

CADVANCE V16 Release Notes

V16 contains all updates made for V15.5 and V15.6.

The V15.6 and V15.5 release notes are also contained in this V16 release notes.

Note:

If you have edited, i.e., customized your cadvance.ini file, please save it in a folder other than CADVANCE V16 folder, or rename it before the CADVANCE V16 installation.

And after the V16 was installed, edit cadvance.ini file to add/change as your customized cadvance.ini file.

New Features

DWG file translation

V16 can process DWG file to VWF file conversion and vice versa up to R2018 format.

V15.x processes up to R2013 format.

Improvements

MTEXT process improvement

V16 has improved to process MTEXT data which is contained in DWG file.

Reference Filename length

The reference file name length has been expanded from 64 characters to 256 characters.

There is a case that The whole reference file name length including folders' names exceeds 64 characters if the reference files(s) are located under several long name folders.

However, it is limited up to 64 characters for VWF to DWG file transcation. In this case, specify the output as Primitive instead of Xref.

Example of long file name:

#	Reference	Positioning		Wind
	File name Path	X Org Y Org	Rotate Scale	Left Top
1	COPY-PASTE.VWF	0"	0.00	0"
	C:\TODD\LONG FOLDER NAME 1234567890\LONG FOLDER NAME 2 1234567890\LONG FOLDER NAME 3 1234567890\LONG FOLDER NAME 4 1234567890\	0"	1.00	0"
2	1327 FRAMING PLAN.VWF	27'9"	0.00	0"
	C:\TODD\LONG FOLDER NAME 1234567890\LONG FOLDER NAME 2 1234567890\LONG FOLDER NAME 3 1234567890\LONG FOLDER NAME 4 1234567890\	27'9"	1.00	0"
3		0"	0.00	0"
		0"	1.00	0"
4		0"	0.00	0"
		0"	1.00	0"

CADVANCE V15.6 Update Notes

File Open Path

CADVANCE always keeps tracking the last accessed file type and the path.

File/Open dialog box shows the last accessed file type and the path.

Example:

Let's assume we have a file floor.vwf under a folder "c:\my drawings\project 1" for vwf files for project 1, and a folder "c:\my drawings\project 1\dwg files" for translated dwg files.

File/Open, VWF, Select Floor.VWF in a c:\my drawings\project 1

User may edit the drawing and save it,

Or open another vwf file from the folder.

File/Open

The File Open dialog box shows the path c:\my drawings\project 1, and File of type VWF.

Edit the drawing and File/Save As DWG into a folder c:\my drawings\project 1\dwg files

File/Open

The File Open dialog box shows the path c:\my drawings\project 1\dwg files, and File of type DWG.

User may want to bring up another dwg file,

Or if he/she wants to continue to work on the floor.vwf, then select it from the file list.

File

Select a file from the File List, Floor.vwf. Floor.vwf is displayed.

The file type and path change to a folder c:\my drawings\project 1, and File of type VWF.

When a file under File menu file list is selected (opened), the path and file type become the last used file type and path.

File/Open dialog box shows the file type and the path, and user can continue to work on the file(s) under the same path.

DDE

Initiates live links with other applications

Connect	Establishes a DDE conversation.
Update	Displays the properties of selected objects in the linked worksheet.
Highlight	Highlights the objects whose properties are displayed in the linked worksheet.
Disconnect	Ends a conversation with an Excel worksheet.
Clear	Clears the properties in the linked worksheet and removes the corresponding objects from the selection list.

DDE stands for Dynamic Data Exchange. The CADVANCE implementation lets you set up live two-way conversations between CADVANCE drawings and Excel® worksheets.

You can use DDE to display the properties of selected 2D or 3D objects in a worksheet. Changes you make to object properties in the worksheet are automatically reflected in the drawing, and changes made to the objects in CADVANCE are automatically reflected in the worksheet.

For example, during a DDE conversation, you can change the color of a primitive object by changing the color property for that object in the Excel worksheet. You can also move an object by changing the object's coordinates in the worksheet.

DDE/Connect

Establishes a DDE conversation

- 1 To establish a DDE conversation, run CADVANCE, and open the desired drawing. Run Excel and open the desired worksheet.

- 2 Choose Utility/DDE/Connect (ALT, U, D, C).

If a drawing is already connected to a worksheet, the worksheet is not listed in the dialog box.

- 3 Select OK.

An asterisk is entered in the worksheet's cell A1. It indicates that a conversation is active.

NOTE: Do not change the asterisk in cell A1. The cell is reserved for CADVANCE use. If you close the drawing or worksheet, a dialog box will ask if you want to abort the close. This allows you to cancel the close and save the drawing or worksheet.

DDE/Update

Displays the properties of selected objects in the conversing worksheet

- 1 To display object properties into Excel, start a DDE conversation with Utility/DDE/Connect (ALT, U, D, C).
- 2 Select the objects whose properties you want to display.
- 3 Choose Utility/DDE/Update (ALT, U, D, U). The objects' properties are displayed in the worksheet.

NOTE: This command removes the previously selected object's selected if any.

DDE/Highlight

Highlights objects in a drawing that are subjects of a DDE conversation

- 1 To see which objects in the drawing are subjects of a DDE conversation, choose Utility/DDE/Highlight (ALT, U, D, H).

Objects in the drawing that are subjects of a DDE conversation are highlighted.

The DDE highlight color can be different from the normal highlight color. You can set it in the CADVANCE.INI file in the [System] section under DDE Highlight = *color#*.

- 2 To add objects to Excel sheet.
Select object(s) with Shift Key pressed and choose Utility/DDE/Update.
Objects are added to Excel sheet.
- 3 To delete objects from Excel sheet.
Select object(s) with Control key pressed and choose Utility/DDE/Update.
Objects are deleted from Excel sheet.

DDE/Disconnect

Ends a conversation with an Excel worksheet

- 1 To end a conversation, choose Utility/DDE/Disconnect (ALT, U, D, D).

The asterisk is removed from worksheet cell A1 and the conversation is ended.

DDE/Clear

Clears the selection and properties in a conversation

This command clears the selection and clears the data from the spreadsheet but maintains the link.

DDE Operation

DDE operation procedure is described below:

We can save both CADVANCE VWF files and Excel file for later use.

If we don't need to save, then we can skip steps 4, 10, 11, 12.

1. Open Excel --- Blank sheet
2. Start CADVANCE (CV)
3. File/Open a VWF file (for example. Build2a.vwf)
4. Excel
File/Save As --- Name it to associate the vwf file for future reference.
For example Build2a_dde
5. Utility/DDE/Connect --- The excel file name is written to the vwf file.
The message box shows up. Select OK.
An * is written at the cell A1 in the Excel sheet.
6. Select CADVANCE object(s),
Utility/DDE/Update
The selected object data are written into the Excel sheet.

7. Utility/DDE/Highlight

The objects in the Excel sheet are highlighted.

8. Add objects to the Excel sheet.

Utility/DDE/Highlight

Select object(s) with Shift Key pressed.

Utility/DDE/Update

The selected object(s) with Shift Key pressed are added to the Excel sheet.

9. To delete objects from the Excel sheet.

Utility/DDE/Highlight

Select object(s) with the Control key pressed.

Utility/DDE/Update

The selected object(s) with the Ctrl key pressed is deleted from the Excel sheet.

10. File/Save or File/Save As the VWF file (for example, build2a_dde.vwf)

11. File/Close, OK

12. Excel - File/Save

CADVANCE and Excel update

- When CADVANCE objects are updated in the Excel sheet and edited/changed in the CADVANCE, Excel data are updated accordingly.

- When data in the Excel sheet are changed, CADVANCE objects are updated.

Note: Data types that can be changed in the Excel sheet depend on the type of data.

For example, properties of primitive objects can be changed on the Excel sheet and it reflects the updated data on CADVANCE drawing.

CADVANCE V15.5 Updates Release Notes

CADVANCE V15 Release version contains up to Build 15.06.

This update note explains Build 15.07 through Build 15.50

Build 15.07 through 15.50

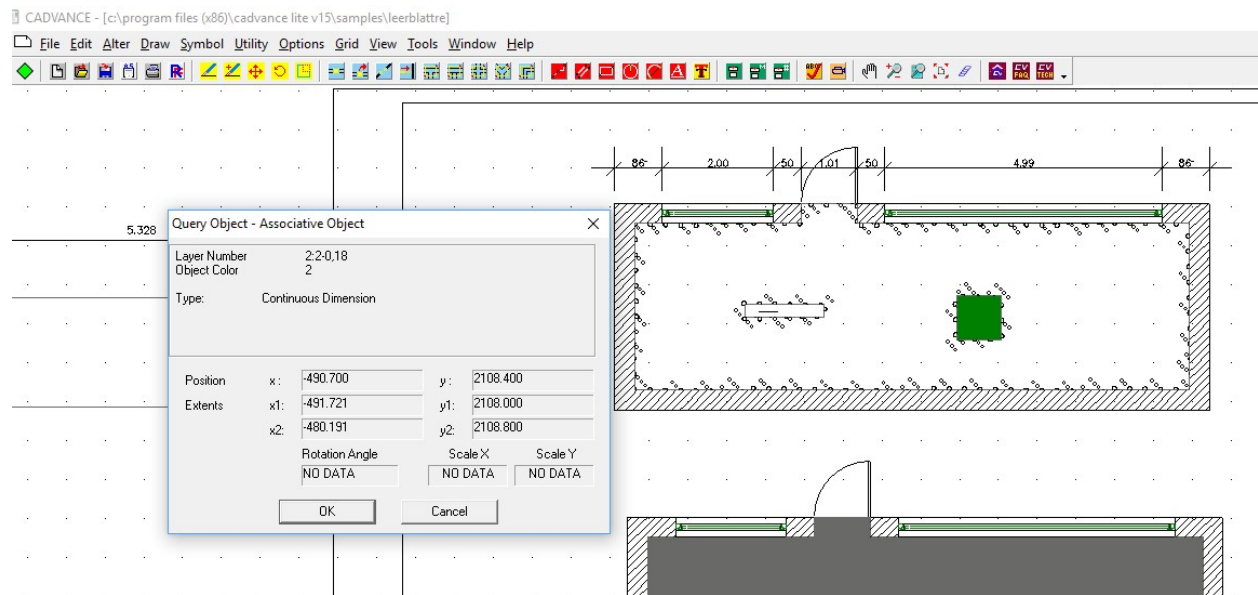
New feature only for LiTe version.

LiTe version can display vwf files with an associative dimension which was created by CADVANCE Full version and can save it with the associative dimension data.

However, VWF to DWG translation doesn't output such assoc. dim. data to DWG file. So, the LiTe version must change the assoc. dim. data to static to output it to dwg file.

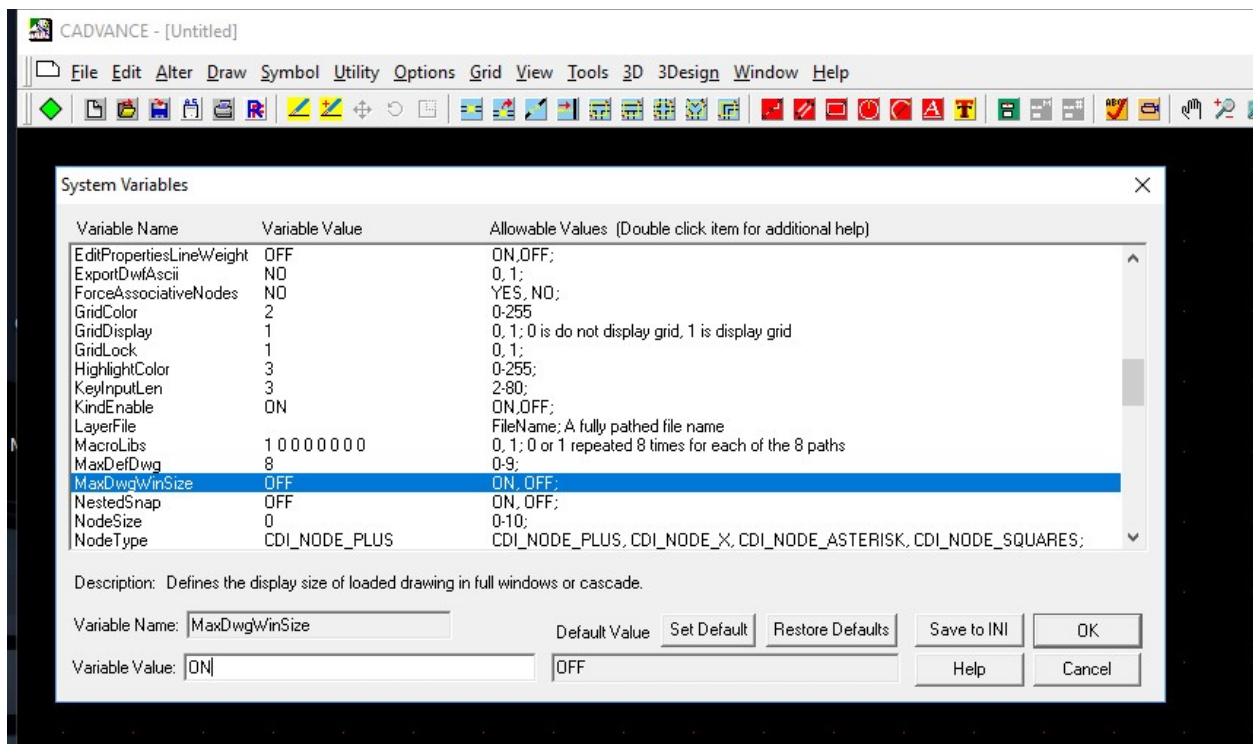
CADVANCE.MNU update:

MENUITEM "Assoc. Dim. Ma&ke Static", IDM_EDIT_ASSOC_MAKE_STATIC
is added under Edit menu in cadvance.mnu file.



New features:

1. Added "MaxDwgWinSize" to **Option/System Variables** menu.
When it is ON, all loaded drawings are displayed in full view in CADVANCE window.
To see/display a pre-loaded drawing, you have to select it from a menu Window.
OFF is as now in cascade view.



Or user can edit cadvance.ini file under [system].

Change “MaxDwgWinSize=ON” as below and save the ini file.

LayerFile=

MaxDefDwg=8

MaxDwgWinSize=ON

NestedSnap=OFF

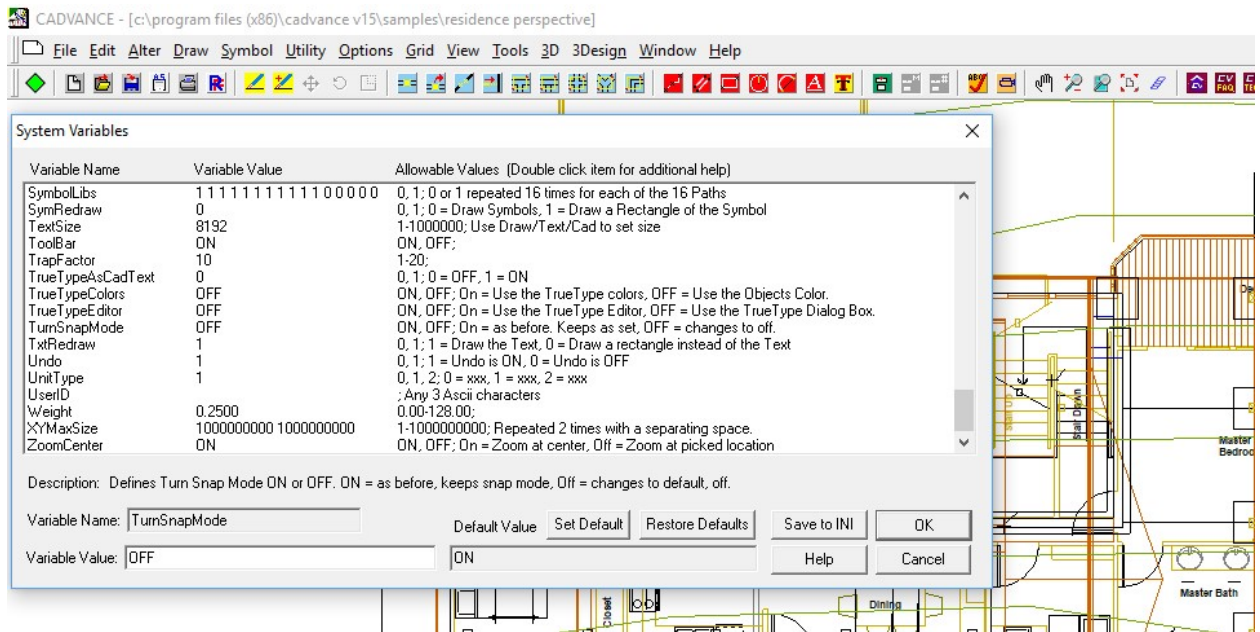
NodeSize=0

- Added “Turn Snap Mode On/Off” to **Option/System Variables** menu.

Added a line “TurnSnapMode=ON” to cadvance.ini under [System] section.

ON: as now. As set.

Off: When asked “Turn Snap Mode Off, Yes or No”, and select Yes, the snap mode changes to default “off” and stays off.



Or, edit cadvance.ini file under [system] as below:

SelectionWindow=0

TurnSnapMode=OFF

ToolBarCreationOption=ON

ZoomCenter=ON

Example:

If Turn Snap Mode is set OFF, the following is the operation theory.

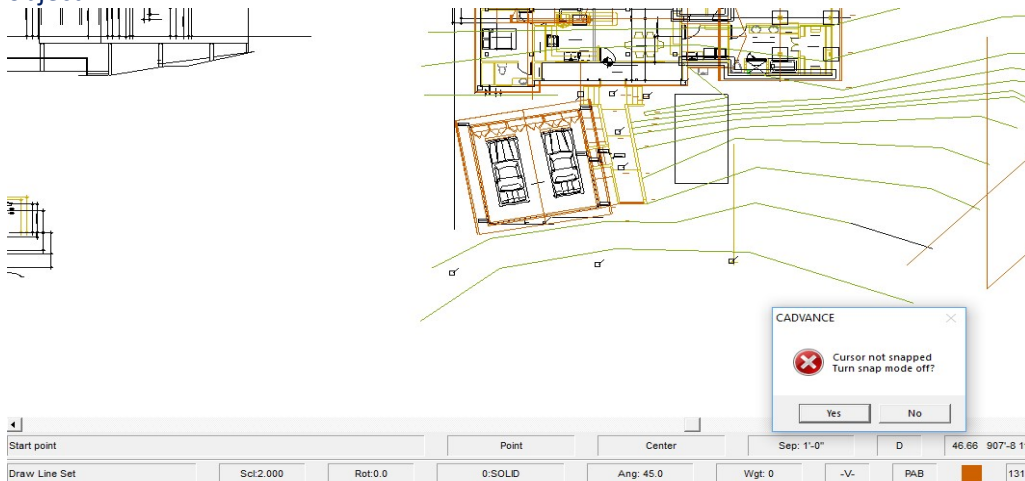
Let's say, you set snap mode to Vertex and you will see "-v-" in the 2nd status line at the bottom of CADVANCE window.

You will pick vertex of objects to draw a few times.

If you pick/select non-vertex, a message box "Cursor not snapped. Turn snap mode off? YES|NO" will be displayed.

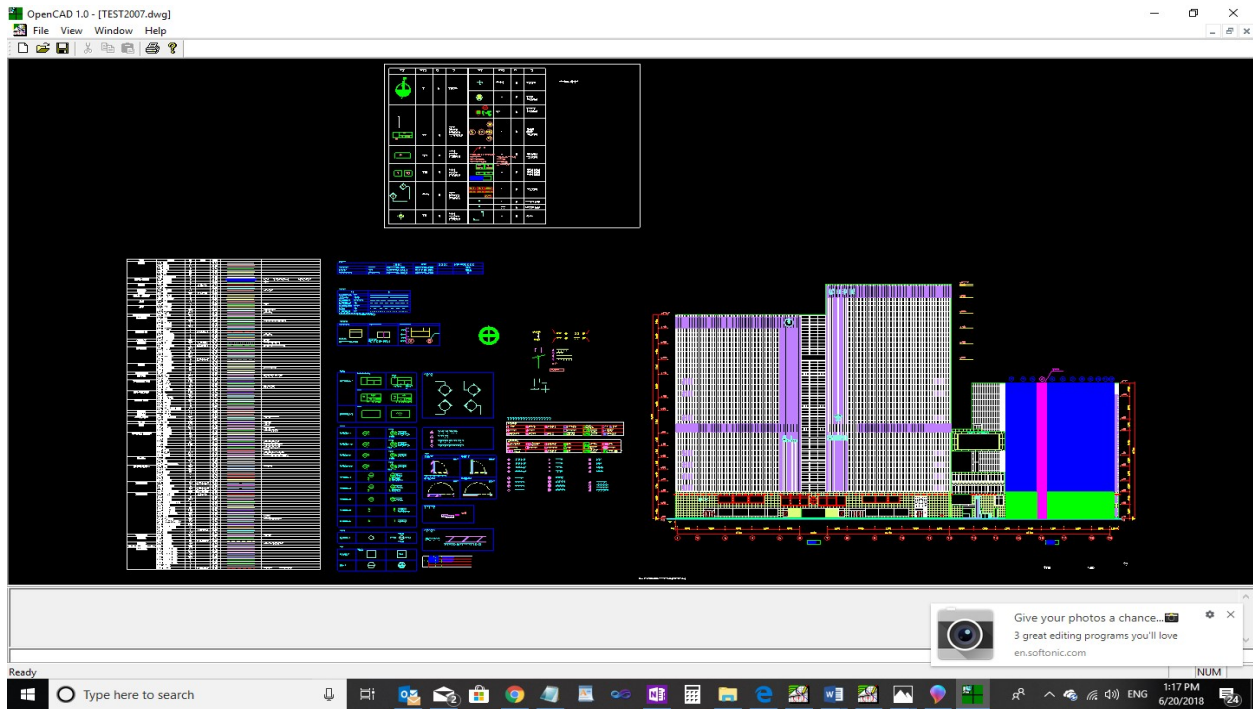
If you select YES, the status line snap mode changes to "---" (off) and stays off.

If you select NO, it stays as vertex snap mode and CADVANCE expects you to pick vertex of object.

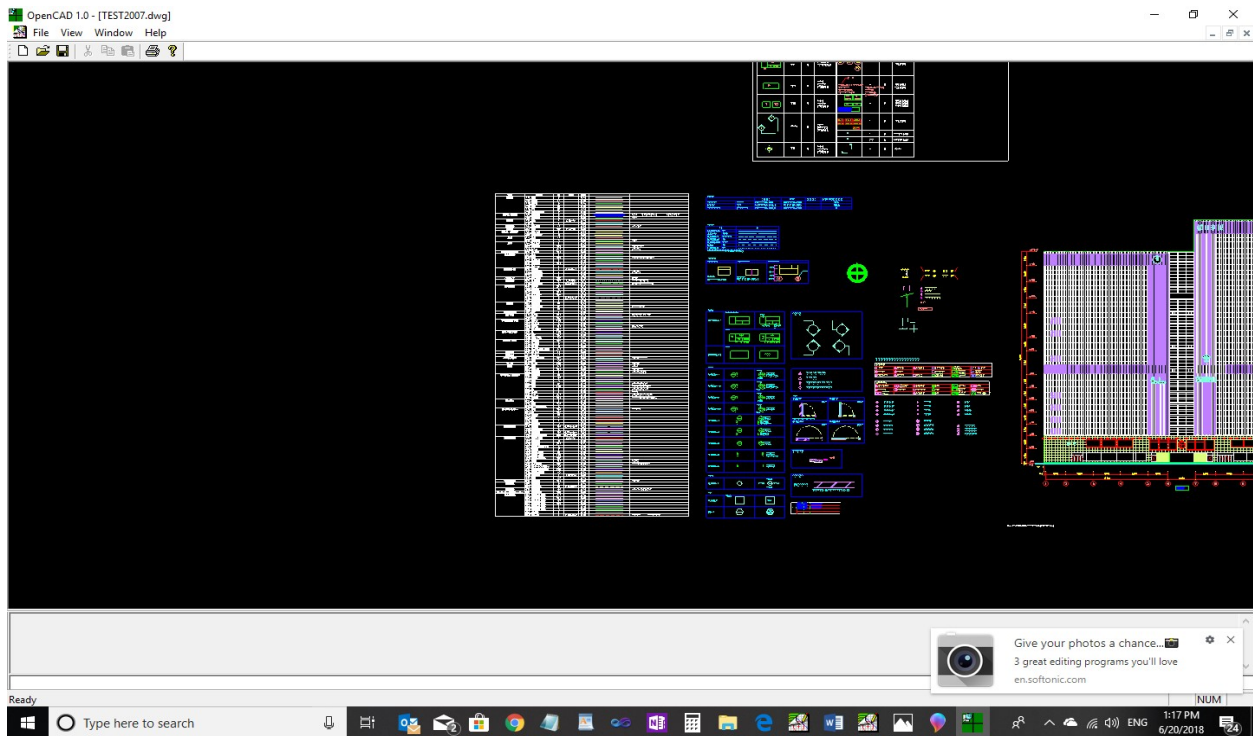


3. New **DWG Preview** program with zooming and panning functions.
Added PREVIEW box in File/Open dialog box in addition to Tools menu.

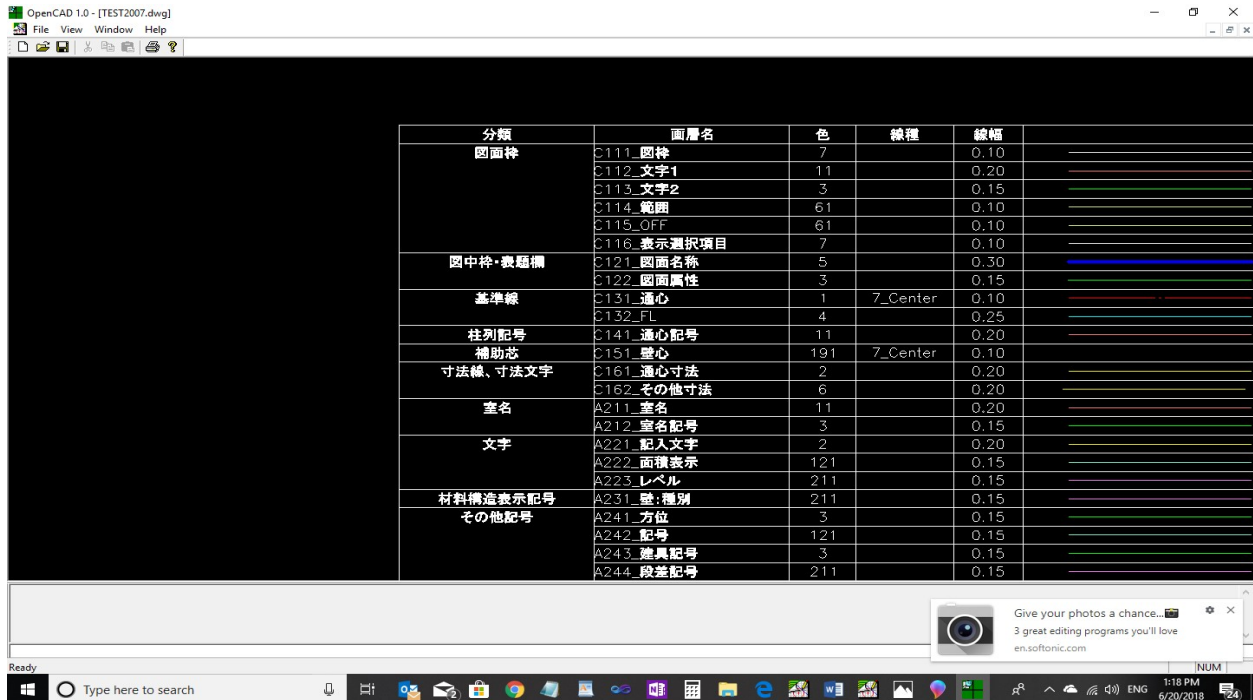
View Fit



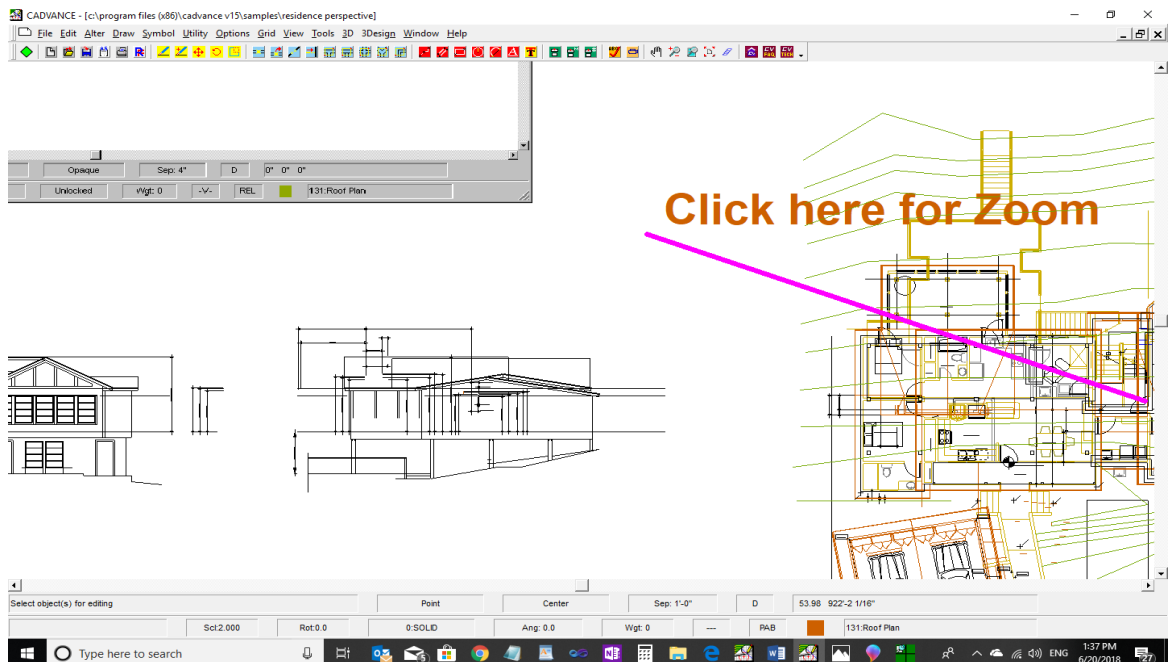
View - Pan



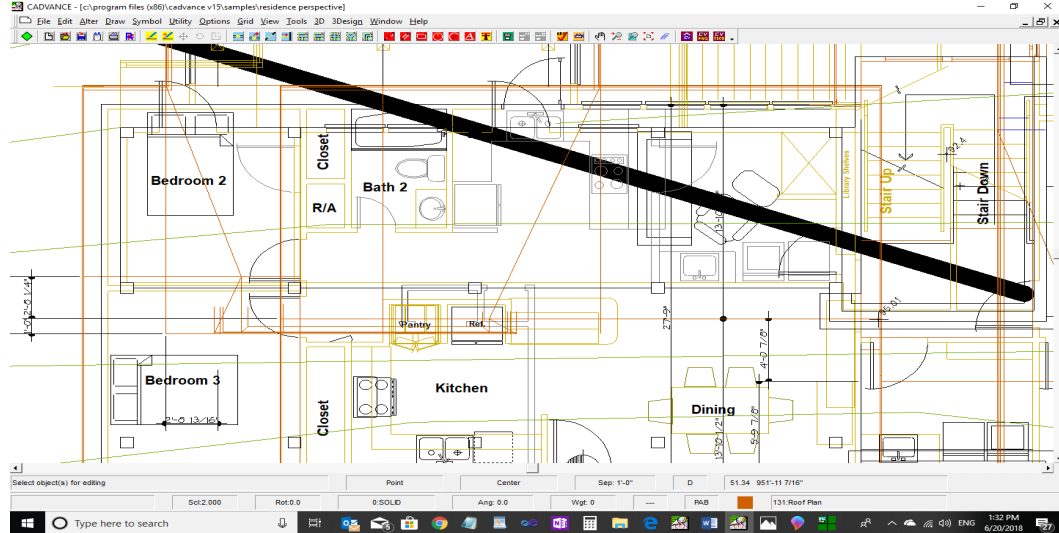
View - Zoom



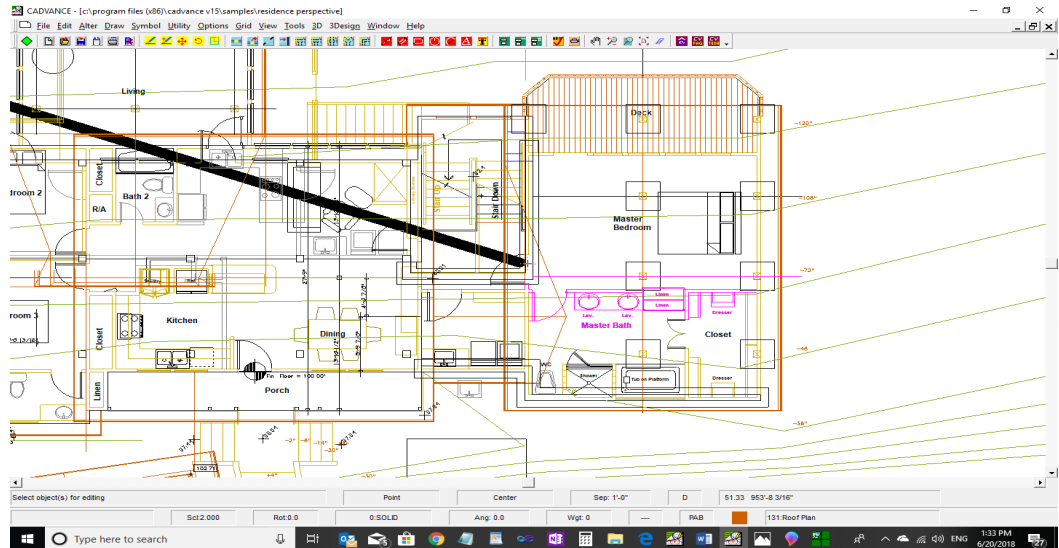
- Added “ZoomCenter On/Off” to **Option/System Variables** menu.
 Added a line “ZoomCenter=ON” to cadvance.ini under [System] section.
 ON: Zoom at the center of drawing.
 Off: Zoom at the cursor position. Needs PAN to move the zoomed-up area.



ZoomCenter = Off



ZoomCenter = ON



Improvements:

1. Symbol/Count update
Increased number of symbols from 132 to 1024.
A menu Symbol/Counts shows a table with symbols names and the counts.
The limit was 132 symbols.
Now it shows up to 1024 symbols.
2. DWG to VWF translation Font name string improvement.
CADVANCE vs DWG font mapping table is set up with input DWG defined fonts and CADVANCE fonts during the translation process. If unknown font name is input, CADVANCE makes a default font mapping.

There is a case that ACAD 3rd party software adds/uses its own special font which is a file located in a drive C, D, or E and under several folders. And thus the text string to represent the font results in longer than 80 characters.

When such input font string with longer than 80 characters is read into memory and compared with font names that CADVANCE recognizes, somehow Microsoft C library routine causes an error and thus CADVANCE results in a crash.

Now CADVANCE DWG translator checks such font name string and uses CADVANCE default font instead.

3. To process (display) vwf file with bad left-over associative dimension data.

We don't know why such bad associative dimension data are left with the saved VWF file, but with this improvement, such VWF files are displayed properly.

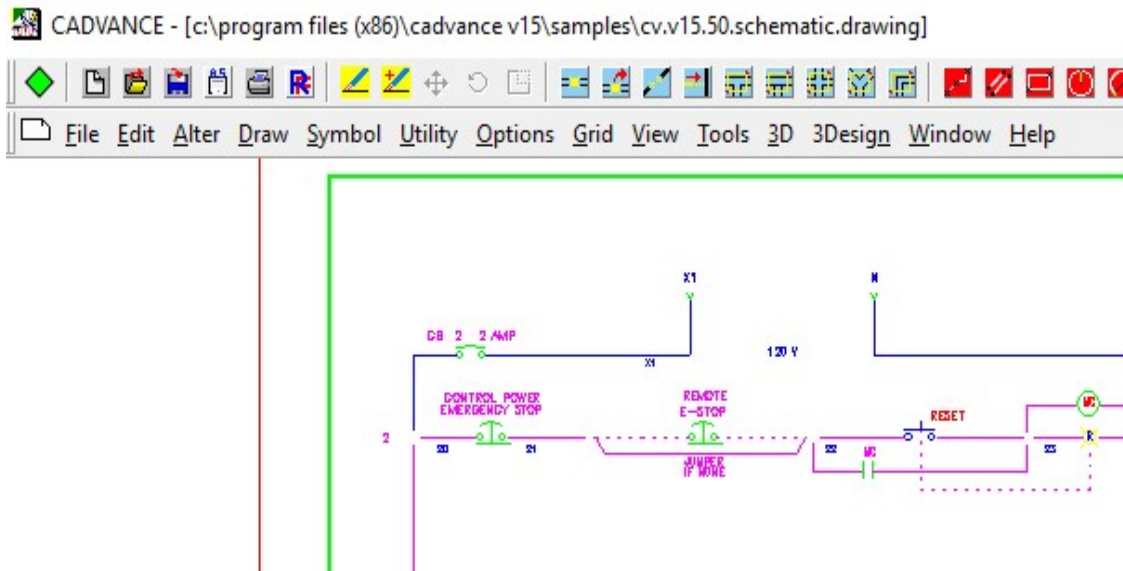
In CADVANCE V15, File/Save and File/Save As automatically run "PACK" data before saving data to vwf file. PACK eliminates bad data.

4. Filename at the top of CADVANCE windows shows all file name string except the extension.

Example:

If the file name was cv.v15.50.schematic.drawing, only CV was displayed at the top of CV window.

Now it displays cv.v15.50.schematic.drawing at the top.



5. Menu Tools/Erase bug fix.
Eliminated the crash and it works as expected.
6. Options/Drawing
The password problem has been corrected.