.

2. Identify the required 2D line object.  
The distance will be indicated temporarily by a line.

## Information line-line distance

---

from *view & plot pro*



This command can also be activated using this icon.

This command determines the minimum distance between two 2D line objects (line, circle, ellipse, spline) and displays the value in the Status dialog box.

1. Identify two line objects.  
The distance will be indicated temporarily by a line.

## Information - Determine Distance

---

from *view & plot pro*



After clicking on these icons, the command **Determine Distance** is started.

The command determines the distance between objects and / or points and displays the value in the Status dialog box.

1. Identify a 2D line object or define a point.
2. Identify another 2D line object or define another point.  
The distance will be displayed in the status dialogue window as value and in the graphic as line.

### Please note:

If you determine the distance between 2 points in 2D, the **dX**- and **dY** Coordinates are displayed as well, in 3D also the **dZ** Coordinate. In 3D the coordinates are related to the global Coordinate system.

When determine the distance in 3D, the help line which is created when pressing the **SHIFT key** will only be created, when both identified points are on the current work plane.

When identifying two 2D line objects (Line, Circle, Ellipse, Spline) always the minimum distance will be determined.

## Information - angle

---

from *view & plot pro*



This command can also be activated using this icon.


This command determines the angle between three points and displays the value in the Status dialog box.

1. Define the summit of the angle.
2. Define the endpoint of the first side of the angle.
3. Define the endpoint of the second side of the angle.  
The angle is determined in an anticlockwise direction from the first to the second side.

## Calculate 2D faces

from **view & plot pro**

The procedure for calculating the geometric values associated with 2D faces is described below:

1.  If the selection list is empty, activate the command by clicking the icons **Information, 2D Face Calculation**.
2. Identify an individual 2D face, or select several at once by holding down Ctrl at the same time.
3. The values for the 2D faces selected above will be presented in the status dialog box.

## Information continuous line length

from **view & plot pro**



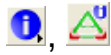
This command can also be activated using this icon.

This command determines the length of a line consisting of line objects (line, circle, ellipse, spline) and displays the result in the Status dialog box.

1. If the objects forming the continuous line are not in the selection list, take them over into the action list.
2. Define the starting point and end point on this continuous line for computing the length.  
The value that is determined will be displayed in the Status dialog box.

## Information about Distance and Angle of Faces

---



By clicking the displayed icons in the 3D window the command **Information, Distance / Angle between Faces** is started.

The minimum distance between faces is determined and the angle between Surface normals and is displayed in the status block dialogue window.

1. Start the command by clicking the displayed icons.
2. Identify the first surface.
3. Identify the second surface.

The calculated values are displayed in the status block dialogue window.

### **Please note:**

The angle can only be determined between planar surfaces.

## Solid Calculations

---

from **View&Plot pro**



In order to be able to display solid data, use this command in the toolbar.

1. Start the command **Information, Solid Calculations**.
2. After starting the command identify a single solid  
or  
collect multiple solids with the pressed **Ctrl** key  
or  
select the solids by dragging a window.

A dialog window will display the calculated values for the selected solids.

While the dialog is open, further solids can be integrated or excluded to the calculation by identifying them.

### **Solid calculations dialog box**

This dialog box displays the values determined for the selected solid.

The following options are also available:

### **Calculate weight**

In order to calculate the weight of the solid, enter the specific

weight of the solid in the **density** text box and confirm the value with *Enter*.

### Save values in a file



Click on this command button.

In the dialog box which then appears, determine the directory and name of the file. It will receive the extension GEO unless a different one is defined.

### Option card - General

#### Display solid centre of gravity



Click on this icon, in order to identify the centre of gravity of the solid with a marker.

Once the drawing is redrawn, the identification marker is removed.

#### Regard Density

If a material resp. a density is assigned to the solids using the menu command **Process, Object Display** these are taking in account when calculating the mass or the moments. For all solids without a material a density of 1 is assumed.

#### Calculate weight

If a material resp. a density is assigned to the solids using the menu command **Process, Object Display** these are taking in account when calculating the mass or the moments. For all solids without a material a density of 1 is assumed.

If no material was assigned before, a material can be selected from the list or a density can be entered. The properties, assigned in this dialog window are not stored with the model and are lost when leaving the command.

### Save values in a file



Click on this icon.

In the dialog box that then opens, enter the directory and name of the file. It has the extension GEO – as long as no other is defined.

### Option card - Moments

#### Enter vector



Click on this command button in order to generate the vectors as solid axes.

### Option card - Inertia

The inertia tensor is a matrix display of the mass distribution of a rigid solid referenced to a certain coordinate system. Its diagonal

elements  $I_{ii}$  are called **moment of inertia**, the non-diagonal element  $I_{ij}$  **deviation moments**.

By a main axis transformation, the inertia tensor can always be brought onto the diagonal shape. The axes **(1,2,3)** of this system are called **principle inertia axes** and run through the centre of gravity of the solid. The diagonal elements correspond to the **principle moments of inertia**  $I_1$ ,  $I_2$ ,  $I_3$ .

With a rotation about the principle inertia axes, the rotation impulse and the angle speed are parallel.

The inertia tensor is calculated based on the global coordinate system. To calculate the inertia tensor based on a different coordinate system the solid has to be transformed in a way that the relative coordinate system and the global coordinate system are identical.

---

## Calculate a Surface Area

from **View&Plot pro**



After clicking the displayed icons the command **Information, Calculate Surface Area** is started.

The command calculates the area of a surface and displays the calculated value in the status block window.

---

## 3.8 Print model

Whenever you execute the menu command **File, Print**, the contents of the current window will be printed.

There are two different dialogs and the one you see will depend on whether you are dealing with a 2D drawing window or a 3D view window. In either case the dialog is for specifying the settings.

In addition, with the 2D drawing window you have the option of copying the 2D objects visible in the 2D drawing window to your clipboard.

---

### Printer setup

If a different printer to the Windows standard printer is to be used for output or the settings and attributes are to be modified, select the menu command **File, Print**.

1. Using the **Setup** command button, load the Windows dialog box for printer setup.



2. Carry out the necessary modifications and confirm with **OK**.

## Print 2D drawing

---

In order to print the currently selected 2D drawing, select the menu command **File, Print**.

1. If necessary, using **Printer setup** ..., determine another printer to the Windows standard printer or modify the settings.
2. Determine the required settings for the output of the drawing (see below). Check the size and position in the preview window.
3. Start the print procedure with **Print**.  
If the setting **Output to file** is active, type the name of the file in the dialog box. If a drive and folder are not included, the file will be saved in the folder .USER.

### Determine print settings for a 2D drawing

#### **current printer**

The name of the Windows standard printer is displayed in this text box.

If the drawing is to be output on another printer, select the **Setup** command button in order to select another printer.

#### **max. print area**

Shows the dimensions (in mm) of the maximum area, which can be used by your plotter/printer relating to the determined paper size.

This value is taken over from the Windows printer settings and cannot be entered.

When the drawing is to be printed out to scale, you must not create the drawing itself or the standard sheet for it, until the drawing sites that are specified (x,y) are not greater than the maximum print area that your printer can cope with.

#### **number of copies**

Determines the number of print outs.

#### **output to file**

If this setting is active, print output can be sent to a file.

#### **Print scale**

After you have deactivated the setting **fit into print area**, you now have the following setting options if the drawing is not to be printed to the print area but rather to scale.

- Select a desired scale from the list of predefined print scales.

- Enter into the **print scale** input field: A value in **percent** for the scale. An example of an input if the drawing is to be printed in a reduced scale of 1:2 is:  
50% (confirm input with the Enter key)
- Enter into the **print scale** input field: (in the percent display area) a **factor** for the scale. An example of an input if the drawing is to be printed in a reduced scale of 1:10 is:  
0.1 (confirm input with the Enter key)
- Enter into the **print scale** input field: (in the percent display area) a **scale** to which it should be scaled. An example of an input if the drawing is to be printed in a reduced scale of 1:25 is:  
1:25 (confirm input with the Enter key)

### **scale line width**

On screen the 2D linear objects will be displayed in the line width that was allocated with reference to layers or objects, if **File, Options** was used to activate the option **Display thick lines**. If this option is not active, they will be displayed with a width of one pixel.

Independent of the display on the screen, line widths are always output with their defined values when printing.

If the option is not active, the line widths are output as they are displayed.

If the option is active, the factor for the print scale also has an effect on the line widths.

### **print black/white**

If this setting is active, all visible 2D objects are printed with the colour black.

If the setting is not active, the visible 2D objects are output according to the printer settings or possibilities, in colour or grey levels.

### **print text with clip face**

Activate this setting if you are using a **pen plotter**. By doing this your clipped texts and scale texts will be printed, but will be without the clipped area that would otherwise be plotted as a filled area.

Deactivate this setting, however, if you are working with a grid device: this is the only way to take care of the on-screen clipping of the texts even when they are to be printed.

### **current boundaries**

Activate this option if all objects which are within the set drawing borders of the 2D drawing, are to be plotted.

**current view**

Activate this option in order to plot the currently displayed screen content.


**geometry box**

All visible elements in the drawing will be printed, including the ones which maybe are outside of the drawing boundaries.

**User Defined Boundaries**

To print a section of the drawing activate this option. All objects outside the defined section are not printed.

**Boundaries by values:** A rectangular section can be defined by entering the values into the input fields for X and Y coordinates: (x1,y1) and (x2,y2). The values must be separated by a comma. Please consider that the values are adjusted to the paper size and not to the drawing coordinates if a scale different from 1:1 is used.

**Boundaries by 2 points in the drawing:** After clicking the  icon the dialogue window is minimized and the rectangle can be defined by two points. After defining the second point the dialogue window is opened again. The defined coordinates of the section are shown in the input field and can be edited there.

**Orientation**

In this area the orientation for the print is defined. You can select between

Upper - Left	Upper - Middle	Upper - Right
Middle - Left	Middle - Middle	Middle - Right
Lower - Left	Upper - Middle	Upper - Right

Additionally the print can be oriented to the printable area or the paper edge and can be moved horizontally or vertically.

**Rotation**

Select, if the drawing should be rotated for the print. This can be defined in 90° steps.

**Printable area**

The actual dimensions for outputting the drawing are displayed for checking purposes, i.e. a window is calculated which contains all drawing objects, even objects which are possible outside the defined drawing borders.

If the dimensions of this section are too large, a warning message is displayed during the print process.

### **Portrait**

Select this option to print a drawing in portrait format.

### **Landscape**

Select this option to print the drawing in landscape format.

### **Preview**

The black rectangle identifies the paper size.

The green rectangle identifies the **max. print area**.

The blue rectangle identifies the **defined size**.

The red rectangle identifies the position of the **drawing**. If the respective command button is pressed, a preview of the drawing to be printed is displayed in addition to the dimensions of the area.

## **Print 3D model**

---



Use the following procedure to print the 3D model. First select the 3D window as the current window and then continue:

1. If necessary, use **Setup** to specify a printer other than the Windows standard printer, or change the settings.
2. Specify the defaults for the output (as described below).
3. Trigger the printing by clicking **Print**.  
If the setting **Output to file** is active, type the name of the file in the dialog box that is presented. If you do not specify the drive or the folder, the file will be saved in the folder called **..\USER**.

All the objects that are visible will be printed. The dimensions of the section to be printed are the same as those of the monitor area that is displayed after you have loaded **View, Display All Objects**.

### **Settings for printing a 3D model**

#### **current printer**

This box shows the description of the Windows standard printer.

Should you want to specify a different printer, click the **Setup** button and then select it.

#### **maximum print area**

This shows the dimensions (in mm) of the largest possible area that your printer can use with reference to the size of the paper that you have indicated in the settings.

This entry will be taken over from the Windows printer controls and cannot be edited.

**no. of copies**

This entry will determine how many copies are printed out.

**output to file**

If this setting is active, you can write the print-out to a file.

**scale line width**

If this setting is not active, lines will be drawn in the widths specified for layers or for objects.

If the setting is active, however, the line widths will be in the same scale as the drawing that has been adapted to the **defined size** for the print area.

**print black/white**

If this setting is active, all the visible objects will be printed in black and white.

If the setting is not active, however, the visible objects will be printed in the colours specified or in shades of grey, depending on the printer settings.


**View Selection****Current focal point**

With the option **Current focal point** the actual view is printed in the maximum print area of the selected paper format.

**Current window section**

With the option **Current window section** only the content of the actual window will be printed.

**Picture**

With the option **Picture** everything will be printed, what will be shown after activating the command  **Show All Objects**.

**defined size**

The entry you make here will determine the size of the area to fit in the output of your 3D model. The values must not be greater than those entered in the box indicating **maximum print area**.

**displacement**

The values entered here will result in a move of the origin of the drawing on the paper.

The total of the values for the **defined size** and the **displacement** horizontally and vertically must not be greater than the maximum print area size.

### hardware offset

As a rule a printer cannot print on the entire sheet.

Type a distance from the lower left corner of the sheet as the starting point or the origin for the printer.

The entry to be made is listed in your printer manual.

The standard values will be taken over from the Windows printer control program.

### Print quality

After you have opened the dialog box, **pixels per inch** will show you the current maximum resolution of your printer in a horizontal and vertical direction.

You can enter lower values in the two boxes to reduce the **estimated memory requirement** for completing the calculations.

For Pre Prints in the options **Current focal point** and **Picture** the quality of the output can be defined. The scale is from **1** for **very fine** to **10** for **very rough**.

## Print 3D solids

---



After clicking the menu commands **File, Print...** with a 3D window active a dialogue window appears to print from the 3D solids, start the print process and save the printer settings into the model.

Below please find the commands for the preview



### Move View

Click in the preview window with the middle mouse button pressed.  
Move the mouse to move the view.



### Zoom In / Zoom Out

Zooming can be done with the scroll button or by pressing the right mouse button and moving the mouse up and down.

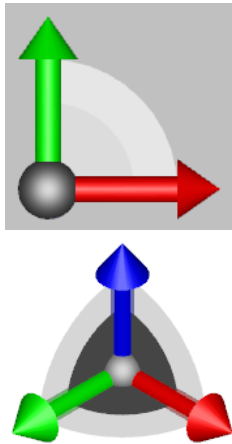



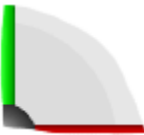



### Rotate View

Click in the preview window with the left mouse button pressed.  
Move the mouse to rotate the view.

### Display the interactive coordinate system

If this option is active, the view will rotate by a click on the following position....

		... in view direction of the arrow
		... by 90° (arrow shaft)
		.. displays all objects
		... in the perspective view (inner circle)
		... in a view from the front or from behind (outer circle)

## Define Settings for the 3D Print



After clicking the menu commands **File, Print...** with a 3D window active a completely new dialogue window appears to print from the 3D window, start the print process and save the printer settings into the model.

### Current Printer

In this field the current printer is displayed.

In case you wish to print the 3D view to a different printer, you can make select a printer from this list or via the button **Printer Setup**.

### Number of Copies

Defines the number of copies.

### Use Driver Optimization

If supported from the driver, the program will try to create one print job, which will be printed as often as given in *number of copies*. If the driver doesn't support a copy optimization, the print job will be send as often as defined in *number of copies*.

### DPI (dots per inch)

Input field to define the resolution of the points for the output. The higher the value, the better the quality of the output.

### Portrait / Landscape

Select the desired option to print in portrait or landscape format.

### **Pixel Size**

Displays the resulting size of the output in pixels according to the resolution DPI and the current paper size.

### **Maximum Print Area**

Displays the size (in mm) of the maximum print area of the current printer corresponding to the selected paper size.

This value is taken from the Windows Print Control and cannot be edited.

### **Offset**

Entry field to define the offset of the print area starting from the left and upper paper frame in mm. Positive and negative values are allowed here. Positive values will move the print area to the right or downwards. When choosing other units than mm, the unit must be entered after every value and confirmed by pressing the **Enter** key: 1cm x 0,02m

### **Paper Size**

Selection list of the supported paper sizes of the current printer.

### **File Size**

The file size is calculated from the resolution DPI and the selected paper size. The file will temporarily be stored in the TEMP directory, before transmitting to the printer.



By clicking the displayed icon the actual view from the 3D window is taken over into the preview window.

Using an orthogonal view in the preview the view angle and the zoom is adapted, in a perspective view due to the different view modes only the view angle can be adapted.

### **Perspective View**

When activated, the current view is printed in a perspective view instead of an orthogonal view.

### **Display Coordinate System**

The interactive coordinate system allows to modify the view in the preview window. It will also appear in the output, when activated.

### **Antialiasing**

This option is dependent on the computer hardware, especially from the graphic board and the installed drivers. With the antialiasing option the lines can be smoother. The higher the value the more the lines are smoothed.

### **3D Gratic**

Allows the print of several 3D file formats, without loading the file



before. After clicking this button, you can select a 3D object you wish to print.

### 3D Model

The view of a 3D graphic is closed and the current 3D model is displayed again.

### Activate Color Gradient

Allows to place a gradient in the background for printing. The color will change from the first selected color to the second selected color. When this option is deactivated the background is printed in the given color.

### Stamps

In the **Stamp** area, you define additional information for the printout, such as the name of the model or the current date/time. By activating the **headline** or **footline**, the information selected in the adjacent list is also printed out.

You can extend the list of information to be output with your own attributes, for example, an attribute from the title block of a drawing in the relevant model. To do this, open the `3DPlot.ini` file located in the user directory with a text editor. The information to be output is defined using Python expressions.

## Technical Information about 3D Stamp Printing

All user-specific settings for printing additional information in the 3D view are defined in the `3DPlot.ini` file in the user directory. For example, because update installations do not lose their own customizations.

The basic specifications for the additional information are loaded from the `3DPlot.ini` file of the same name, which is located in the *CADdy++* installation directory **\Program\Bin**. If necessary, copy the passages from this file to the file in the user directory whose settings you want to change. When you start the **File, Print ...** menu command in the 3D window, the entries of the ini file in the user directory are loaded last and are therefore decisive.

### General settings

The section [General] contains the following general settings:

[General]

BeforePlot=false	true = Text is printed at the beginning false = Text is printed at the end
------------------	---

	When printing from 3D only false possible
FontName="Arial"	General font
Rows = 50	Number of rows on the paper; the result is the text height.
Indentation = 0.02	Factor for feed related to the paper width. 0 = Text starts at paper edge
PlotHeadline=false	true = Header line is printed false = Header line is not printed
Headline=Datum tt.mm.jj	Current selection to be printed for the header line
PlotFootline=false	true = Footer line is printed false = Footer line is not printed
Footline=Modellname mit Pfad	Current selection to be printed for the footer line

### **Creating Custom Information**

Additional attributes for the expression can be created by Python expressions from `_Runtime.py`, whereby a transfer parameter 'opModel' is provided, which represents the Op model. This allows you to access any further information in the model via the Python API, e.g. the active drawing, including a title block, including an attribute value.

The transfer parameter 'opModel' does not necessarily have to be used, e.g. for date output you do not need it.

Even fixed texts must satisfy the rules of a Python expression, i.e. surrounded with `""`. If umlauts occur, a "u" must be placed in front of them.

Line up several Python texts by + to combine several 'Python formulas' and / or fixed texts in one line. An additional empty line can be reached by an empty text string `""`.

#### **Examples:**

```
[Date and time]
```

```
DateTime('%d.%m.%Y %H:%M')
```

```
[Date dd.mm.yy]
DateTime('%d.%m.%y')
```

```
[Date dd.mm.yyyy]
DateTime('%d.%m.%Y')
```

```
[Time hh:mm]
DateTime('%H:%M')
```

```
[Time hh:mm:ss]
DateTime('%H:%M:%S')
```

```
[Model name]
ModelName(opModel,0)
```

```
[Model name with path]
ModelName(opModel,1)
```

```
[Signature]
u"mit freundlichen Grüßen"
unicode(os.getenv('USERNAME'),'cp1252')
```

```
[CopyRight]
"CopyRight by DataSolid GmbH 2019"
```

```
[Username]
unicode(os.getenv('USERNAME'),'cp1252')
```

```
[Computername]
unicode(os.getenv('COMPUTERNAME'),'cp1252')
```

## Create Pixel File from 3D View



The new menu command **File, Create Pixel File..** creates a pixel file from the 3D view. The main topic of this command is to create high resolution pixel images in the file formats BMP, JPG and TIF.

The dialogue window to save the pixel file is offered after clicking the **Create** or **Create & Close** button.

In addition to the extended settings for a perfect print described later on, also an interactive preview is implemented to show the 3D view in relation to the page size. Below please find the commands for the preview:



### Move View

Click in the preview window with the middle mouse button pressed. Move the mouse to move the view.



### Zoom In / Zoom Out

Zooming can be done with the scroll button or by pressing the right mouse button and moving the mouse up and down.


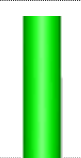




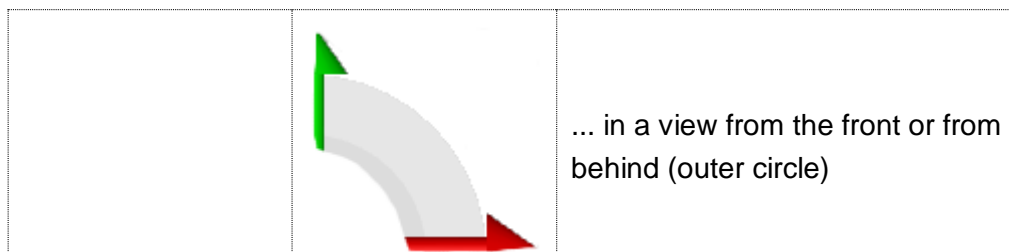
### Rotate View

Click in the preview window with the left mouse button pressed. Move the mouse to rotate the view.

### Display the interactive coordinate system

If this option is active, the view will rotate by a click on the following position....

		... in view direction of the arrow
		... by 90° (arrow shaft)
		.. displays all objects
		... in the perspective view (inner circle)



## Define Settings for to Create a Pixel File

After clicking the menu commands **File, Create Pixel File...** with a 3D window active a completely new dialogue window appears to create a pixel file from the 3D window.

### DPI (dots per inch)

Input field to define the resolution of the points for the output. The higher the value, the better the quality of the output.

### Portrait / Landscape

Select the desired option to print in portrait or landscape format.

### Pixel Size

Displays the resulting size of the output in pixels according to the resolution DPI and the current paper size.

### Size

Input and Output field for the paper size in mm. the size can be entered here or be selected from the underlying list of standard paper sizes. When choosing other units than mm, the unit must be entered after every value and confirmed by pressing the **Enter** key: 1cm x 0,02m.

### Border

Entry field to define the border width to the selected paper size in mm. When choosing other units than mm, the unit must be entered after every value and confirmed by pressing the **Enter** key: 1cm x 0,02m.

### Paper Size

Selection list with standard page formats.



By clicking the displayed icon the actual view from the 3D window is taken over into the preview window.

Using an orthogonal view in the preview the view angle and the zoom is adapted, in a perspective view due to the different view modes only the view angle can be adapted.

### Perspective View

When activated, the current view is printed in a perspective view instead of an orthogonal view.

### Display Coordinate System

The interactive coordinate system allows to modify the view in the preview window. It will also appear in the pixel file, when activated.

### Antialiasing

This option is dependent on the computer hardware, especially from the graphic board and the installed drivers. With the antialiasing option the lines can be smoother. The higher the value the more the lines are smoothed.

### 3D Grafic

Allows the print of several 3D file formats, without loading the file before. After clicking this button, you can select a 3D object you wish to print.

### 3D Model

The view of a 3D graphic is closed and the current 3D model is displayed again.

### Activate Color Gradient

Allows to place a gradient in the background for printing. The color will change from the first selected color to the second selected color. When this option is deactivated the background is printed in the given color.

## Copy 2D data to the clipboard

---

If you wish to copy the drawing or only the visible window of the current 2D drawing window to your clipboard so that you can re-use it in other application programs, proceed as follows:

1. Select one of the menu commands **Process, Vector Data → Clipboard.** or **Bitmap → Clipboard.**

When the output is as bitmap the program will generate a pixel graphic, and when the output is in the form of vector data you will obtain vector graphics that can be scaled.

To access the settings from the **Print** dialog window while transferring to the clipboard, use the option **Print to Clipboard.**

2. Here you have all the options from the **Print** dialog window and any printer driver can be selected to influence the quality of the output.

To print to the clipboard use the menu command **Process, Clipboard, Print to Clipboard** or use the key combination **Ctrl+C**.

With the key command the current printer settings are used.

Afterwards you can change to a different application and paste the data using **Process, Insert** or by using the key combination **Ctrl V**.

**Please note:**

When you scale vector data that have been pasted in documents of other programs, the changes will also affect the line thickness.

## 3.9 Display information from Model Management

---


If you have installed *CADdy++ Model Management*, you can display the parameter settings for the program and the records saved in the database.

The data cannot be edited.

### Display administrative settings

---

If you wish to see the administrative settings for *CADdy++ Model Management*, follow this procedure:

1. Load the **EDM system** toolbox.  
Do so by using the menu command **View, Show Toolbar**.
2.  Load the command **Administrative settings** by clicking the symbol shown here.


Further information on these settings are to be found in the documentation on the full version of *CADdy++ Model Management*.

Close the display by clicking **OK** or **Abort**.

### Display record

---

If you want to see the data relating to the current drawing in *CADdy++ Model Management*, use the following procedure:

1. Load the **EDM system** toolbox.  
Do this with the menu command **View, Show Toolbar**.
2.  Then load the command **Display record** by clicking this symbol.



Further information on these settings are to be found in the documentation on the full version of *CADdy++ Model Management*.

Close the display by clicking **OK** or **Abort**.

## 3.10 Exit the Program

---

There are a number of options available for quitting a work session of *CADdy++ View & Plot*

- Using the menu command **File, Exit**
-  By clicking this system command button in the main *CADdy++ View & Plot* window.
-  By clicking this system command button in the main *CADdy++ View & Plot* window and then selecting the menu command **Close**.

If changes have been made to the model, you will see a prompt asking whether the model data are to be saved.

If your reply is **Yes**, you will see a dialog box so that you can save the file under a different name.



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