

CADVANCE V15 Updates Release Notes for Users

CADVANCE V15 Release version contains up to Build 15.06.
This update notes explains Build 15.07 through Build 15.50

Files that are updated and loaded into CADVANCE V15 folder:
cadvance.mnu (only for CADVANCE LiTe V15)
cad1.dll, dwgxvwf.dll, cv_assoc.dll, mfcddl.dll, cv_preview.exe
cadvance.ini

Note:

If you have edited, i.e., customized your cadvance.ini file, please save it in a folder other than CADVANCE V15 folder, or rename it before the CADVANCE V15.50 installation.

And after the V15.50 was installed, edit cadvance.ini file to add/change as your customized cadvance.ini file.

Build 15.07 through 15.50

New feature only for LiTe version.

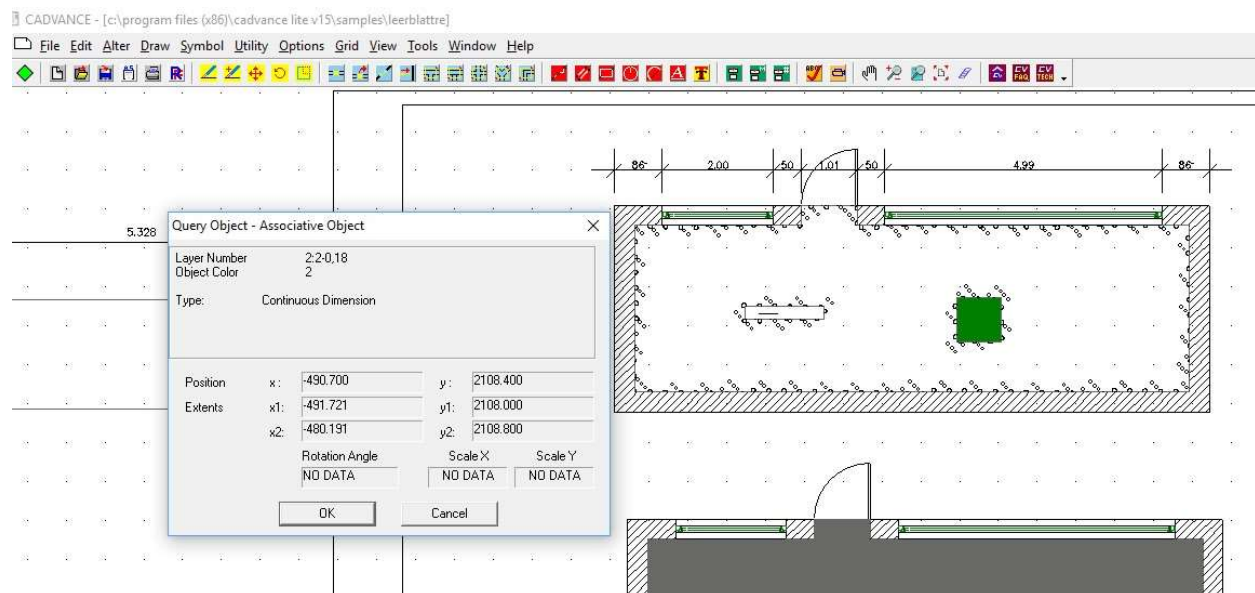
LiTe version can display vwf files with associative dimension which was created by CADVANCE Full version and can save it with the associative dimension data.

However, VWF to DWG translation doesn't output such assoc. dim. data to DWG file.

So, LiTe version must change the assoc. dim. data to static to output it to dwg file.

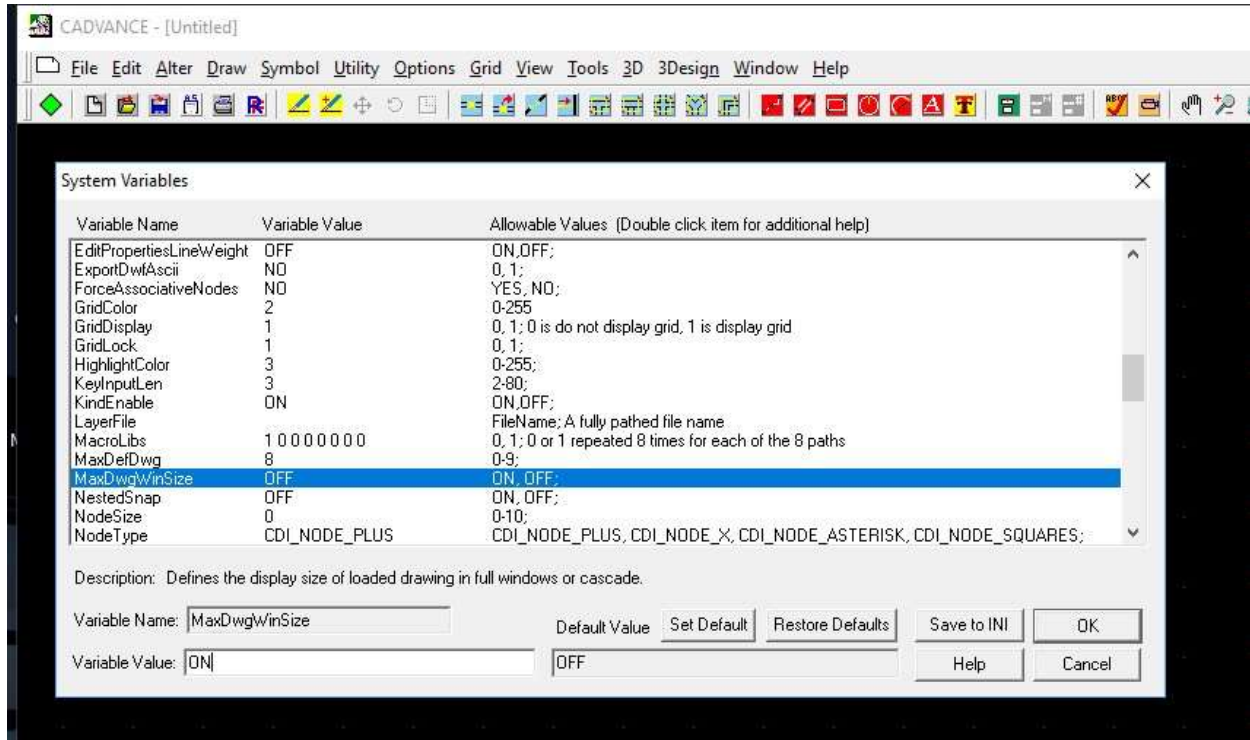
CADVANCE.MNU update:

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=

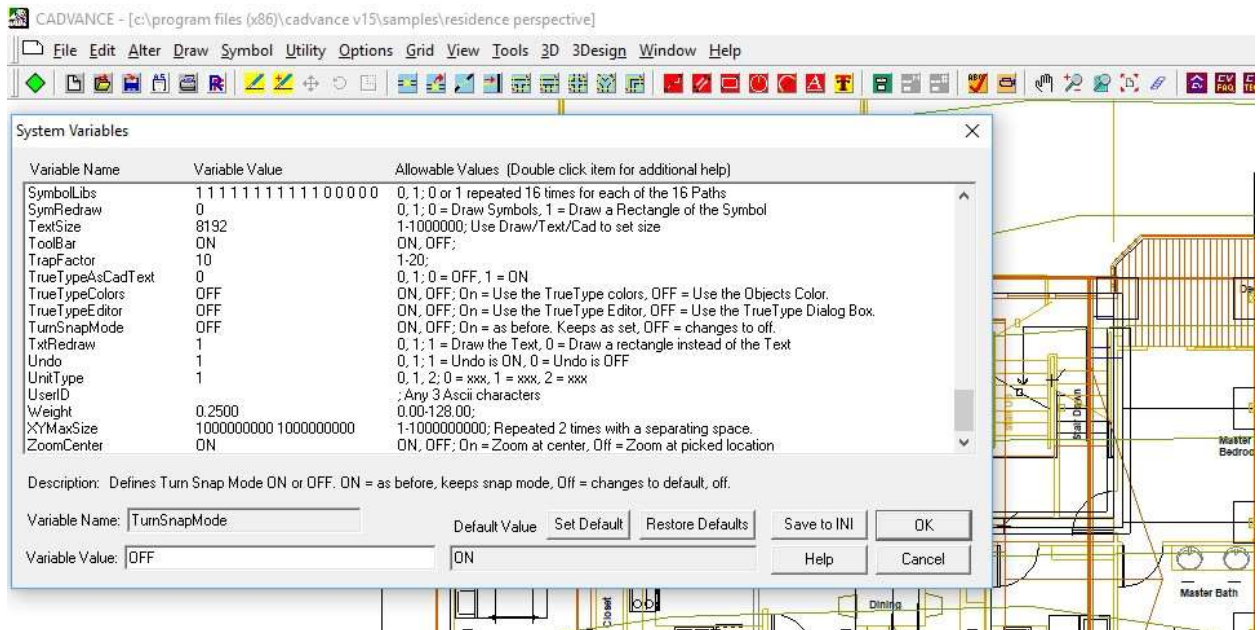
MaxDefDwg=8

MaxDwgWinSize=ON

NestedSnap=OFF

NodeSize=0

2. 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.

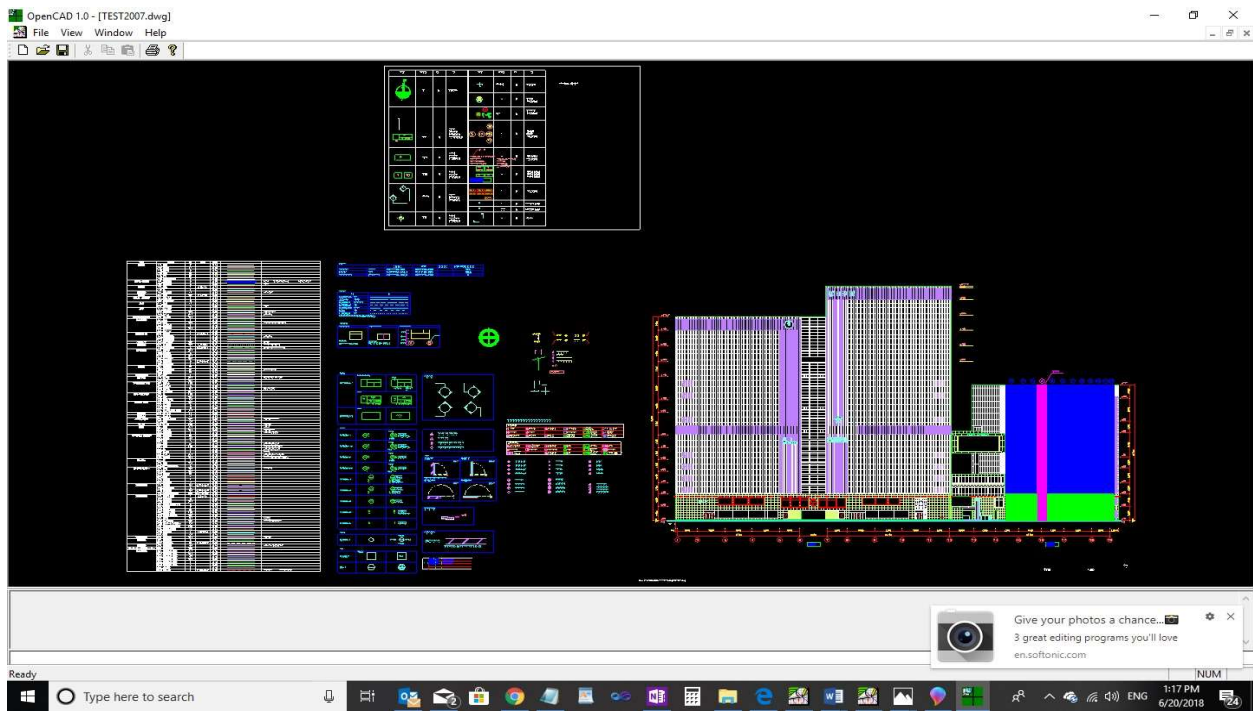
If you select YES, the status line snap mode changes to "---" (off) and stays off.

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

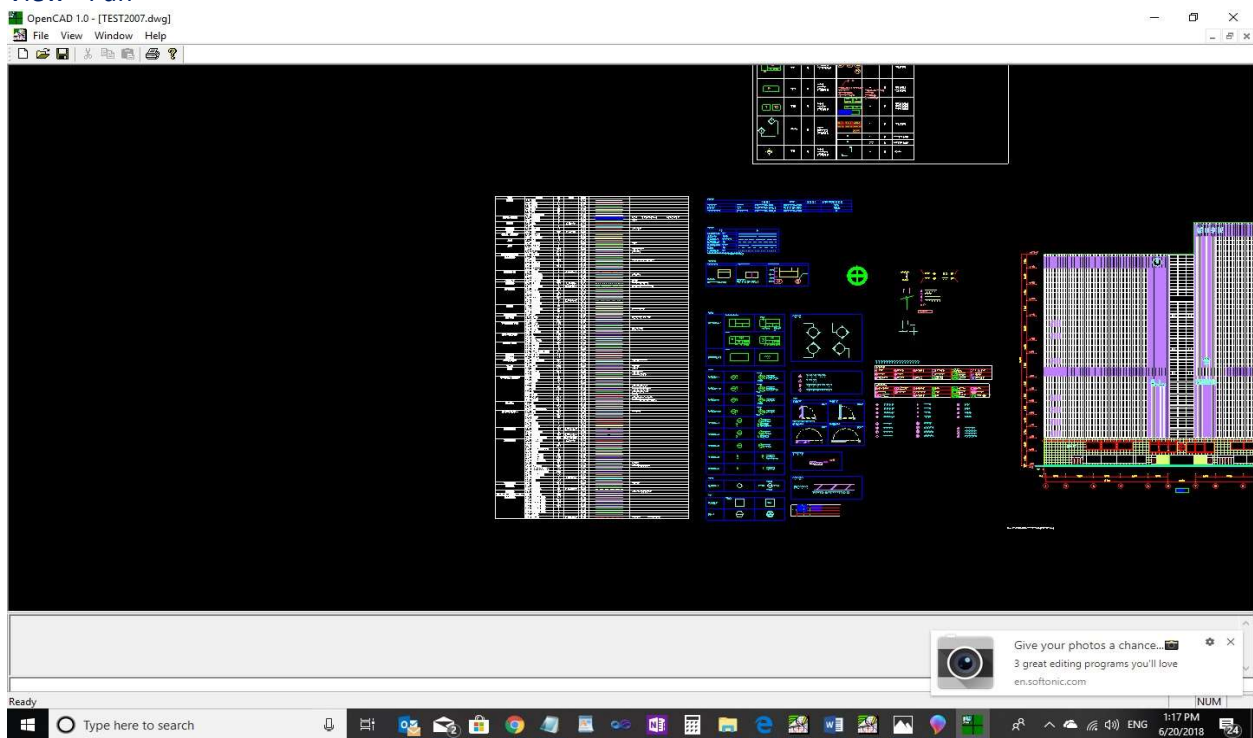


3. New **DWG Preview** program with zooming and panning functions.
Added PREVIEW box in File/Open dialog box in addition to Tools menu.

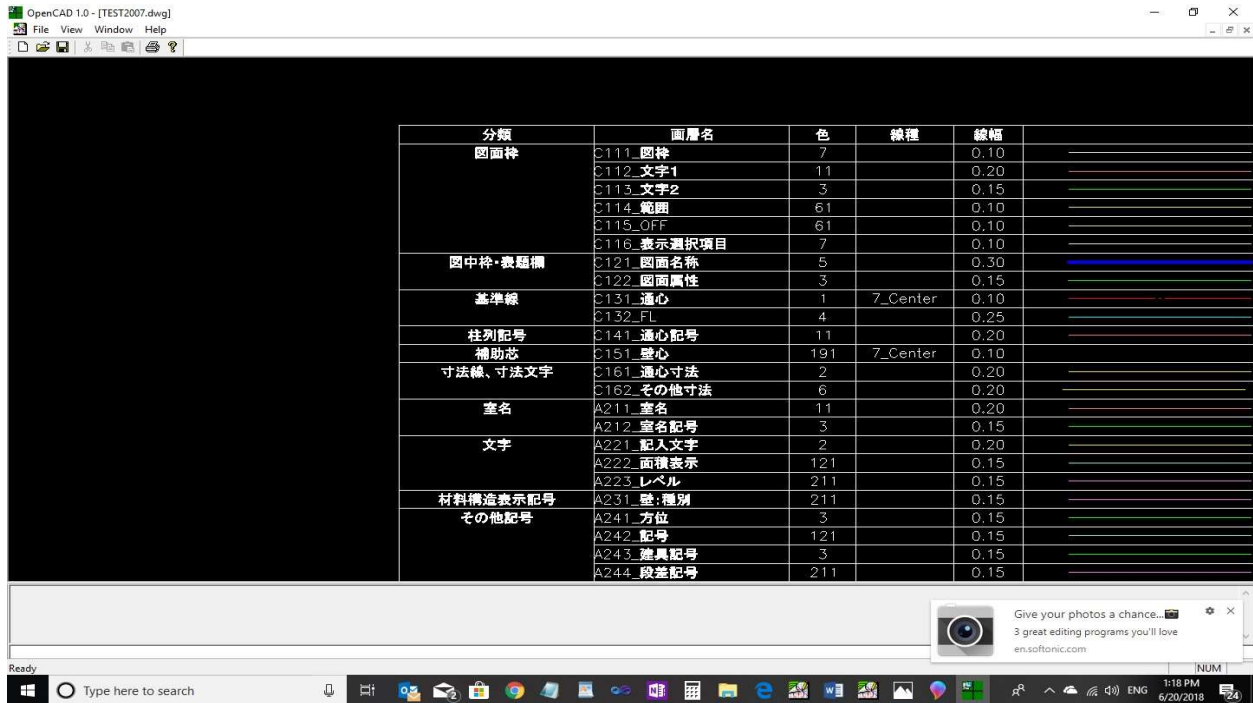
View Fit



View - Pan



View - Zoom

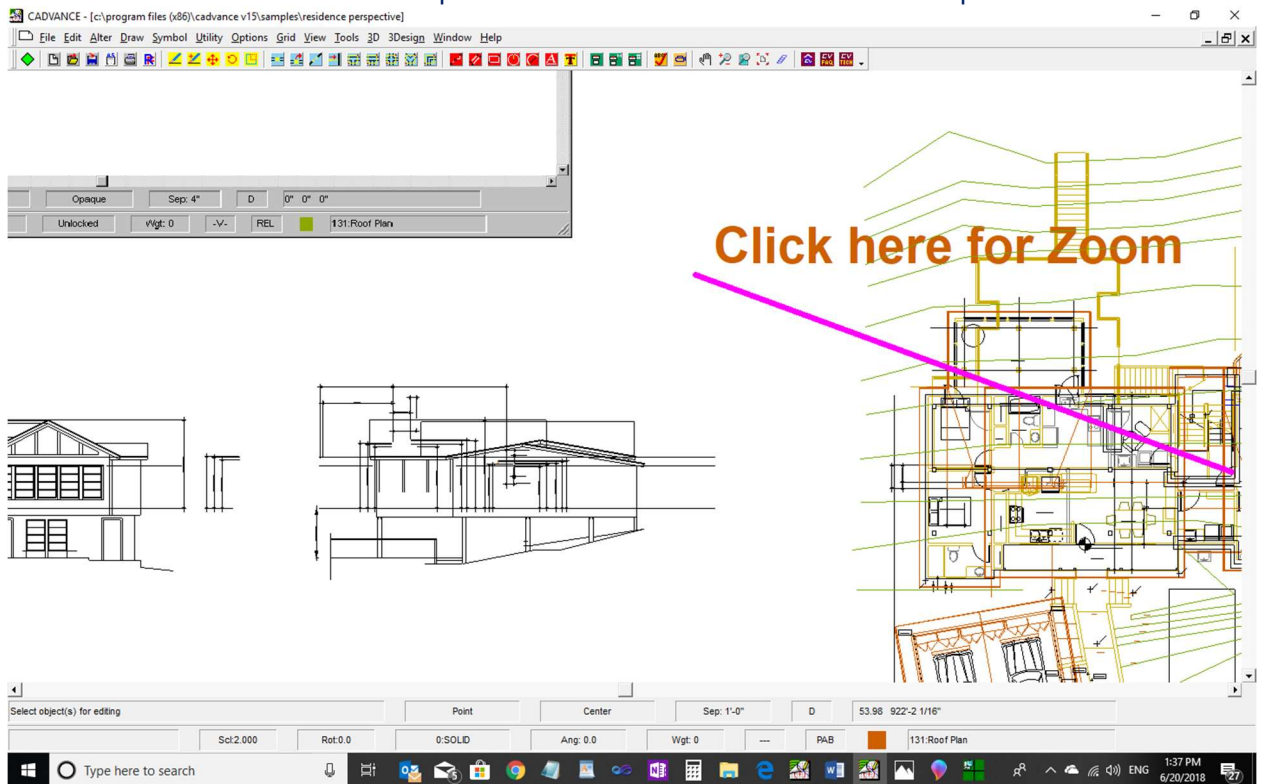


4. Added "ZoomCenter On/Off" to Option/System Variables menu.

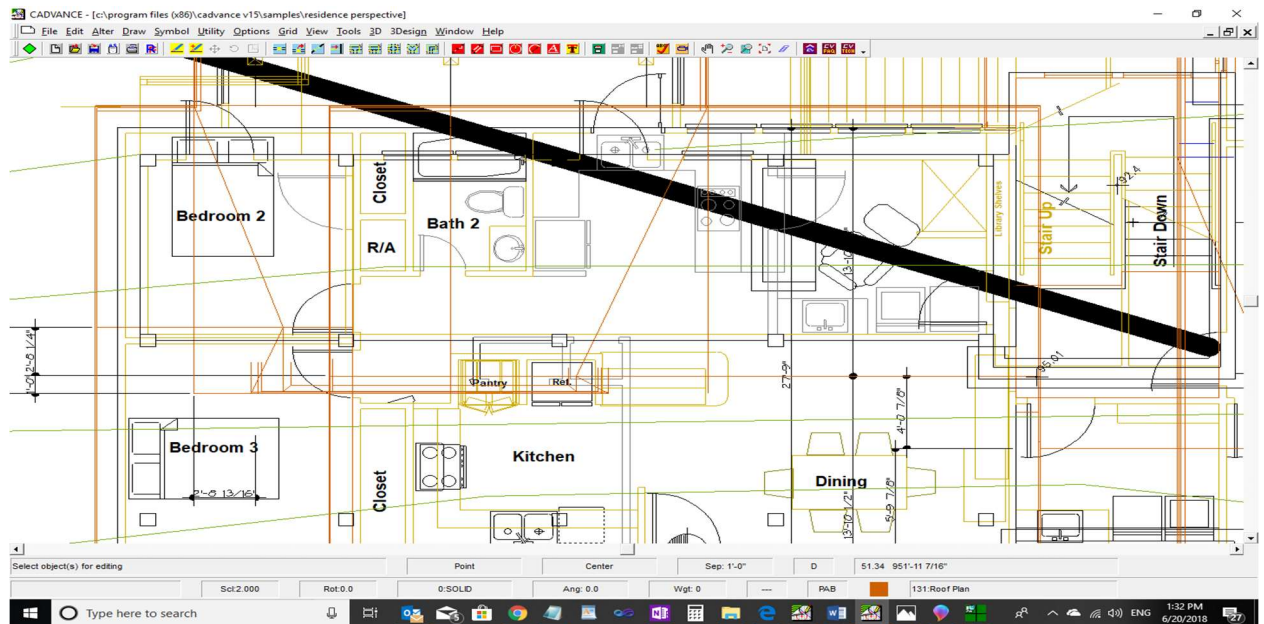
Added a line "ZoomCenter=ON" to cadvance.ini under [System] section.

ON: Zoom at the center of drawing.

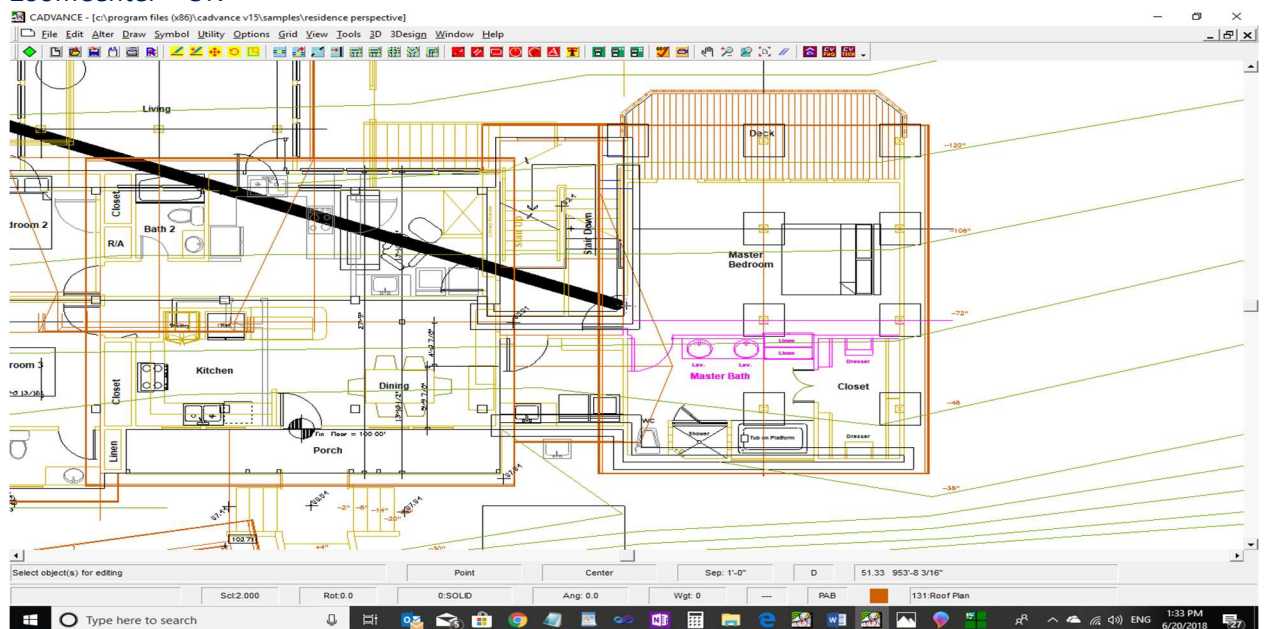
Off: Zoom at the cursor position. Needs PAN to move the zoomed-up area.



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 crash.

Now CADVANCE DWG translator check 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 left with the saved VWF file, but with this improvement, such VWF files are displayed properly.

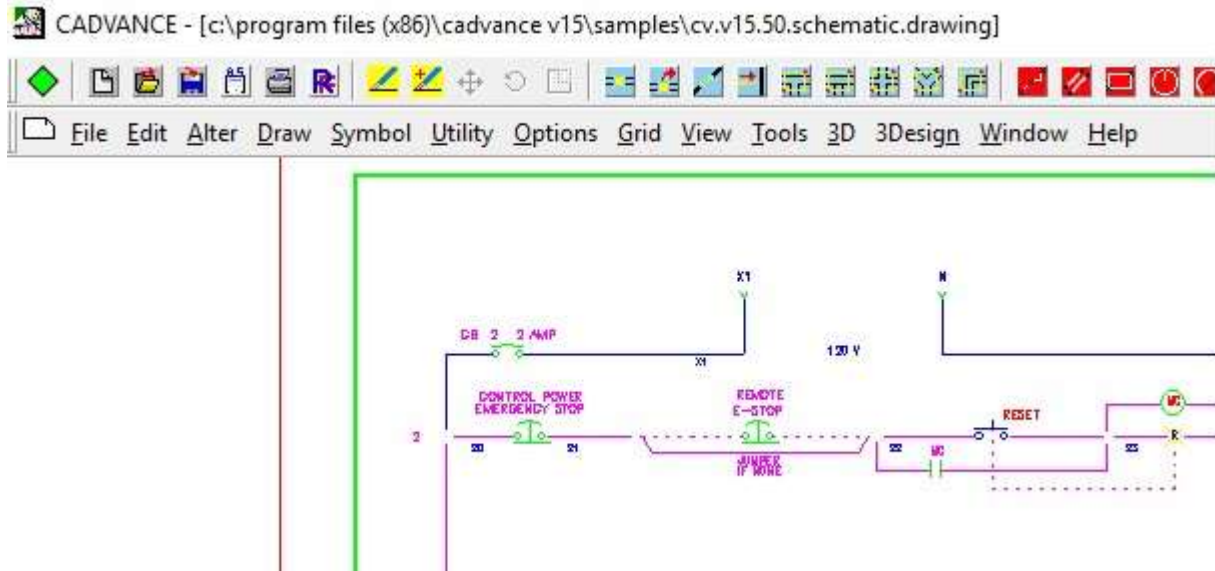
In CADVANCE V15, File/Save and File/Save As automatically run "PACK" data before saving data to vwf file. PACK eliminates bad data.

4. File name 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
Password problem has been corrected.