

## FREQUENTLY ASKED QUESTIONS

return to: <http://www.padeepro.com/padeeing.html>

Updated: 05/23/2023

Frequently Asked Question performed by PADEE's users, if you do not see your question here, please send it by mail [jorgematheus@gmail.com](mailto:jorgematheus@gmail.com)

### INDEX

1.	What Software and Hardware is required to run PADEE .....	2
2.	In COMMASW: Another person have enter part of the materials on his computer, my question is? If I can, with pen drive or shared network drive, bring that information to my computer and keep working on that; the procedure is normal or is there a special way for the system? .....	2
3.	How can i change the material prices quickly using scaling formulas? .....	3
4.	Is ComWIN is multiuser? .....	3
5.	ComWIN computations make construction units duplicate and it cost double. How do I fix this?.....	4
6.	In the PNAP module, after identifying it says: Don't have ORDERING, what should I do? .....	5
7.	ANALYZING: When I run the PANP ANALYZE option, sometimes shows the following: READING RESULTS: NOT COMPLY, KVAINT: 7990 DEM: 8808 3730KVAR ... What does this expression means?.....	9
8.	One of the two error messages that gives the program after running it is that the circuit has no data, why does this?, if the data were inserted by selecting the first line coming out of the S/S.? .....	9
9.	What does this Message means: "Substation's name was not found in Step 2 for the feeder xxx .....	9
10.	When I put capacitors banks the result is equal to the previous one. (Without capacitors).....	10
11.	I have a substation 115/34.5 KV, Mumbai, which feeds another substation 34.5/13.8 kV (Hospital), It is required to run this 34.5 kV line in PADEE, but the program gives me error because it sees the substation Hospital as a source instead as a load. How can I do? this (See next figure).....	10
12.	In PANP module IDENTIFY option.. I get the error: Failed to open the file C:\PADEE\TEMP\CARNOD.DAT. How can I fix this? .....	12
13.	In IOP module, when I do an operation or maneuver, the program gives me an error and keep the switch in the original position (as it was). .....	12
14.	What is for the option ASSIGN LOAD? .....	13
15.	When the program runs IOP gives the message "Error in BUSORD" .....	13
18.	When PADEE is run the message "ERROR Writing No 2" is presented. What does it mean? .....	15
19.	UPDATING tells me I can not connect to the FTP server .....	16
20.	SGMAP no "Google Map" image is presented .....	16
21.	SGMAP "Google Map" display a SCRIPT error u other message .....	16
22.	When the PNAP is running the "exceeded the maximum number of nodes" message appears .....	16

23. When I'm Identifying the message is not in line z = 0 appears. How to solve this?.....	17
24. How does PADEE simulate the "Distributed Generation" ?.....	17
25. When I'm updating, I get the error "Can not connect to the FTP matmor.cloudns.org or 192.168.10.12" .....	18
26. To install the new PADEE 2023 version, should I uninstall the older version first? .....	19
27. After finishing setup NO PADEE menus appear in Autocad.....	19

## 1. What Software and Hardware is required to run PADEE

Currently PADEE needs Windows preferable version 10 or 11, and Autocad (2006-2023) or Bricscad (V23) or ZWCAD (2023).

The hardware required is depending on MAP sizes. A good computer must open the maps in CAD and allow to work with them without any delay. Actually, any commercial computer with Windows 10 or 11 can handle a city without any problem, but remember CPU speed and RAM resources allow to work faster in CAD. Please see details in our web site: <http://padeepro.com>

## 2. In COMMASW: Another person has entered part of the materials on his computer, my question is? If I can, with pen drive or shared network drive, bring that information to my computer and keep working on that; the procedure is normal or is there a special way for the system?

If you are already working as stand-alone user, the easiest way is; to manually copy the databases found in the folder c:\ComWIN and are listed below:

- MAT\_XXX.DBF
- UNI\_XXX.DBF
- MAUC\_XXX.DBF
- PRAR\_DAT.DBF
- 

Where XXX is the short name entered at the beginning of the program. For example, DAT

You should also delete all \*.IDX files. These are automatically generated when you start the program ComWIN

If you are working with prices unit must follow the same procedure with the following databases:

- PER\_XXX.DBF
- EQU\_XXX.DBF
- MUPU\_XXX.DBF

- PARU\_XXX.DBF

### 3. How can I change the material prices quickly using scaling formulas?

To change materials prices, labor and equipment quickly, please use the COMMASW program.

There are two forms:

- a) When a new price area is created, the program asks if user want to use a previous area multiplied by a constant factor.
- b) In each of the menu there is an Export option. With this option you can export the databases to EXCEL and this program can sort by material type and multiply it price with individual factors or materials group.

Be especially careful, and please follow the next suggestions:

- a) Create a backup before you begin.
- b) If you are creating a new area. Create it before exporting
- c) Do not change the size and order of the columns. Users may use more columns to the right side of the existing columns to perform calculations and then copy with the option of "values only" in the area corresponding to the price column, these additional columns will not be taken into account.
- d) When saving changes in Excel, please select the option to keep the current format.
- e) The spreadsheet is automatically deleted after updating the database. IF you want to keep the spreadsheet for future revisions it may be copied with another name before importing.

### 4. Is ComWIN is multiuser?

Yes, it is. The ComWIN program must be installed on a client machine and then copy the folder to a directory named ComWIN on the server. On the server user must create a shared folder with a letter assigned for example w:\ComWIN

On all the client's computer the program must be installed to introduce the access key but then you can delete the executable and make a "direct access" to the program file in the server. You can edit existing shortcuts and change where it says c:\ComWIN to w:\ComWIN.

You can share the fixed data (materials and labors) or the "Hard Disk Data" but not the works files (several units needed to perform the work). The works files can be also created on the server (for easy backup purpose), but in a different folder for each client, or different hard drive or pen drive.

When the COMWMAS is used in multiuser mode, other users can not use the database files: materials, construction units and equipment with the same short name. (le:DAT)

5. [ComWIN computations make construction units duplicate and it cost double. How do I fix this?](#)

At the beginning of ComWIN databases in the folder C:\ComWIN are combined with the database in the folder WORKS.

These databases are called exactly the same but they are in different folders. The idea is to keep the database updated and locked in application folder ComWIN and if any particular work requires a different construction unit or change the content of one construction unit this change should not affect other works and this new construction units or materials are created in the WORK folder. Work database's folder should be empty is no new material or cost is entry for a particular work.

If by mistake the database in c:\ComWIN is copied in the folder works. At the start of the application the two databases will join materials and constructions units producing duplicates.

To solve this problem, you can proceed as follows;

- Delete all files that have .IDX extension. If you can not see the extensions, check the folder options to see the extension of all files.
- Open a new folder where works files will be located
- Copy files that begin with "P" and "U" followed by the code of the work  
DO NOT COPY THE MAT\_XXX, UNI\_XXX, etc. FILES
- Run the ComWIN application and change the working directory to the new location. (IT will create the files again materials and units)
- Ready.

If after this procedure material or construction unit are missing, check the folder of previous works and entry the data again. If many units are missing materials and proceed as follows:

- Change the in the work folder the name of the original databases MAT\_DAT,

UNI\_DAT, etc to a new shot name for example MAT\_DA2, UNI\_DA2, data etc.

- Start ComWIN please change the data base short name to DA2 and when It ask to create files; say NO and when it ask to working with WORK data; say YES
- Finally export the materials and construction units to Excel. And import then in the original short name .DAT with a similar procedure as described in question 2.

6. In the PNAP module, after identifying it says: Don't have ORDERING, what should I do?

Problem with ORDERING, is because there is a problem with the identification procedure. Please check feeder logic or topology, following the nodes and section.

User must verify the drawing, looking for:

- Repeated sections
- Misplaced switches or disconnectors.
- Loops or rings.
- Others will be explained in the following paragraph

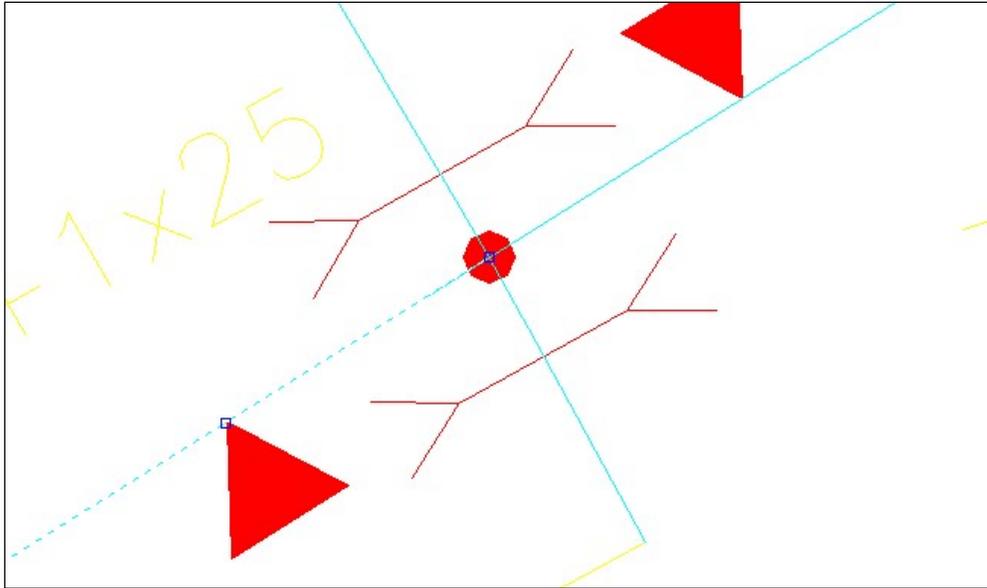
**Repeated or overlap sections:** When making the drawing, some times one line is drawn twice or is over another. The detection could be done opening feeder by parts to achieve isolating and identifying the part with problems.

You should look for sites where several identifications numbers look very closely each other and touching the lines with the mouse to verify each lines blue dots or "GRIPS".

If the numbers of nodes and sections font are very large and are mounted on top of each other, you can reduce the font size using the Utility Macro-> SCBL. Touch a yellow number and enter, type 0.2 and then type ALL. The same procedure is performed with blue numbers.

In the next figure an overlap sample is presented.



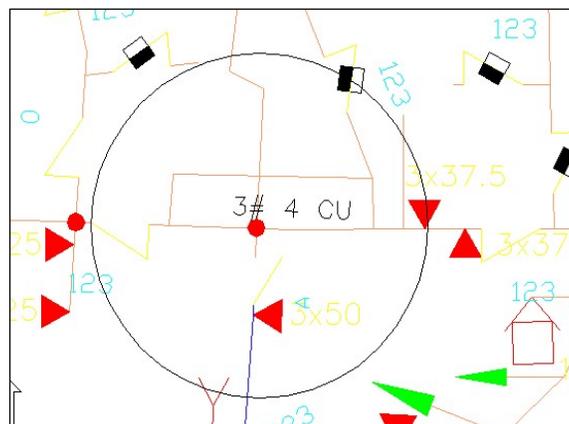


User may notice that the line on the right side ends just at the intersection. In this case the left line is overlapping the right line.

To solve this problem, you must touch the right line and move the GRIP to the end, of the left line using the snap command "endpoint".

**Repeated:** When a repeated section shows that the numbers of the sections are overlapping each other

**Loops:** When a circuit is closed to himself doing a loop or ring. Here is an example:

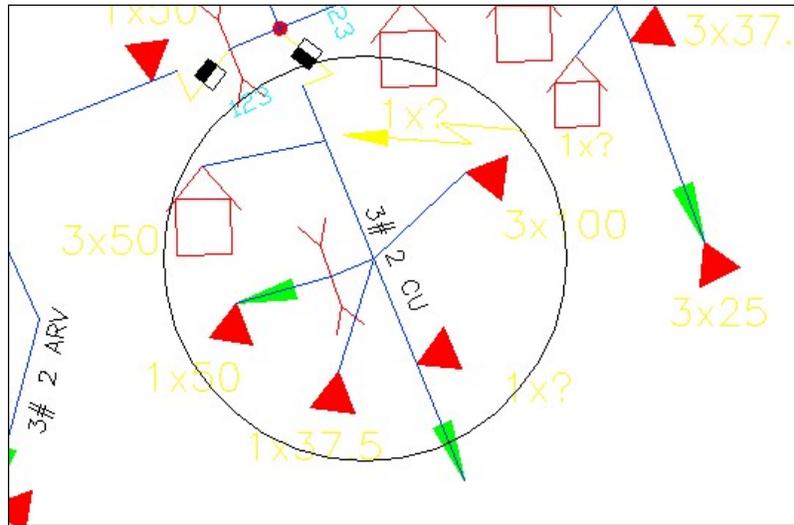


Loops can not be easily detected, so it is recommended to "cut" by half the circuit and try to identify. If the identification is correct, it means that the error found after half of the circuit. It should proceed iteratively to reduce the area where the error

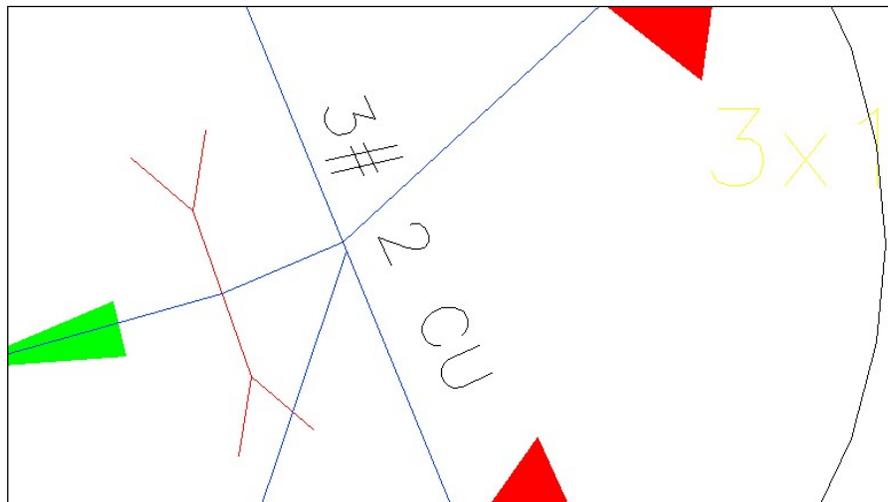
located. In some cases, it is quite difficult so users have to have some patience as it usually is an error while surveying the network in the field.

**Problem for more than 4 branches at a node.** The system allows up to four branches starting from a node.

In the figure below is a sample of this problem. The node enclosed within the circle, has five (5) branches. One section must be separated a little bit.



The following figure shows how the drawing should be to correct the problem.



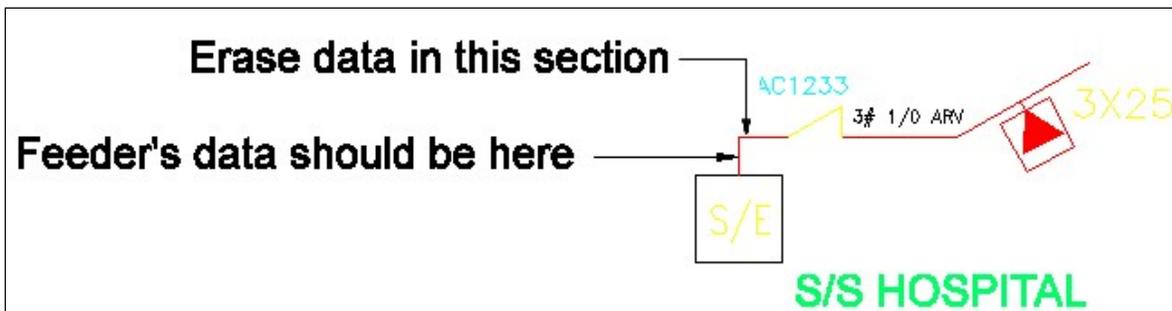
7. ANALYZING: When I run the PANP ANALYZE option, sometimes shows the following: READING RESULTS: NOT COMPLY, KVAINT: 7990 DEM: 8808 3730KVAR ... What does this expression means?

When you run ANALYZE and select the option "Draw Results", the program colors the circuit that does not meet the established criteria (The criteria is enter in the menu that appears when this option is selected, to color the part when the voltage drop maximum criteria or maximum permissible conductor load is exceeded). In addition, the entire length of the circuit, which does not meet these criteria, either by voltage drop or load it is presented in the command line.

8. One of the two error messages that gives the program after running it is that the circuit has no data, why does this?, if the data were inserted by selecting the first line coming out of the S/S.?

Feeder's data must enter in the line that is attached to the substation, if you use Macro-> XD and you touch the line coming out of the substation you'll see no data. However, if you touch the long line that goes to the right you will see that if you have data feeder name, current, etc., the program will give the error: "Can't find the Substation".

The data in this last line should be deleted with MACROS-> XD\_LIMP. The Data should be placed back on the line that is near and "stuck" to the substation.



This also happens when a line that already has feeder's data and a new switch is inserted. Then two lines at both switch sides will have the same data and information appear repeatedly in MACROS-> SEARCHS-> SEAFED

Use the MACROS-> XDLIMP program to remove the feeder's data in the unwanted line.

9. What does this Message means: "Substation's name was not found in Step 2 for the feeder xxx

This happens when you haven't entered the feeder's data, namely: number, name, current etc. to the line that this just stuck to the substation, it can also occur when a switch is inserted right in the first line leaving the substation. The line is spliced with the same duplicate information on either side of the switch. See item 7.

10. When I put capacitors banks the result is equal to the previous one. (Without capacitors)

The right procedure to run capacitors process is as follows:

- a) Run the process Identify, Load Place, Assign, Analyze.
- b) Run the CAPACI option
- c) Verify the location of the capacitors banks and place them using the automatic library INSBLO.
- d) Run Identify and Analyze without using Load Place option.

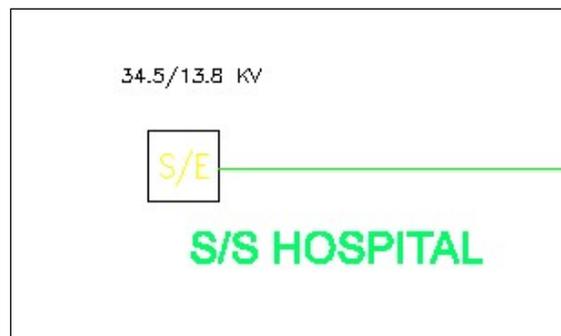
Most problems happen because LOAD PLACE process is run with the same current and power factor at the substation. Please note that in the real world, when the capacitors are placed in a feeder, the current is reduced and power factor improvement in the output of the substation.

When users use the assign loads option there is not need to Load Place again.

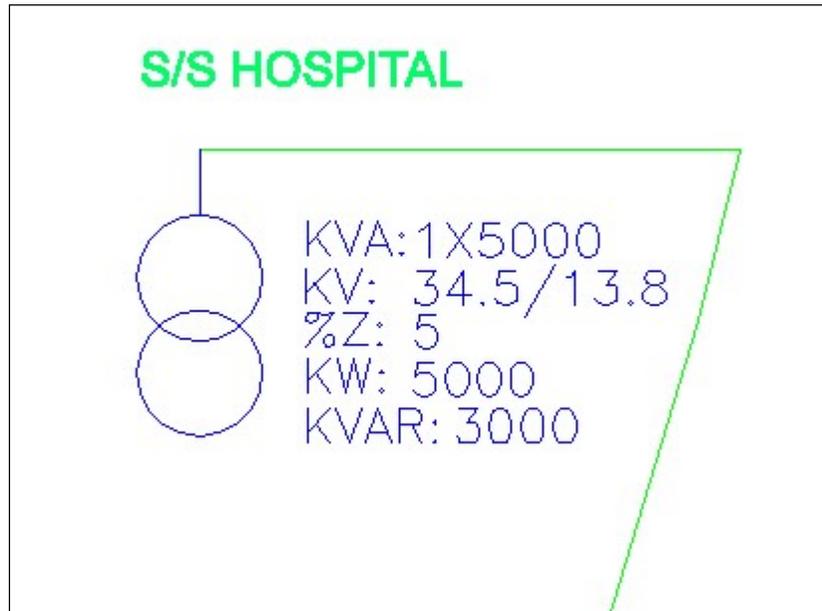
If the user wishes, after the last analysis, we can observe the current and power factor in the analysis output report and modify the feeder's data. Thus you don't have to be carefully in the future. (Results may vary by 1% approx.)

11. I have a substation 115/34.5 KV, Mumbai, which feeds another substation 34.5/13.8 kV (Hospital), It is required to run this 34.5 kV line in PADEE, but the program gives me error because it sees the substation Hospital as a source instead as a load. How can I do? this (See next figure)

The following figure shows how users tend to draw the map

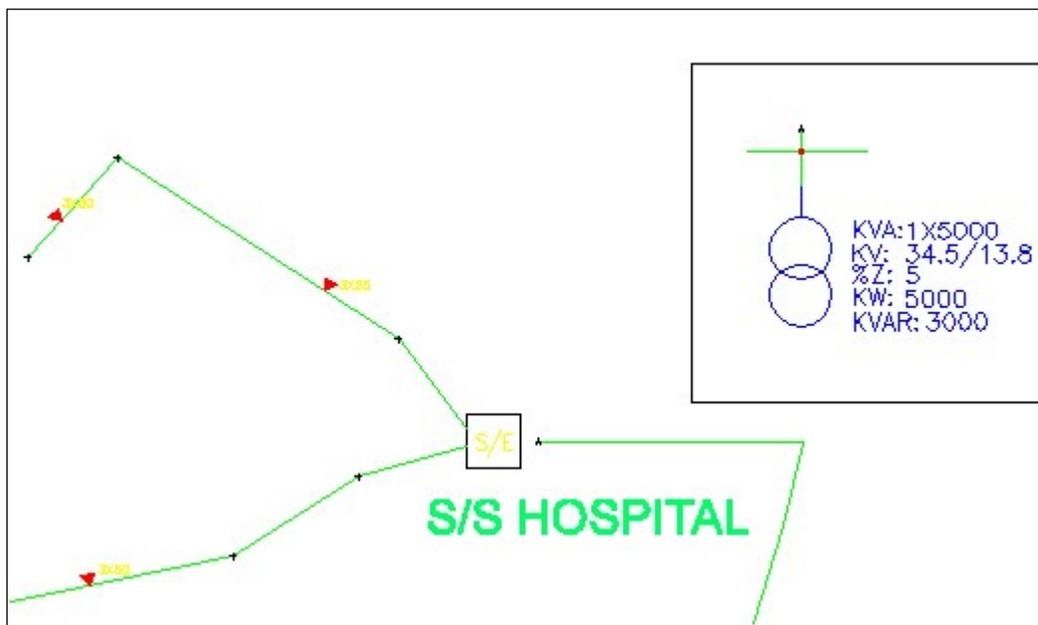


This figure above is incorrect. The symbol used is taken as a source. The correct way to express the network in this case is as shown in the following figure:

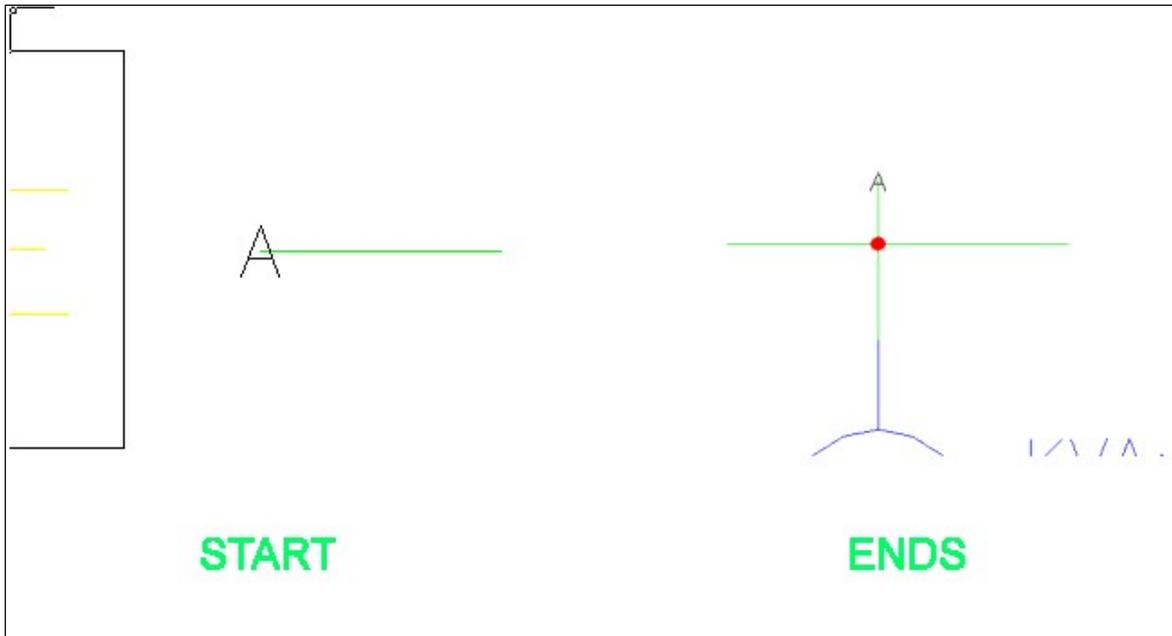


The symbol may be found in the INSSID library. please make sure the block is in layer TRANS, since this library is not the smart one.

If the substation load in question also serves as a source for 13.8 kV feeders, you may represent it with UNIONS, which is also found in the INSSID library. An example is shown in the figure below.



Please notice that 34.5 kV line coming from the left side does not touch the symbol of the substation. Furthermore, the connection block has two letters, where it starts and where its end. The 34.5 kV line have one letter and the end at the detail entrance is the other letter.



The distance between the two "A" will not be taken into account for the analysis

12. In PANP module IDENTIFY option.. I get the error: Failed to open the file C:\PADEE\TEMP\CARNOD.DAT. How can I fix this?

The PADEE needs to write temporary files on the folder c:\padee\temp. TEMP or under drawing/temp (User is allowed to change the temporary folder location, ask Jorge Matheus)

Please check user permissions to see if you can write freely in the c:\PADEE and its sub folders. You can perform a test trying to create a text file in that directory. If the creation of the text file can be done without problems, then you should exit AUTOCAD and delete all files contained in the C:\PADEE\TEMP or under drawing/temp folder, which may have been corrupted by power failure or other previously problems.

13. In IOP module, when I do an operation or maneuver, the program gives me an error and keep the switch in the original position (as it was).

To use the interrupt and operations module, it is necessary that all feeders are properly identified, Load placed, and load assigned. Preferably with the Process All PANP option.

If the feeders are not processed yet, to change the position of a switch, use the switch library INSSWI , press the Edit button and choose the desired switch type and position

Note that IOP module should only be used when you are simulating network operations or maneuvers, the network must be completed and without errors. If on contrary it is being modified to incorporate more network load or to simulate a planning condition INSSWI library to change the position of the switch and then identify and ANALYZE. Thus, it is not necessary that the circuits are completely "identified" to make changes in the network or analyze part of it.

#### 14. What is for the option ASSIGN LOAD?

After running IDENTIFY and LOCATE LOADS, The load demand for each transformer can be write on their drawing's blocks, after doing this you may change the feeder configurations as it was originally when the field survey was done.

This option is used when capacitors are placed, removed or added load to a feeder. These actions increase or decrease the current in the feeders involved in the operation, but the change is not made in the feeder's data. If you run LOCATE LOAD again, without changing the current feeders, the utilization factor will be modified but the currents remain almost the same.

Once the load is distributed, and users assure this analysis is within the expected parameters ( $F_u = 0.3$  for rural loads and  $F_u = 0.6$  for urban areas, approximately) the load should be assigned. This way you can make operations without thinking that the currents in feeders vary in each operation. The currents resulting from the analysis will be presented in the summary.

The IOP system is for use in the Distribution Operation Center D.O.C. the circuits are completely surveyed and analyzed. When the engineer ensures that the map is well built and drawn, then proceeds to make a final load distribution or allocation and then recorded into the block's attributes. CAD this file must be saved and will be available for others operations.

As general rule after the model is setup user shouldn't use Identify, Load Locations and Load Assign options again.

If you want to change the feeder's configuration permanently, not in the DOC, user may use the same routine that is used for switch insertion INSSWI and the MODIFY button.

#### 15. When the program runs IOP gives the message "Error in BUSORD"

As mentioned above the IOP requires that all feeders in the map are processed without errors. When this message appears is because the nodes or sections database are not "right". When you run the "Process all" option, the database is restarted and filled with the correct information. This could happen if user don't save the CAD file after and operation or maneuver.

16. When a switch and / or a circuit breaker are inserted, an identification number of this element is requested; if I put all of them "0" (zero) what could happens? , What are the consequences?

Switches and other disconnect equipment need to be identified to reduce possible maneuver mistakes. When the maps are used to perform operations on the DOC each switching points of the network must be identified without any repetition. The identification may be geographical or any other that way the user's desire. If you do not have these data at the time the network is drawing, it is usual to leave it to 0. But at the time the actual identification numbers is placed in the field users should make the change in the map and enter the right identification number.

17. I need to change the color of the transformer load, because when printing the map on white paper the yellow does not Contrast and it is not well displayed

The best way to do this is using printing pens. At plotting time using the PLOT command pen style appears. You must edit these pens.

In the PEN table the thickness and color for each pen is available. Search for the Yellow pen and select to print in black.

Editing pen's table tells the plotter the color and thickness to give better "expression" when printing and latter making copies. In the user manual a pen table is indicated for the colors that are predefined in the PADEE. This table is copied below.

You may start with the default pen file "monochrome" and edit them to give thicknesses that indicate in the table below. If you are printing in black and white it should be noted that each color would correspond to the BLACK. But if user is printing in color, each color will print with the same as is displayed in the screen, except yellow color that it will be changed to BLACK.

<u>COLOR</u>	<u>PEN THINKNESS</u>	<u>LEROY Equivalent</u>
Magenta	0.15 mm	0000
White, Cyan	0.25 mm	00
Yellow, Bluel	0.35 mm	1
Red	0.50 mm	2

Green	0.70 mm	3
Red ( 010 )	0.60 mm	
Cream (021)	0.60 mm	
Brown (025)	0.60 mm	
Orange (30)	0.60 mm	
Yellow ( 050 )	0.60 mm	
Olive Green (054)	0.60 mm	
Light Green ( 90 )	0.70 mm	
Dark Green (104)	0.60 mm	
Aquamarine (121)	0.60 mm	
Light Blue (130)	0.60 mm	
Dark Blue (160)	0.60 mm	
Violet (212)	0.60 mm	

Pen file can be saved and keep for latter use. The file also can be exported and imported to other machines or other CAD versions, but some time different plotter brand doesn't allow this.

Another way is to use a PADEE's utility, you have to manually load it.

```
(load "c:/padee/lisp/GATCHA")
Then
Command: gatcha
Block name specification <*>: ecadct01
Attribute tag specification <*>: capacity
Attribute value specification <*>:
Attedit: Style/Layer/Height/Oblq angle/Rot angle/Color/<eXit>: C
```

New value: 1

```
Working.....
Attedit: Style/Layer/Height/Oblq angle/Rot angle/Color/<eXit>: X
```

This must be done for each block, so you better use the other option that is only needed once and the maps are printed with expression.

### 18. When PADEE is run the message "ERROR Writing No 2" is presented. What does it mean?

This error occurs because the user does not have permission to write on the PC's hard drive. Usually it occurs during installation, but can also occur if any of the users modify their local privileges.

You should check; if you can write freely over the following folders copying any file (for example a text file) in the folder c:\padee and then delete it again. If you can't do it is because your PC has restricted access to your hard disk.

You should also check that you have free access to the Windows folder by repeating the same process above in this folder.

It is recommended that the PADEE's user has Local administrative privileges on their PC to work smoothly, otherwise should "share" the folder C:\padee and C:\Windows with all its subfolders.

#### 19. UPDATING tells me I can not connect to the FTP server

The UPDATE application needs to connect through port 21 & 22 to check which updates are available and downloaded the necessary updates. Also it required to connect to the page <http://padeepro.com> and <http://matmor.cloudns.org/>

#### 20. SGMAP no "Google Map" image is presented

The SGMAP option requires these several conditions are met:

- a) The map must be georeferenced in UTM coordinates. (REGVEN) or state plane for USA.
- b) Must correctly choose the UTM zone or state plane for USA or map's images will not be presented correctly
- c) The user should be able to access the Google Map application on Internet
- d) The user should be able to run JavaScript
- e) The <http://maps.google.com/maps/api/> application API must be accessible from the user's machine.
- f) The firewalls or antivirus firewalls must allow execution of the program c:\padee\PFIE\track.exe

Rarely the map of a town or city can be found between two UTM zones, so the program could not show correctly the Google MAP image or presented wrongly. The program tries to make an approach in these cases, for ease of use to the user. User must select the UTM zone that covers the most part of the city.

#### 21. SGMAP "Google Map" display a SCRIPT error u other message

Every 3-month Google changes their API which are the main resource to use their maps. Mostly it doesn't affect our application, but when it does, an update will solve the problem. Also you can select to use Microsoft Bing Map.

Update a least every 3 months or once a month is recommended

#### 22. When the PNAP is running the "exceeded the maximum number of nodes" message appears

In the PNAP Options menu on the left side there is an option that says "Start from Zero". Simply select this option to restart the nodes and sections count.

The PADEE has up to 1000 feeders 5000 nodes each. Which it is enough to analyze the vast majority of the cases. This amount can be extended to a much larger number, but a higher processing speed computer is required, in addition to having enough memory capacity to handle larger CAD maps.

In any case it is advisable to work by city or state.

### 23. When I'm Identifying the message is not in line $z = 0$ appears. How to solve this?

This happens when working on a road map with elevations or mapping, ie 3D. Possibly some component of the network is traced and inadvertently left with a lift.

In other word the coordinates of the origin or end of the line are not zero as in the following example

(Source 10,10,2.3) ----- (end 15,10,2.7)

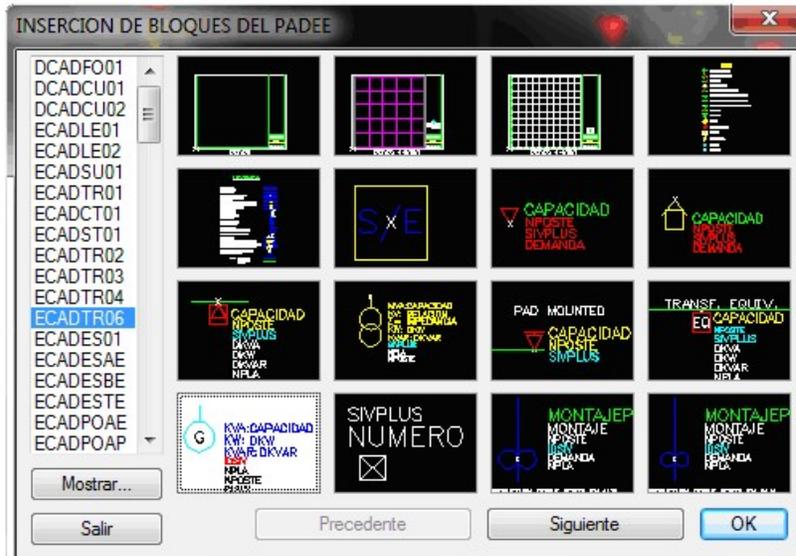
The value 2.3 and 2.7 correspond to the elevation or Z value. This can generate an error in the calculation of the length and thus get the error message.

### 24. How does PADEE simulate the "Distributed Generation" ?

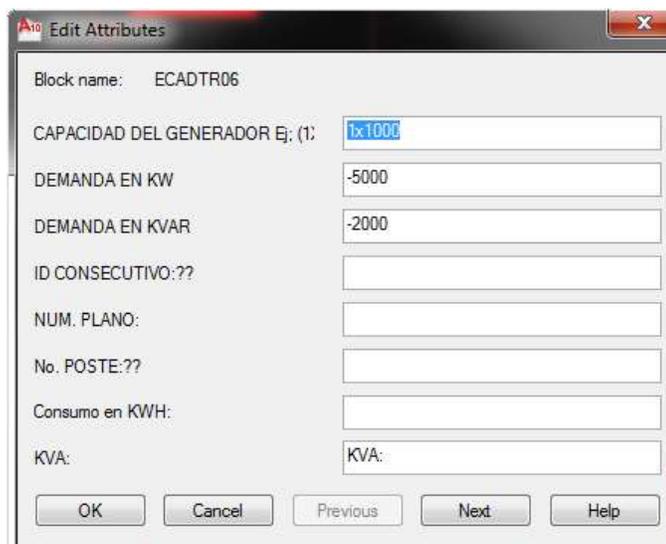
There are two ways:

- a) When there are several generators in a single point. It is modeled like a substation
- b) When it is a "relatively small" generator installed on a circuit

After simulate the feeder, (identify, allocate and assign) insert a block generator using INSSID



It is place like a transformer and generating data that will contribute to the feeder with negative values are filled.



Then a new simulation with identifying and analyzing without load location and Load assign.

25. When I'm updating, I get the error "Can not connect to the FTP matmor.cloudns.org or 192.168.10.12"

The reference problem is basically due to users' restrictions to browse the Internet freely.

Since the restriction is done on their servers and / or user equipment is difficult

for me to raise solutions. However, given the recent tests conducted with staff from various regions ATIT, here is a list of some of the points that users must check:

- Should be able to seamlessly access the page <http://matmor.cloudns.org>  
In some cases it was necessary to add this address to the list of addresses allowed by the server in the region. (Check the white lists)
- In other cases also they had a problem with the DNS service. They had CANTV including DNS 200.44.32.12 (please use the google dns 8.8.8.8 alternative) on the server connection
- They must have free access read and write to the folder `c:\padee`  
An easy test is to create a text file in the folder `c:\padee` and then delete it. If the creation and deletion is possible, they have free access.
- It is also necessary for the user or server allow access to the FTP protocol and port 21 & 20.

If it is not possible to have a free Internet the connection, a "Pen drive type" MODEM type was used with connection via GSM phone.

In 99% of cases the problem was solved after adjusting user connections.

We have several servers in several countries so there is a very small chances that problem is on our side.

## 26. [To install the new PADEE 2023 version, should I uninstall the older version first?](#)

It is no necessary. The new version Will be installed overwriting the old files in the same folder.

But please remember that the new version will requiere new compiler system files, so the use of the setup Disk is necessary

For the other program like COMWIN or MAINTENANCE the data base is 100 % compatible so if you have changed these databases or create new projects, do not overwrite them.

## 27. [After finishing setup NO PADEE menus appear in Autocad](#)

If after installing PADEE no menus appear in CAD, you can proceed as follows:

- Run the application `c:\padee\Autocad\INSCADPA.exe`

Eng. Jorge Matheus

If after running this application the menus didn't appear, carefully check whether an error occurred during the execution of the application INSCADPA.exe. If an error is observed please make screen image and send it to [jorgematheus@gmail.com](mailto:jorgematheus@gmail.com) . But if no error appears, please search in the folder c:\padee\autocad\Acadver.sal file and send it to [jorgematheus@gmail.com](mailto:jorgematheus@gmail.com)