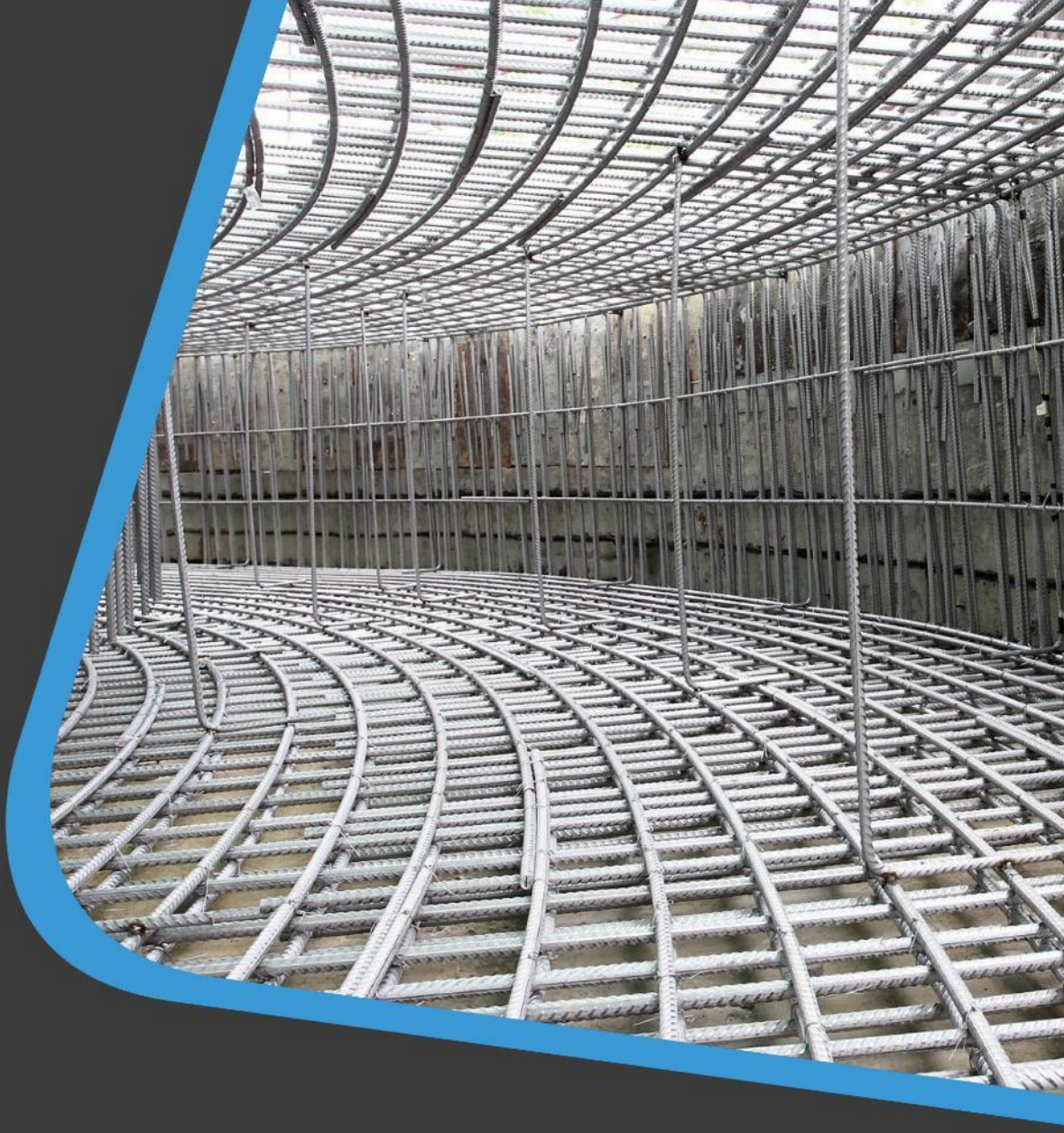


CADS Drawing Environment
User Guide

CADS
GLOBAL CONSTRUCTION
SOFTWARE AND SERVICES

AUTODESK
Authorised Developer

Microsoft
Partner

Contents

1 Introduction

CADS Drawing Environment has been designed to work with AutoCAD and AutoCAD Architectural version 2026.

2 Program Features & Installation Requirements

2.1 Hardware and Software Requirements

2.1.1 Hardware

A computer capable of running AutoCAD 2026 or higher; or AutoCAD Architecture 2026 or higher is a requirement. Please refer to the Autodesk website for detailed specifications of hardware for AutoCAD.

2.1.2 Software

CADS Drawing Environment supports Windows 10 and 11 installed with AutoCAD and AutoCAD Architecture version 2026.

2.2 Program Features

2.3 Installation

Full detailed instructions for installing all CADS Detailing Applications which include CADS Drawing Environment can be found in the accompanying CADS AutoCAD Detailing Applications Installation Guide.

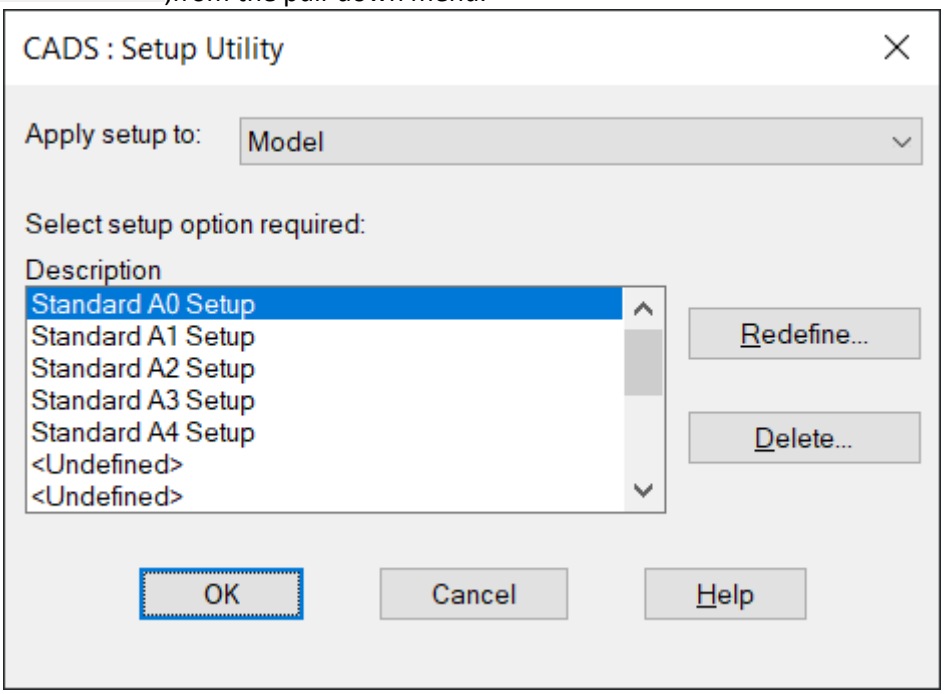
2.3.1 Loading the CADS Drawing Environment Toolbar

CADS Drawing Environment Toolbar

The CADS Drawing Environment toolbar is not loaded automatically. However, if the toolbar is required, load the pull down menus by typing “Menubar” at the AutoCAD command line and set it to 1. Then go to the Tools menu, select Toolbars then CADS-DE and then CADS-DE.

3 Drawing Setup

If you wish to set up a drawing from within CADS Drawing Environment, select Drawing Setup... ,from the pull-down menu.



This will bring up a dialog box which lists the currently-available drawing set-ups, together with the following buttons:

- Paper space setup;
- Redefine;
- Delete.

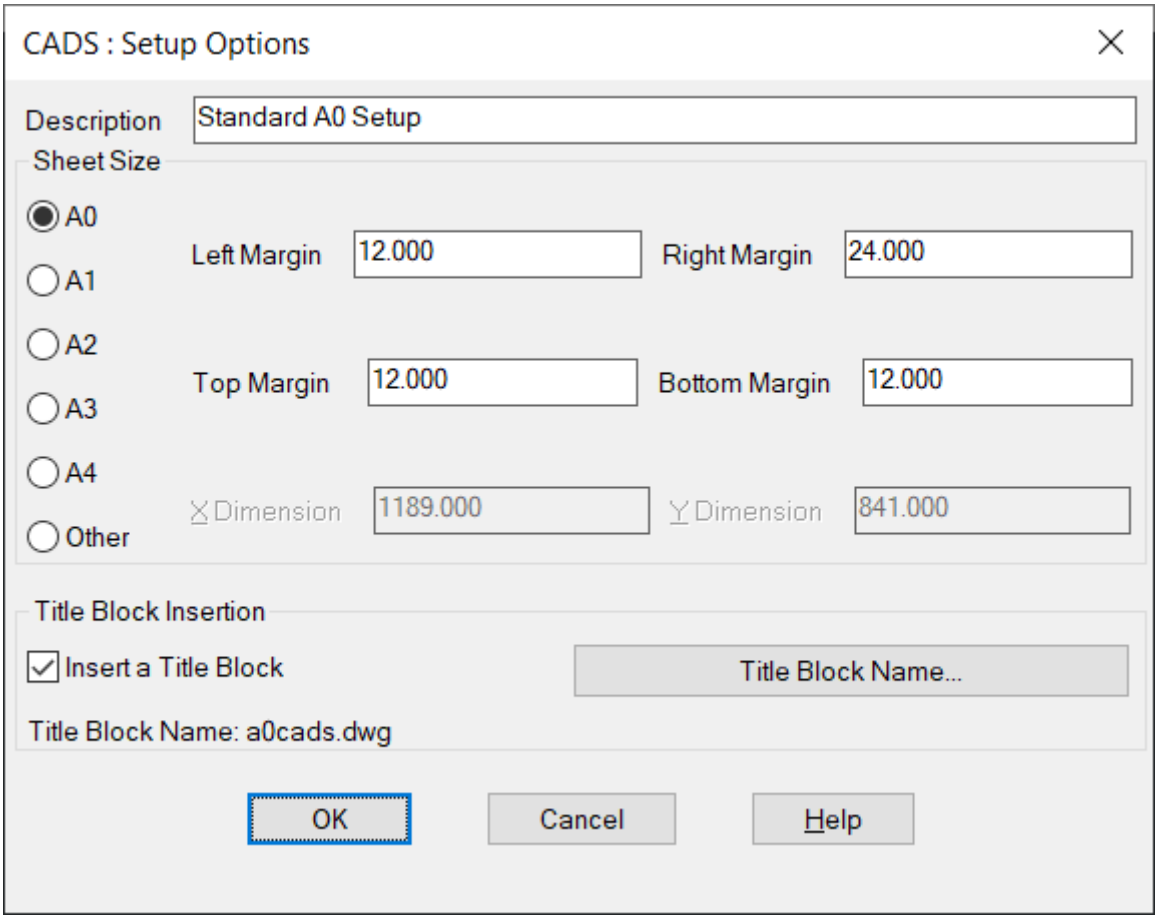
If you do not wish to Redefine or Delete an option in the menu then select the option containing your desired setup parameters and press the OK button.

3.1 Paper Space Setup...

A 'Paper Space Setup' inserts the title block onto the paper space; only after creating an MVIEW viewport (by using 'Create Scale area' in CADS Scale or the AutoCAD command 'MVIEW') can the user begin to create their drawing. It is recomended that unless you are an experienced user of MSPACE/PSPACE, Scale is used as a means of handling paper space drawings of this type.

3.2 Redefine...

The parameters associated with the currently highlighted setup option are displayed in a new dialog box.



3.2.1 Redefine Setup Otions.

These can be edited to reflect the setup required:

- Description;
- Sheet Size;
- Title Block Setup.

3.2.1.1 Description...

This may be modified to change the description displayed on the main Setup menu for this setup 'style'.

3.2.1.2 Sheet Size...

A standard sheet size can be selected or, alternatively, 'Other' may be chosen allowing the user to enter sheet dimensions into the X & Y Dimension fields.

If margins are entered and no title block has been specified then the polyline drawn to show the extents of the drawing is offset accordingly. If there is a title block selected only the bottom and left margins are used to offset the insertion point of the title block. Under both of these circumstances AutoCAD Point entities are inserted at the corners of the 'paper' - these may be 'turned off'

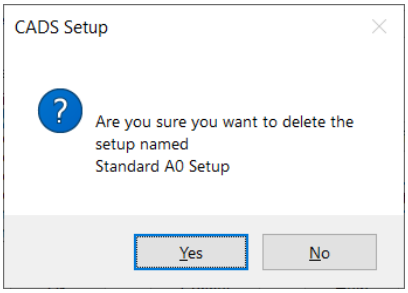
by setting all the margins to zero.

3.2.2 Title Block Insertion...

When toggled to on (marked), the user is able to select the 'Title Block Name' button in order to select their desired title block.

3.3 Delete...

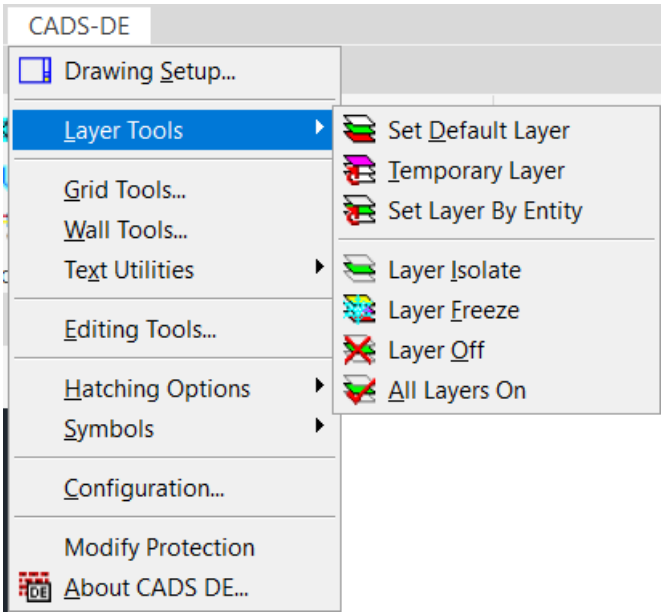
Use this option to clear the currently highlighted setup option.



This facility is only reversable by using 'Redefine...' to change the setup name.

4 Layer Tools

If you wish to set up layers within a drawing from within CADS Drawing Environment, select 'Layer Tools', from the pull-down menu.

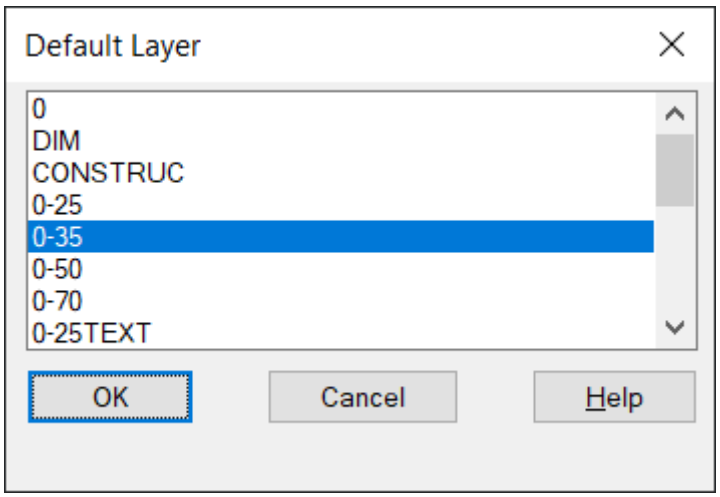


The Layer Tools offers the following options:

- Set Default Layer;
- Temporary Layer;
- Set Layer By-Entity;
- Layer Isolate;
- Layer Freeze;
- Layer Off;
- All Layers On.

4.1 Default Layer

The 'Default Layer' is a layer that is automatically set whenever any of the AutoCAD draw commands (line, polyline etc..) are called from the pop down menus.



The 'Default Layer' is set by the user in the Miscellaneous Configuration or by picking the Set 'Default Layer' command. It can also be switched On or Off in these two places.

If the 'Default Layer' is switched off then when the user picks any of the draw commands, they will be drawn on AutoCADs currently set layer rather than the 'Default Layer'.

There is also a 'Dimensions Layer'. This is the 'bed partner' of the 'Default Layer' and works in the same way, except that it is set when any dimension drawing commands are picked from the pop down or tablet.

These two tools were created to make it easier for the user to swap between drawing and dimensioning without having to go to the trouble of changing the layer each time.

4.2 Temporary Layer

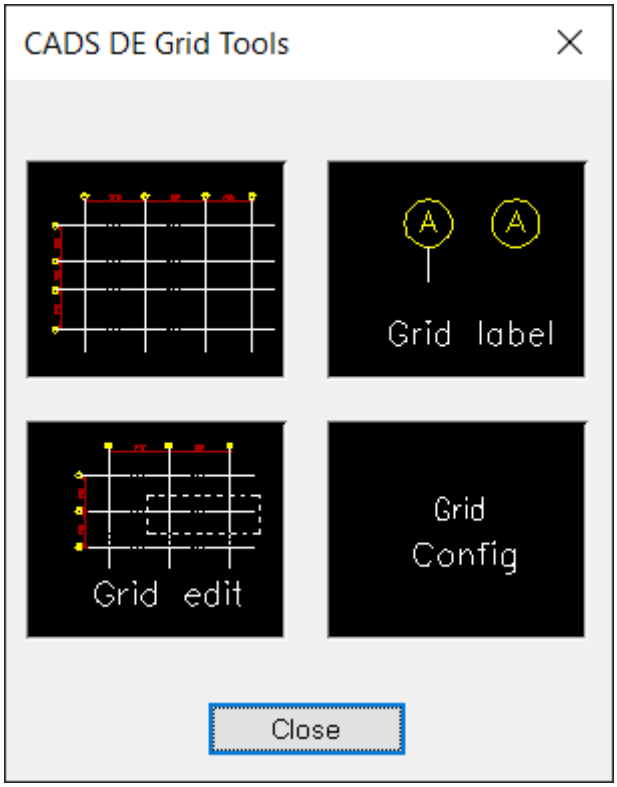
The 'Temporary Layer' is a layer that is toggled on or off by the user, for temporary use. When the Temporary Layer command is picked, the 'Default Layer' takes the value of the 'Temporary Layer'. Therefore, when the 'Temporary Layer' is set, all entities drawn, that would have appeared on the 'Default Layer', will now appear on the 'Temporary Layer'. Selecting the command again will toggle it off and they will return back to being drawn on the 'Default Layer'.

An example of where the 'Temporary Layer' may be used is to do construction lines. The user can do some drawing on the 'Default layer', select 'Temporary Layer', do some construction line work, then select the command again to toggle it off and do some more non-construction drawing.

If the 'Default Layer' tool if OFF then selecting the 'Temporary Layer' command behaves slightly different. Instead of setting the 'Default Layer' to the 'Temporary Layer', it sets the current layer to the 'Temporary Layer' value. Selecting the command again toggles the previous current layer back.

5 Grid Tools

The Grid Tools allow the user to create a labelled grid on the drawing at user specified size and with user specified labelling.

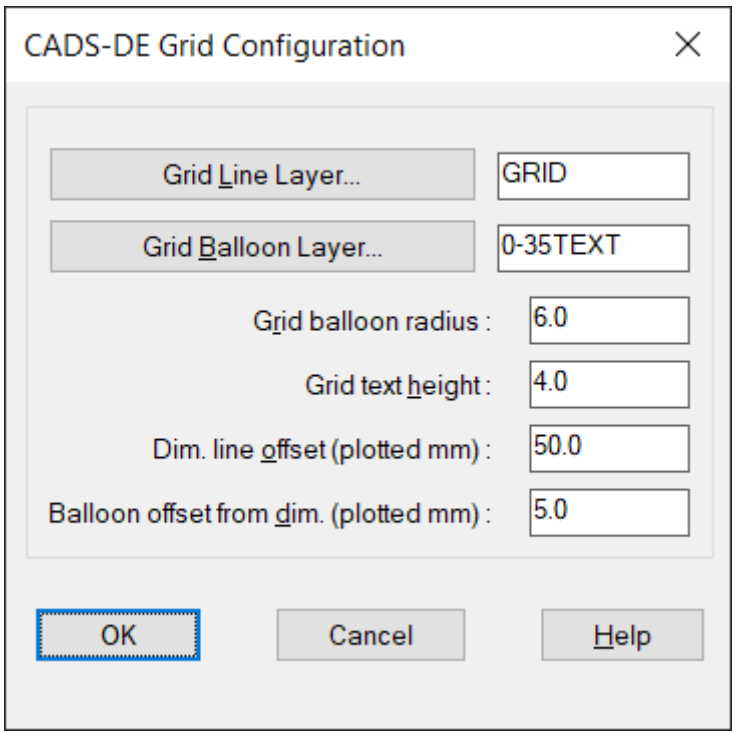


The grid edit tools allows you to edit the grid after it has been drawn on the drawing. You can trim a block from the grid allowing smaller grids to be drawn within larger ones, for example.

There is also the facility to insert one grid line with a label or simply just a label.

5.1 Grid Configuration

There are various grid configuration items that can be set up to affect the grid tools every time they are used.



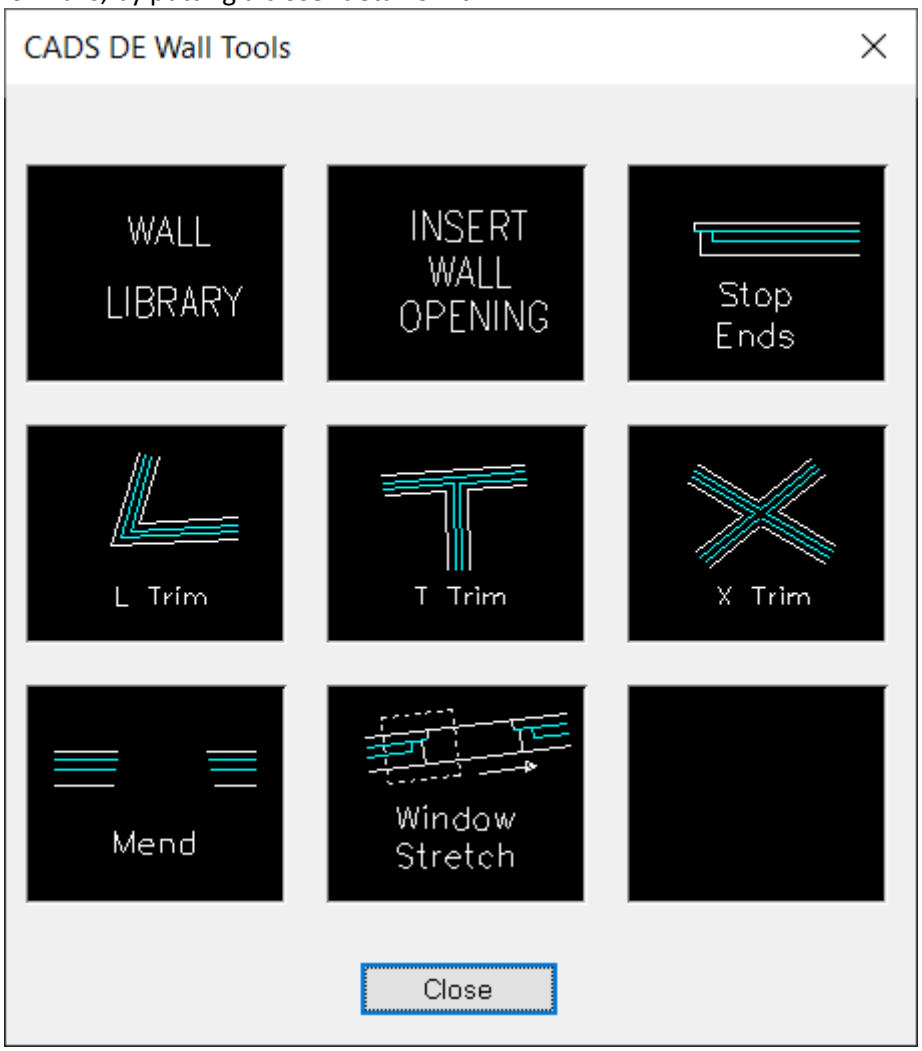
The grid line layer and grid label balloon layer can both be set here. As can the label balloon radius and size of the label text that appears within the balloon.

The dimension line offset setting determines the position that the dimension line will be drawn. It is basically the distance between the outside grid line and the dimension line.

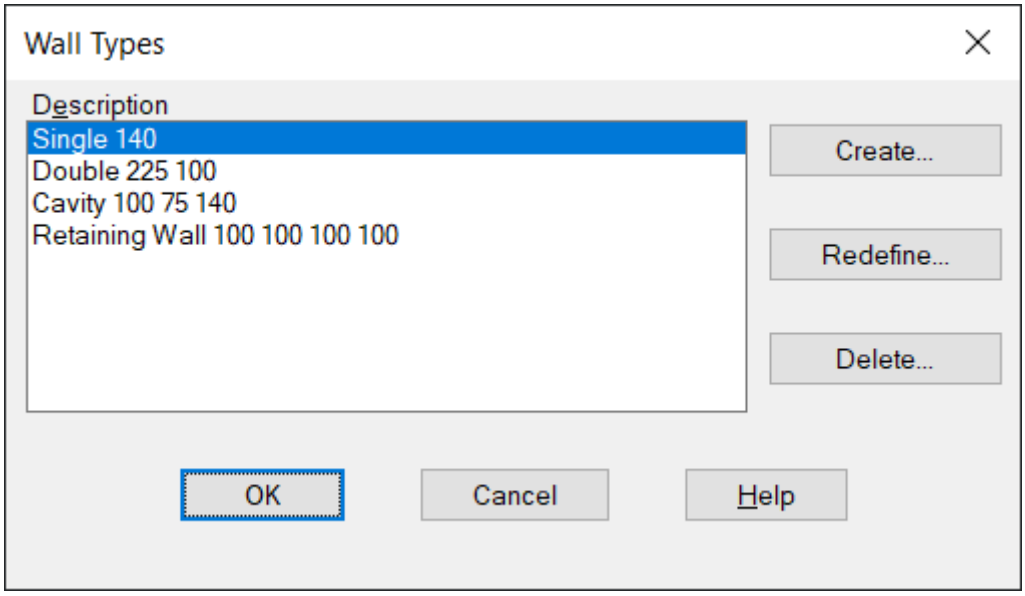
The Balloon offset from dim input allows the user to specify where the balloon labels appear. It is actually the distance between the dimension line and the bottom of the balloon. i.e. The distance between the balloon labels and the outer grid line is, therefore, the sum of the above two inputs (Dimension line offset and Balloon offset from dim).

6 Wall Tools

The user is provided with various wall tools. You can draw walls of varying type, insert various opening symbols into the walls, and trim and tidy the walls in various ways. There is also a tool to tidy the end of walls, by putting a closer detail on it.



6.1 Wall Library



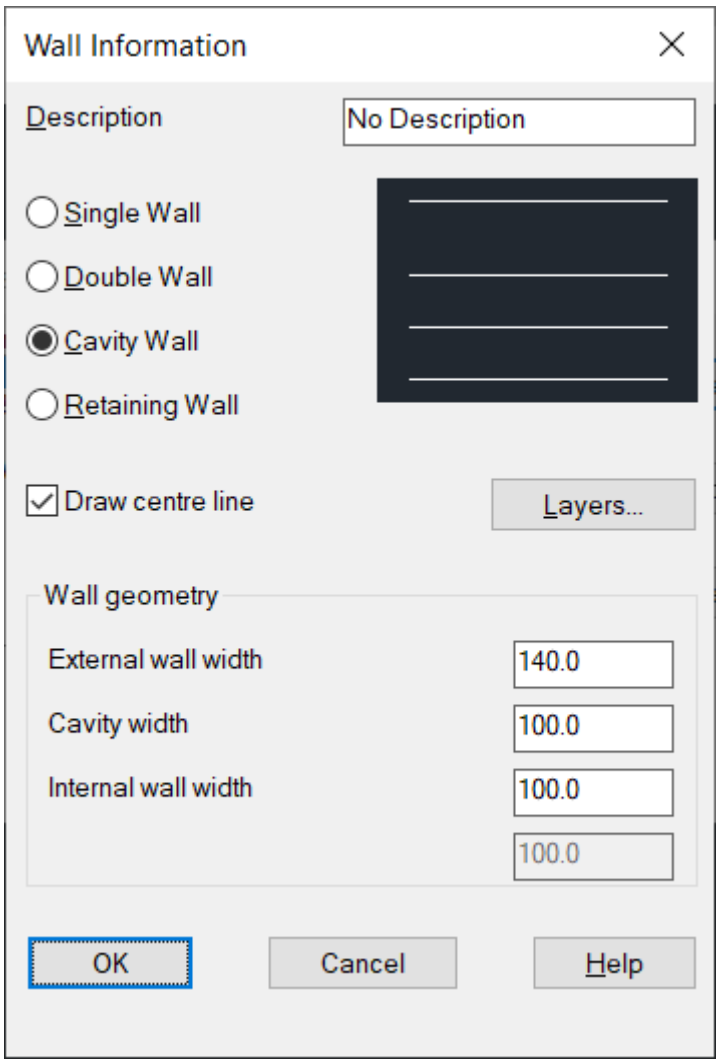
Upon entering the Walls Library dialog box, you will see a scrolling list of walls. The user has the facility to create new walls or edit existing ones. To draw one of the walls from the list simply, double click on the desired wall, or single click on it, to highlight it, and pick 'OK'. You will now be asked to draw the requested wall on the drawing by picking the first point, second point etc.. much like the AutoCAD Line command.

If you wish to add a new user wall to the drawing then highlight one of the undefined fields in the list, by single clicking on it, then pick the 'Redefine...' button. To redefine an existing one, pick on it to highlight it, then pick the 'Redefine...' button.

To delete a wall from the library, highlight the relevant entry and pick the 'Delete...' button.

6.1.1 Creating/Redefining a wall.

Picking the 'Redefine...' button from the Wall Library dialog box will bring up the Wall Information dialog box. This allows you to change the various parameters of a specific wall in the library.



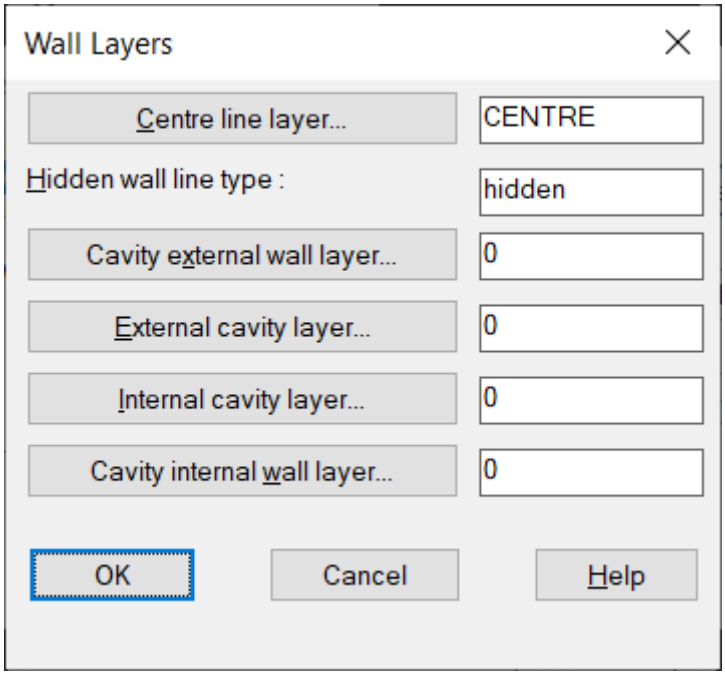
There are four specific wall types, single, double, cavity and retaining wall. Simply click on the relevant radio button to select the desired wall type. You can enter the distances between the wall lines and by picking the 'Layers...' button, you can alter the layers that these lines will appear on.

An image to the side will display a sketch of the wall, to scale, to give an idea what the user will get when he draws it on the drawing.

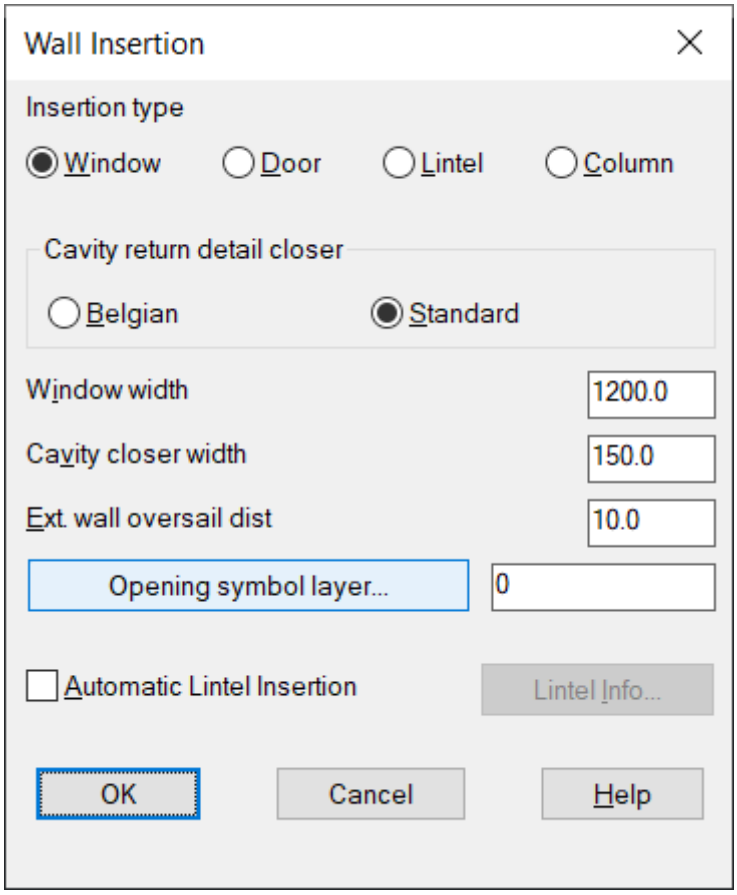
It is important to enter a description for the wall in the 'Description' field, edit box. This will make it recognisable in the wall library scolling list. Failure to do this will make it remain undefined, and therefore, unpickable.

When you are happy with your inputs, pick the 'OK' button to add the wall to the library. Picking 'Cancel' will lose the inputted data.

6.1.1.1 Layers



6.2 Wall openings.



This dialog box allows the user to define a wall insertion before actually inserting it on the drawing. The four types of wall insertion are window, door, lintel and column. The desired option can be

selected by picking the relevant radio button. The inputs in this dialog box will change dependant upon which of the insertion types is picked.

The user can select between Standard and Belgian, for the cavity return detail closer type, by picking the relevant radio button.
He can also enter the various, relevant distances for the insertion object, and the closers.

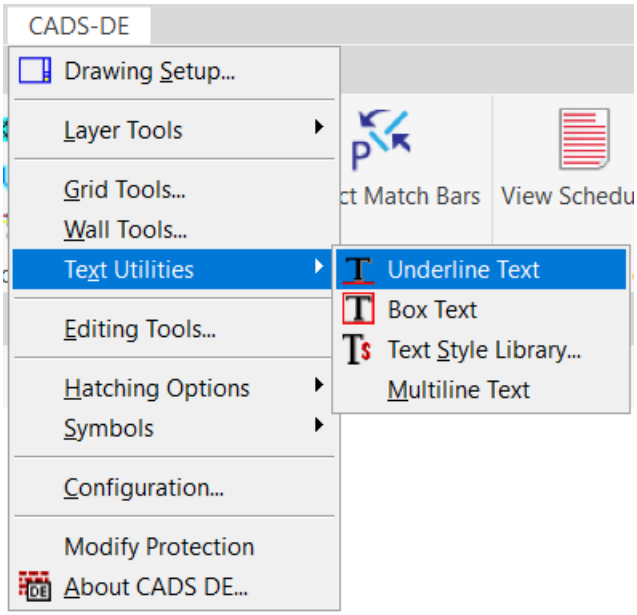
There is also the ability to change the layer that the opening symbol will be inserted on here.

As well as the facility to insert a lintel singularly, there is also the facility to insert one automatically, when inserting a different object. To do this, activate the 'Automatic Lintel Insertion' toggle button. This will also ungrey the 'Lintel Info...' button, allowing you to pick it and change various attributes of the lintel, like layer and size.

When you are satisfied with your inputs, select the 'OK' button to proceed and insert the opening object on the drawing. Select 'Cancel' to abort.

7 Text Utilities

The text utilities menu contains some tools to edit text entities on your drawing.

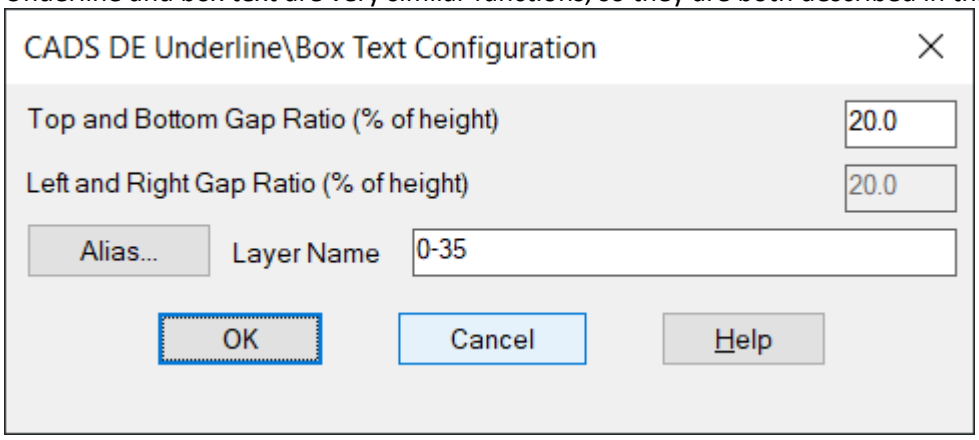


The utilities that included are:

- Underline Text;
- Box Text;
- Text Style Library;
- Multiline text.

7.1 Underline/Box Text

Underline and box text are very similar functions, so they are both described in this help entry.

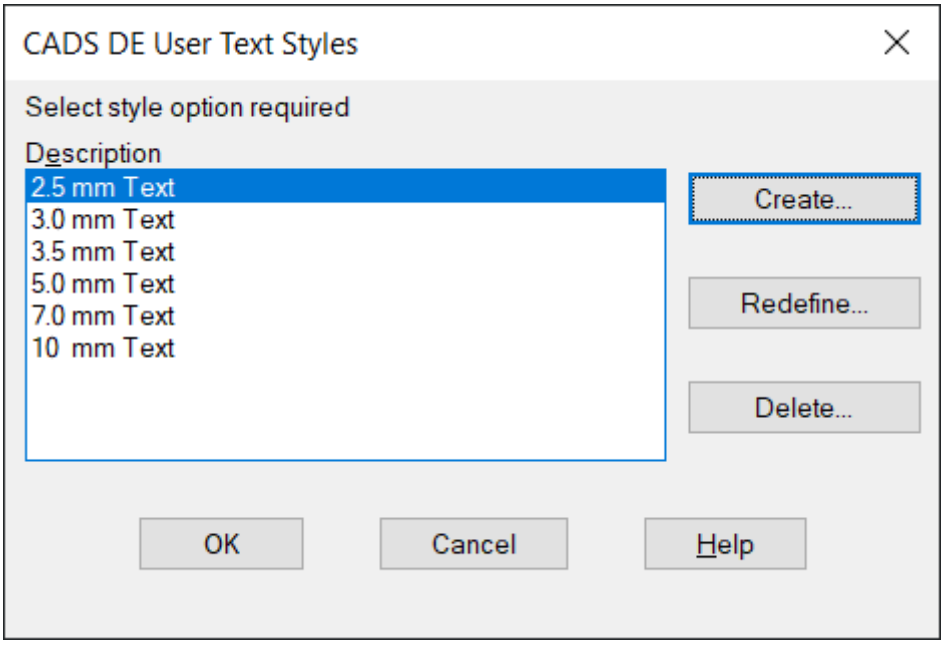


First pick either Box or Underline Text from the Text Utilities menu. A dialog box, with two edit boxes will now appear on the screen. If you have chosen the underline function, one of these will be greyed out. These editboxes control the gap that is left around the text when it is boxed or underlined. These are entered as a percentage of the overall text width. The other editbox is for selecting the layer that

the box is drawn on. When you are satisfied that these are correct press the 'OK' button and select the entities to be boxed or underlined.

7.2 Text Style Library

This is a library tool to allow the user to set up standard styles of text. Each style set-up has the following elements to it.



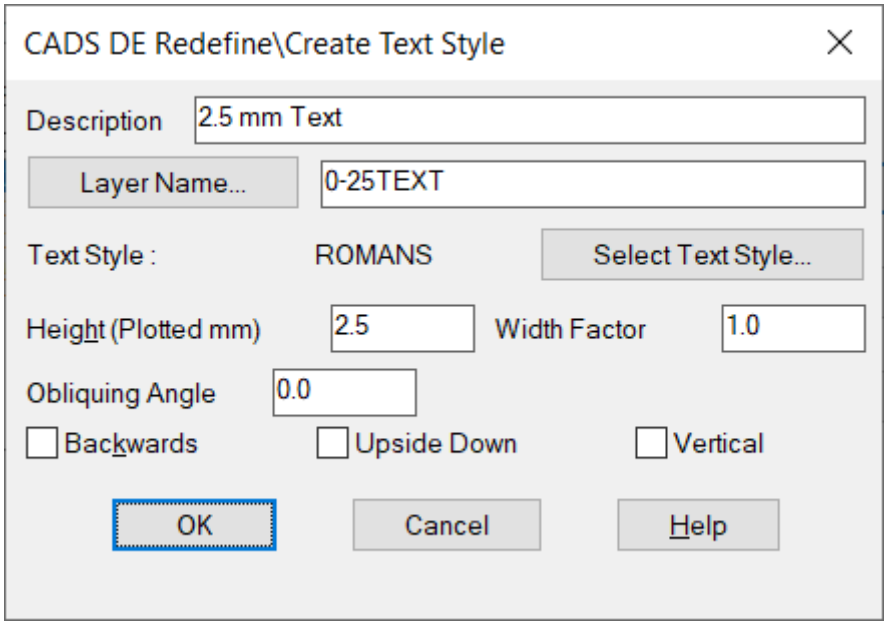
The first dialog box that is displayed has a list box containing the text style set-ups. If you wish to draw text using one of these just double click on the one you want out of the list and the program will go straight into the drawing command, allowing the user to enter the text they want to draw. You can also single click on the list and press the button marked 'OK' to draw text.

7.2.1 Creating/Redefining New Styles

To create a new text style or redefine an existing text style in the library, use the 'Redefine...' button. To create a new style, first select an empty (undefined) entry in the text styles list and select the 'Redefine...' button. To redefine a style, select the style you wish to redefine before selecting the 'Redefine...' button.
The Redefine dialog box will then be displayed. This screen allows the user to enter all the information about the current text style library entry.

The button marked 'Select Text Style...' allows you to select the AutoCAD text style you wish to define this library entry as having, from those available on the drawing. If the text style you require is not listed selecting the 'Add Style' button will allow you to pick a font shape file (.SHX) from disk. The selected font will then be added to the style list.

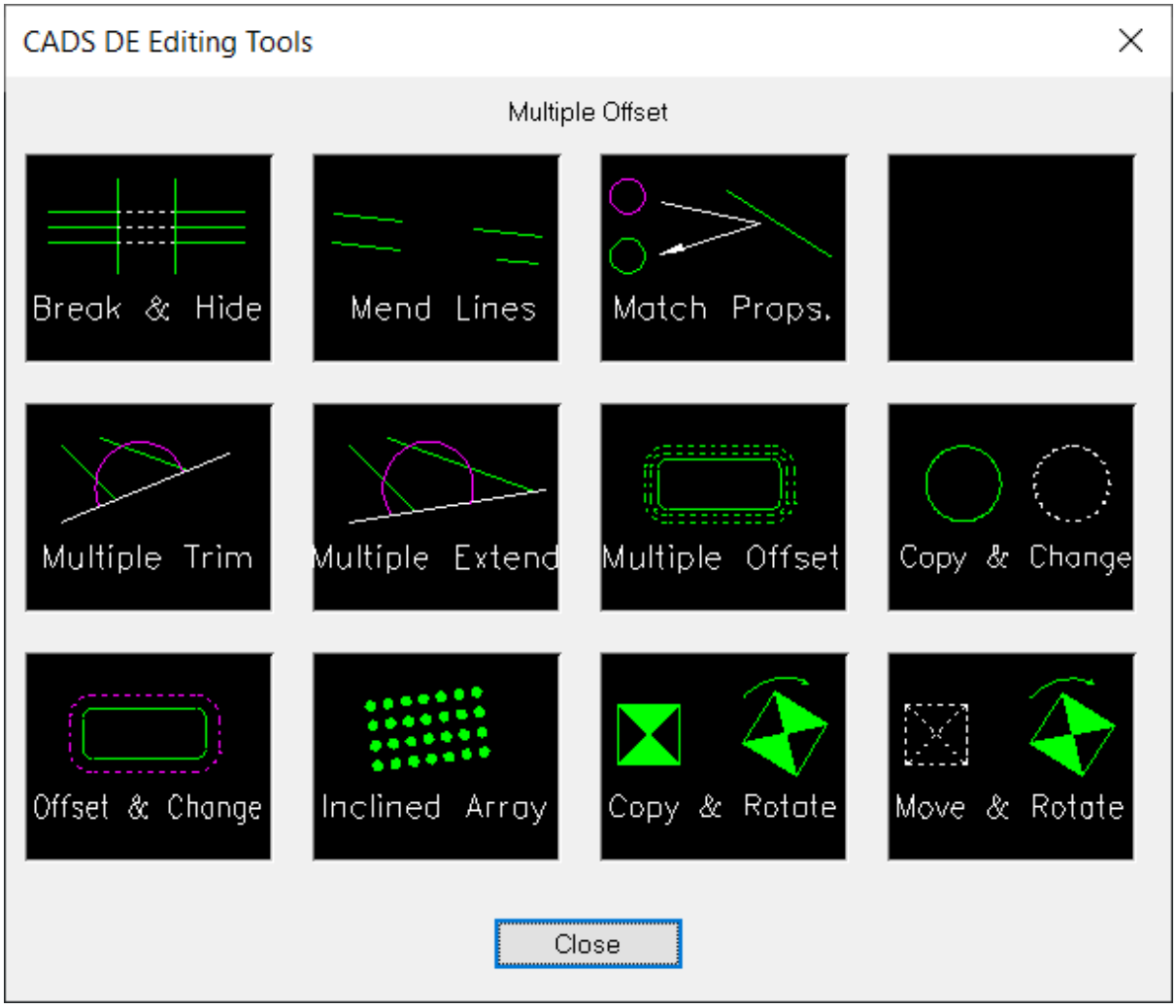
Other parameters that can be set for each style set-up include the plotted text height, the width as factor of the height, and the obliquing angle.



- Description;
- Layer Name;
- Text Style;
- Plotted Height;
- Width Factor;
- Obliquing Angle;
- Backwards;
- Upside Down;
- Vertical.

8 Editing Utilities

The editing utilities menu contains various tools to edit entities on the drawing.



The tools included are:

- Break & Hide - Break and hide line and polyline entities;
- Mend Lines - Join two lines with a gap into one;
- Match Props - Inherit properties of existing entity;
- Multi Trim - Trim more than one entity at a time;
- Multi Extend - Extend more than one entity;
- Multi Offset - Multiple AutoCAD offsets;
- Offset & Change - Offset and change property of entity;
- Inclined Array - Array copy entities at an angle;
- Copy & Rotate - Copy and rotate entities;
- Move & Rotate - Move and rotate entities.

8.1 Break and Hide

This command allows you to hide certain entities behind other entities. First select the entities to hide, press the return key and then select the entities to hide behind. The program will then prompt you to type either Trim or Hide.

8.2 Mend Lines

This command will allow you to join two lines together to become one line. It will only allow you to do this if the end of either of the lines could be extended to form the other line. This is a useful command that could be used after you had trimmed the middle out of a line.

8.3 Match Properties

This command allows you to make a group of entities inherit various properties of another entity. For example, the entities color, layer or linetype. First select the objects to change and then enter the property to be changed. Finally select an entity with the desired property.

8.4 Multi Trim

This command lets the user trim more than one entity at a time. The command works in the following way. First select the cutting edges. The program will then prompt for the points of a fence line that should cross the entities that you wish to trim. The point at which the fence line crosses the trim lines determines which side of the cutting edge they are to be trimmed. This method allows you to trim some lines one side of the cutting edge and some on the other, all in one command.

8.5 Multi Extend

This command lets the user extend more than one entity at a time. The command works in the following way. First select the cutting edges. The program will then prompt for the points of a fence line that should cross the entities that you wish to extend. This method allows you to extend lines one side of the cutting edge and some on the other, all in one command.

8.6 Multi Offset

This command speeds up the AutoCAD offset command allowing the user to continue offsetting objects until he has as many offset copies as he wants. Select the objects to offset and pick a point representing how much you want to offset the object by. You can also enter the offset distance manually.

8.7 Offset and Change

This command creates offset objects and assigns them the properties of other entities. Select the objects to offset. Pick a point to offset by, or enter the distance. The program will then prompt you for the property to change in the offset entity. Finally select an object with the desired property.

8.8 Inclined Array

Select the entities to be copy arrayed, and then pick the base point in the array. The program will then prompt you to enter the angle that the array is to be inclined at. Finally enter the distance to the next item or 'Finish' to quit.

8.9 Copy and Rotate

This command combines the AutoCAD copy and rotate commands. First select the objects you want to copy, pick a base point to copy from. Pick the destination point and new copy will appear there. The program will then prompt for the rotation base point, either type the angle to rotate the object by or pick a point on the drawing.

8.10 Move and Rotate

This command is essentially the same as the copy & rotate command except it does a move instead of a copy. Pick a base point and then the destination point. The program will then prompt for the rotation base point, either type the angle to rotate the object by or pick a
point on the drawing.

9 Symbols

9.1 User Symbols Library

This command allows the user to create and modify libraries of symbol blocks. Other information is stored with each block, this includes the X and Y scales, a description and the layer it is to be drawn on. The symbols can be stored in various places. They do not all have to be stored in the same directory.

When you single click on a symbol block icon, a description of that symbol will be displayed near the top of the dialog box.

To insert a symbol either double click on the symbol you want to draw or just click once and then press the 'OK' button. The program will then prompt you for the insertion point and the rotation angle.

9.2 User Symbols Create

To create a new symbol block, select the 'Create...' button from the User Symbols Library dialog box. The create/redefine dialog box will now appear. It is important to note that if you wish to create a symbol, you should have already drawn all the entities that will make up this symbol.

The first input is the block/slide filename input. This is the name of the file that you wish the new block and slide name to be saved.

The slide and block will be stored in the same directory, with the same name, but, obviously, with different extensions (.SLD for the slide, .DWG for the block). The user has the option here to enter the full path with the symbol block name. If just the name is entered without the path, then the files will automatically be stored in the 'cads-de\blocks' directory.

One point to note, if you make two symbol blocks of the same name (i.e. in different directories), then there will be a clash because AutoCAD does not allow two blocks of the same name to be stored on a drawinging.

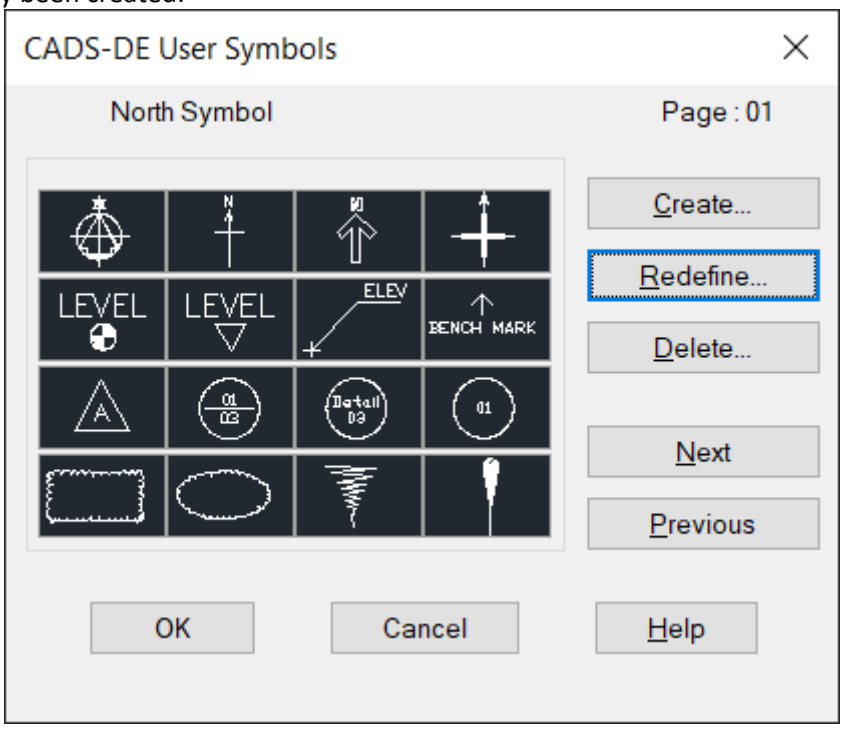
You can enter a symbol description here to aid in identifying the symbols on the library screen. There is also the X and Y scale that you wish you block to be inserted at. If you enter 0.0 for these values (the default), then the block will ask for input of the scales at insertion time.

When you have completed entering the information select the 'Make Slide/Block File' button. The dialog box will now disappear and you must pick all of the entities you want the symbol block to be composed of. When you have done this press return and pick an insertion point for the symbol. The dialog box will now reappear.

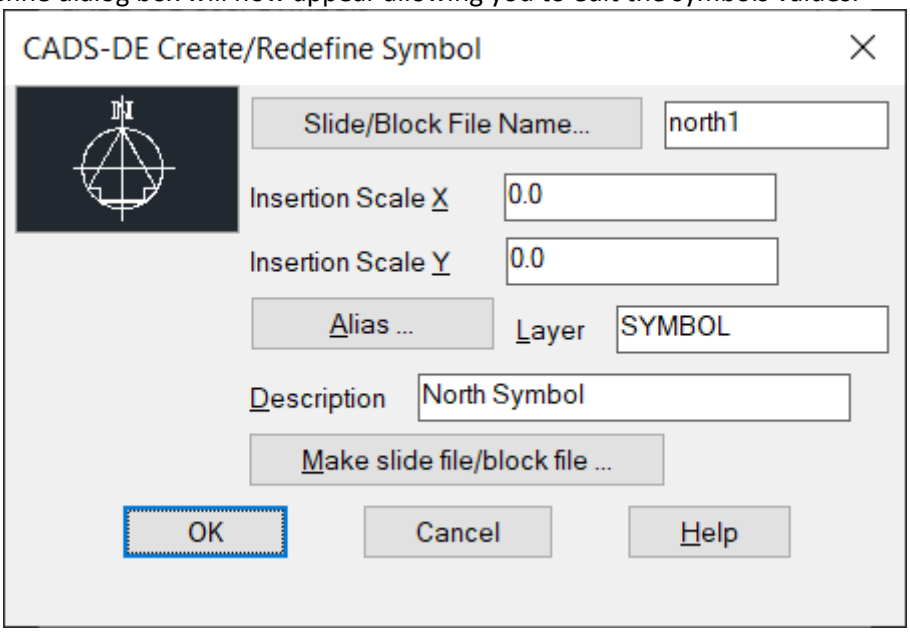
If you want to add this symbol to the library you should select the 'OK' button now. Otherwise, select the 'Cancel' button.

9.3 User Symbols Redefine

This part of the symbol's library works in exactly the same way as Create, except it deals with a symbol that has already been created.



Select the symbol you wish to change and then pick the button marked 'Redefine...'. The Create/Redefine dialog box will now appear allowing you to edit the symbols values.



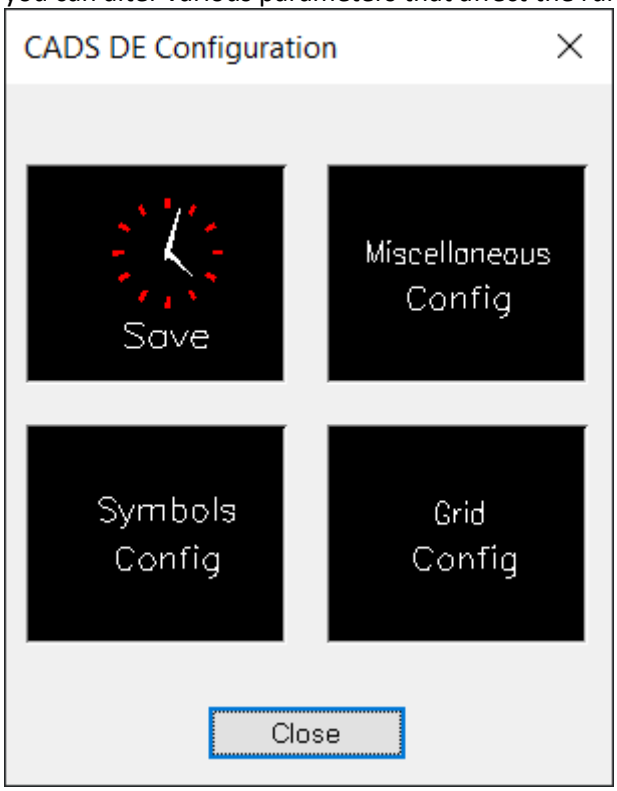
When you have finished press the 'OK' button to keep your changes or the 'CANCEL' button to discard them.

9.4 Dynamic Symbol Library

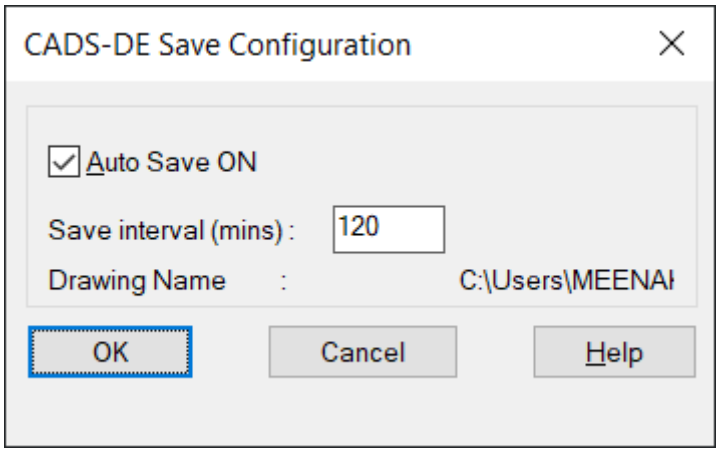
The Dynamic Symbols are symbols that could not be included in the User Symbol Library because they are not blocks. The reason for this is that they have parametric type inputs, variable lengths, specific text positioning etc. Simply double click on the symbol you wish to place on the drawing. You can also single click on the relevant icon, to highlight it, then pick 'OK'. Selecting 'Cancel' will drop the user out without doing anything.

10 Configuration

This screen allows you to select which Drawing Environment configuration dialog box to bring up. From these dialog boxes you can alter various parameters that affect the running of CADS-DE.



10.1 Save Config



The Automatic save feature allows the user to set a timer so that every, say 10 minutes, for example, AutoCAD will automatically save the current drawing. This helps for backup purposes, and means less work will be lost if the machine goes down.

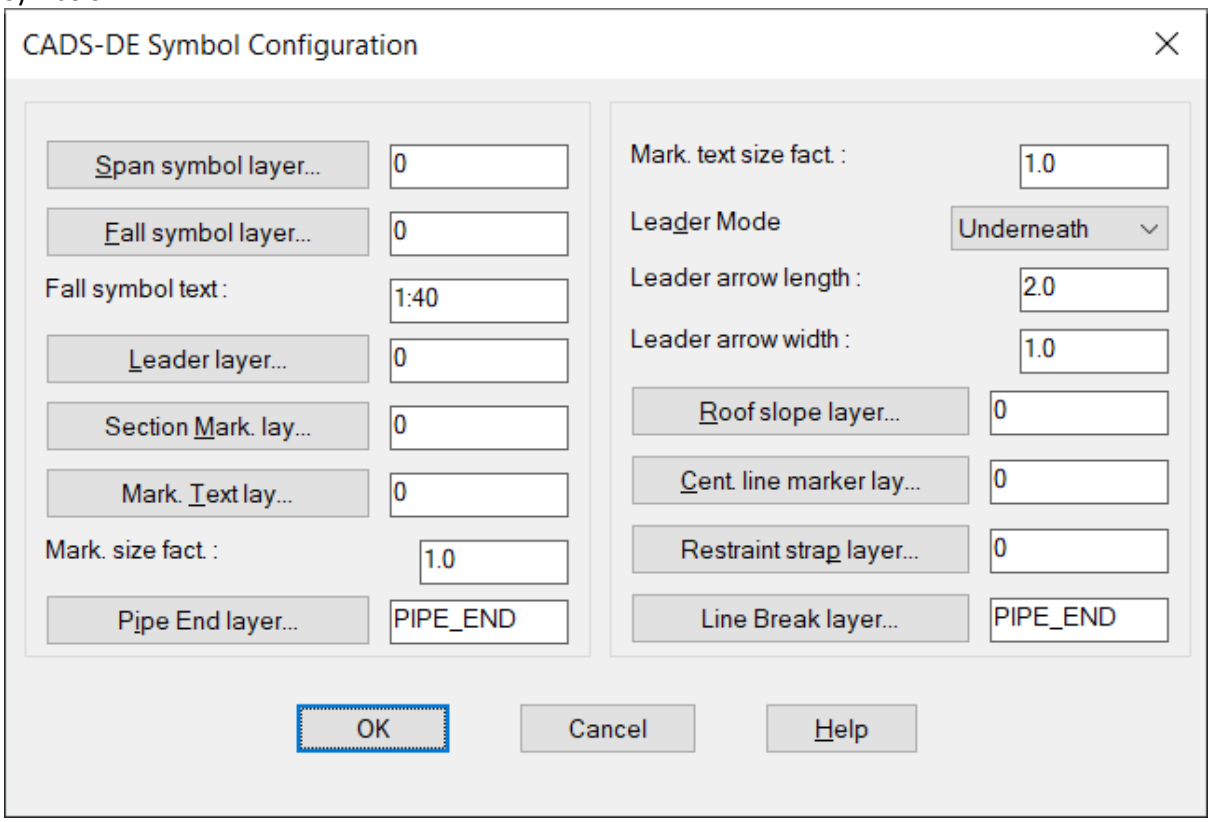
Switch the toggle button on (so it has a cross in it) to switch auto save on.

The timer interval field is the time between automatic saving.

The Drawing name is the name the current drawing will be saved to when the automatic save activates. This drawing name can only be changed through the AutoCAD configuration.

10.2 Symbols Configuration

The Symbolic Configuration allows the user to change various elements to do with the dynamic symbols.



10.3 Miscellaneous Configuration

10.3.1 Layering Tools

The 'Default Layer' is a layer that is automatically set whenever any of the AutoCAD draw commands (line, polyline etc..) are called from the pop down menus or tablet. The 'Default Layer' is set by the user in the Miscellaneous Configuration or by picking the Set 'Default Layer' command. It can also be switched On or Off in these two places.

If the 'Default Layer' is switched off then when the user picks any of the draw commands, they will be drawn on AutoCADs currently set layer rather than the 'Default Layer'.

There is also a 'Dimensions Layer'. This is the 'bed partner' of the 'Default Layer' and works in the same way, except that it is set when any dimension drawing commands are picked from the pop down or tablet.

These two tools were created to make it easier for the user to swap between drawing and dimensioning without having to go to the trouble of changing the layer each time.

The 'Temporary Layer' is a layer that is toggled on or off by the user, for temporary use. When the Temporary Layer command is picked, the 'Default Layer' takes the value of the 'Temporary Layer'. Therefore, when the 'Temporary Layer' is set, all entities drawn, that would have appeared on the 'Default Layer', will now appear on the 'Temporary Layer'. Selecting the command again will toggle it off and they will return back to being drawn on the 'Default Layer'.

An example of where the 'Temporary Layer' may be used is to do construction lines. The user can do some drawing on the 'Default layer', select 'Temporary Layer', do some construction line work, then select the command again to toggle it off and do some more non-construction drawing.

If the 'Default Layer' tool if OFF then selecting the 'Temporary Layer' command behaves slightly different. Instead of setting the 'Default Layer' to the 'Temporary Layer', it sets the current layer to the 'Temporary Layer' value. Selecting the command again toggles the previous current layer back.

10.3.2 Write Prototype Settings

Usually, the various Drawing Environment configuration values are saved to the drawing. Therefore, if you save the drawing, and load it up at a later date, you will still have the configuration values set as before.

Picking the 'Write Prototype Settings' button allows the user to save the various Drawing Environment configuration values to disk.

This means that whenever he starts a new drawing, he will automatically inherit the saved configuration values.