



CAMELIA LĂCRĂMIOARA POPA



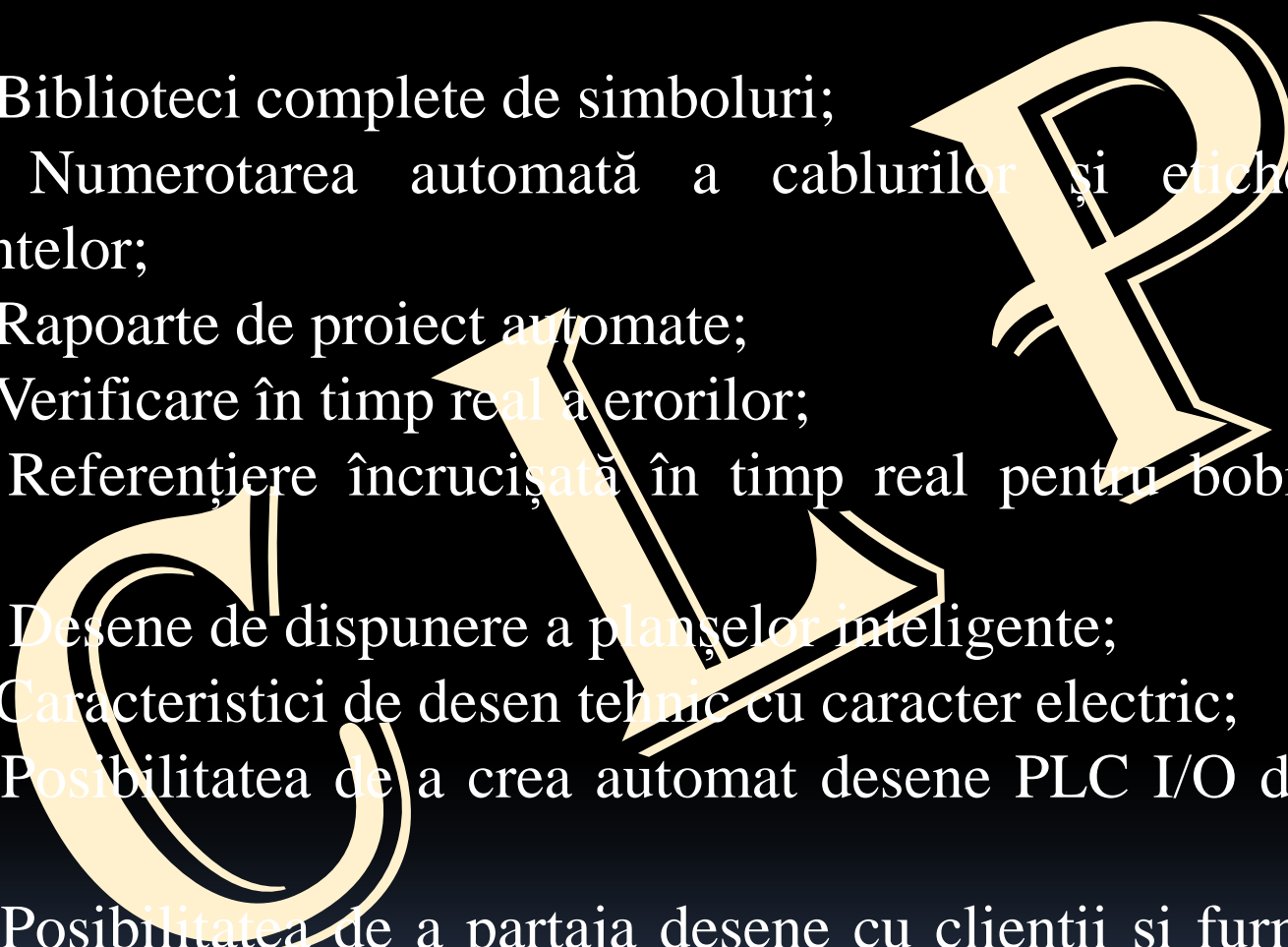
AUTOCAD
ELECTRICAL



*INTRODUCERE ÎN
ACADE*



DE CE ACADĒ?

- 
- √ Biblioteci complete de simboluri;
 - √ Numerotarea automată a cablurilor și etichetarea componentelor;
 - √ Rapoarte de proiect automate;
 - √ Verificare în timp real a erorilor;
 - √ Referențiere încrucișată în timp real pentru bobine și contacte;
 - √ Desene de dispunere a plășelor inteligente;
 - √ Caracteristici de desen tehnic cu caracter electric;
 - √ Posibilitatea de a crea automat desene PLC I/O din foi de calcul;
 - √ Posibilitatea de a partaja desene cu clienții și furnizorii acestora;
 - √ Reutilizarea desenelor existente.

The background features large, stylized yellow letters 'C', 'D', and 'P' with a white outline, set against a black background. The letters are arranged in a way that they appear to be part of a larger graphic or logo.

Componența desenelor electrice

▶ *Schema circuitului* utilizează:

- Simboluri pentru a reprezenta componentele;
- Linii pentru a reprezenta firele pe care le conectează.

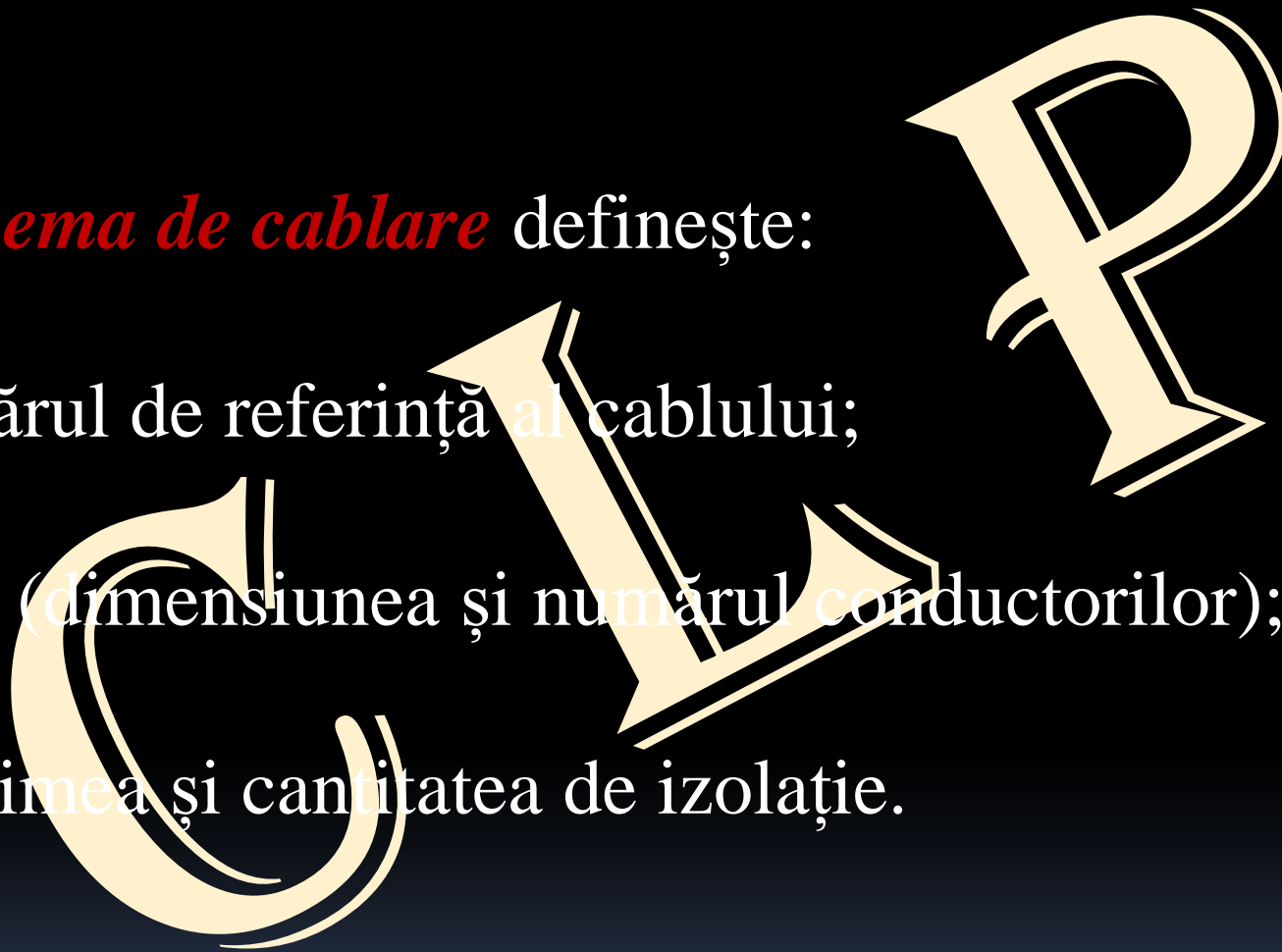



► *Schema de conexiuni* cuprinde:

- Funcții de control sau de semnal;
- Surse de alimentare și conexiuni de împământare;
- Terminarea conductelor nefolosite, a contactelor;
- Interconectarea prin posturi terminale, blocuri, prize.

▶ *Schema de cablare* definește:

- Numărul de referință al cablului;
- Tipul (dimensiunea și numărul conductorilor);
- Lungimea și cantitatea de izolație.





► *Schema bloc* se proiectează înainte
de începerea schemei de circuite pentru:

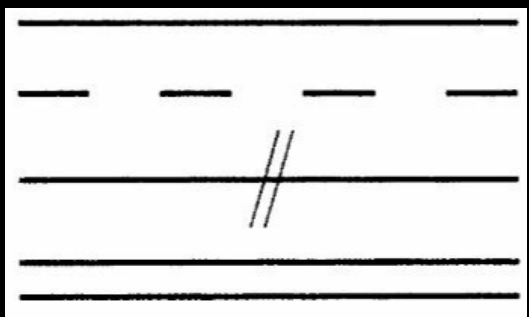
- a ilustra operațiile principale;
- a descrie principiile echipamentului.



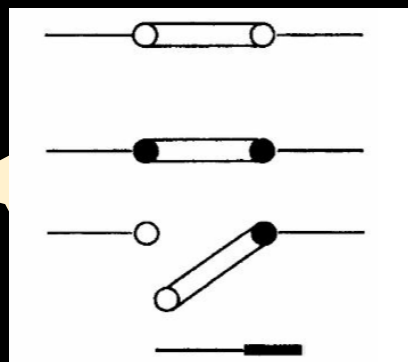
▶ *Lista componentelor* oferă informații despre:

- Tipurile de componente și numerele lor de referință pentru desenarea circuitului;
- Localizarea și traversarea numerelor de coduri.

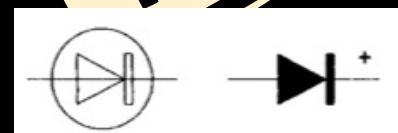
► *Simboluri electrice*



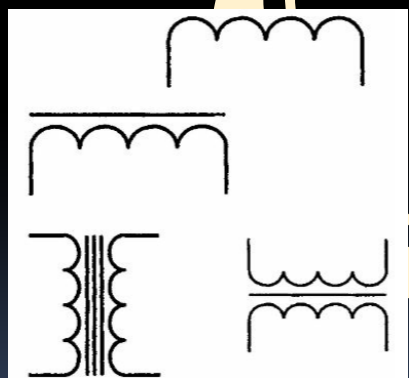
Conductori



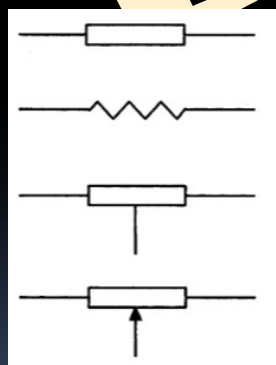
Conectori și terminale



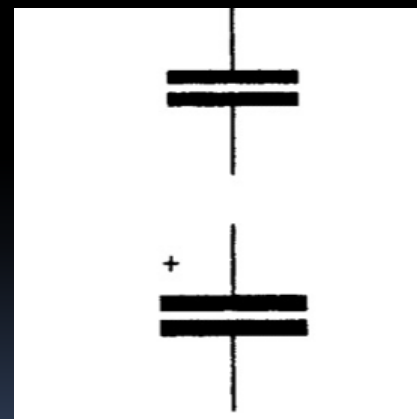
Diode



Inductanțe și transformatoare



Rezistoare



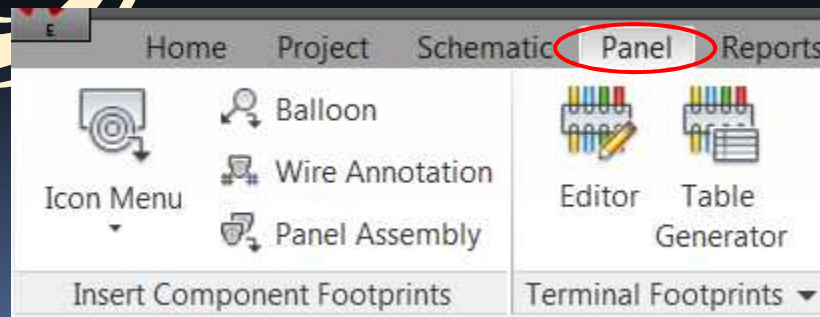
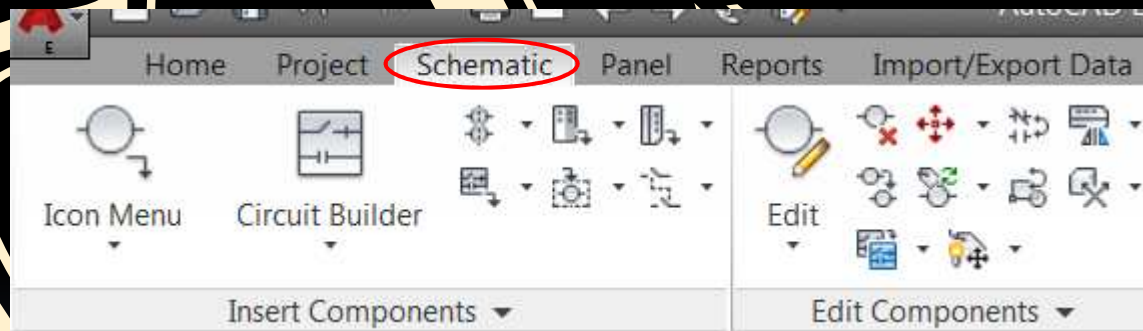
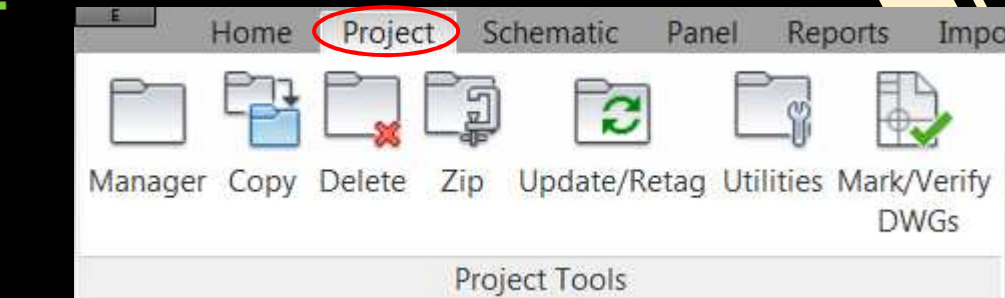
Condensatoare



I. Introducere în AutoCAD electrical

Descrierea interfeței ACADE; ce e nou?

Ribbon

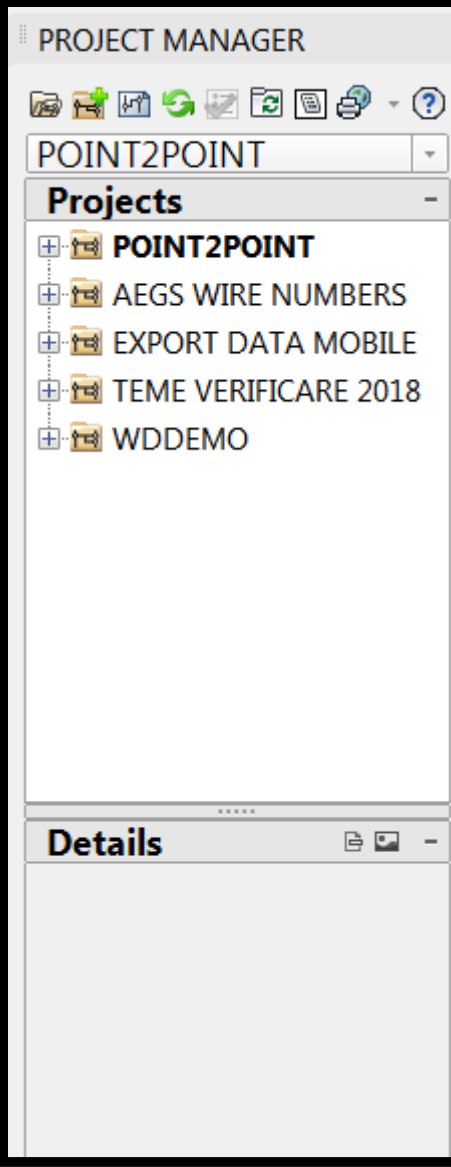


*Instrument de navigare prin
proiectele personale și prin
toate desenele*

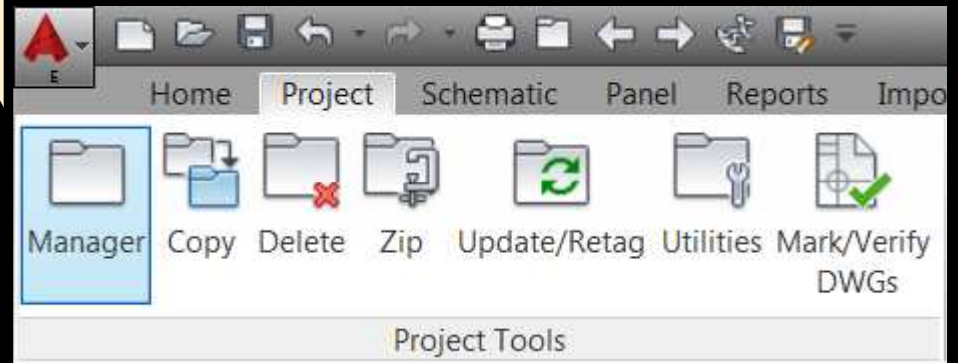
**Project
manager**

PROJECT MANAGER





reafişare

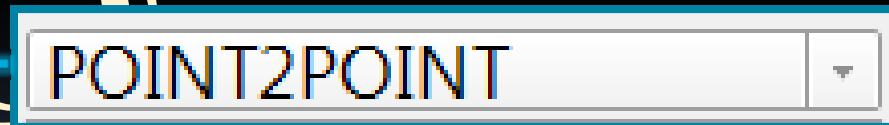


Bara de instrumente

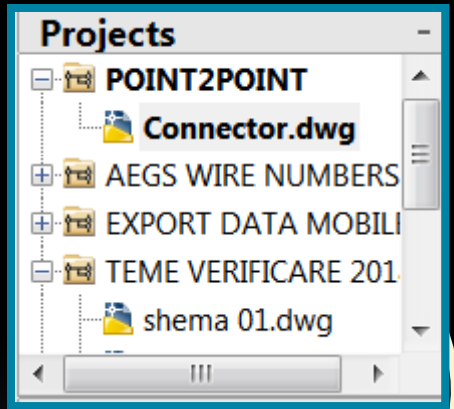


Este compus din

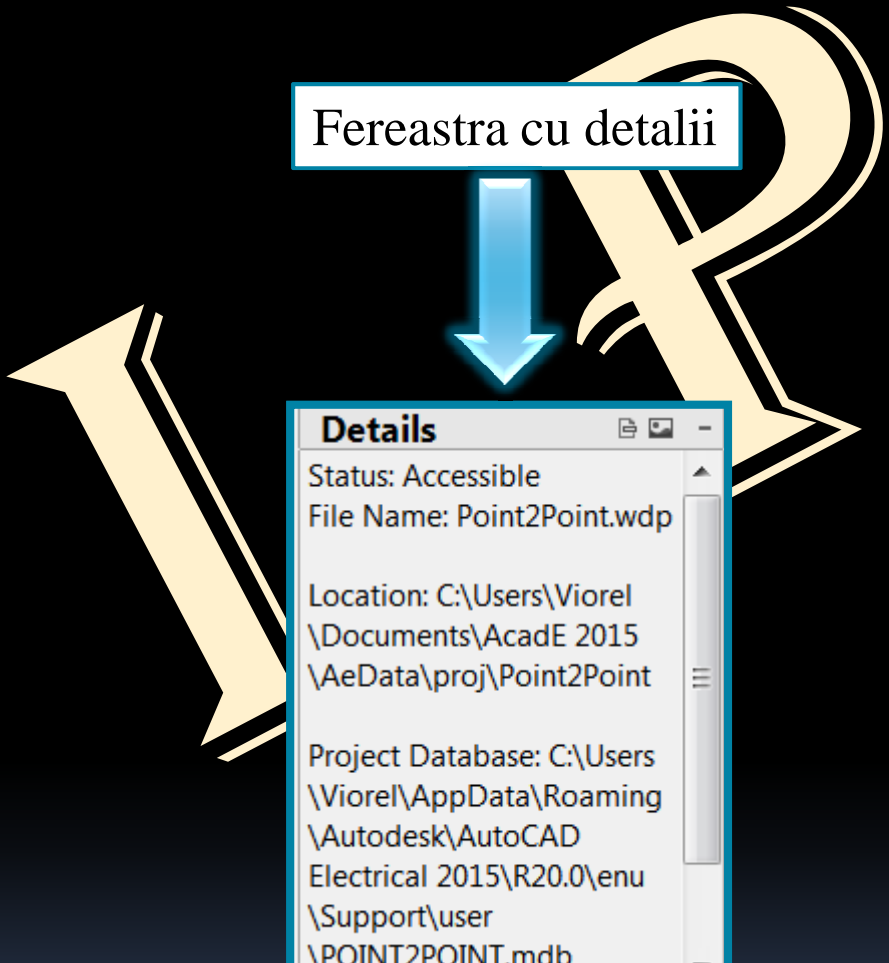
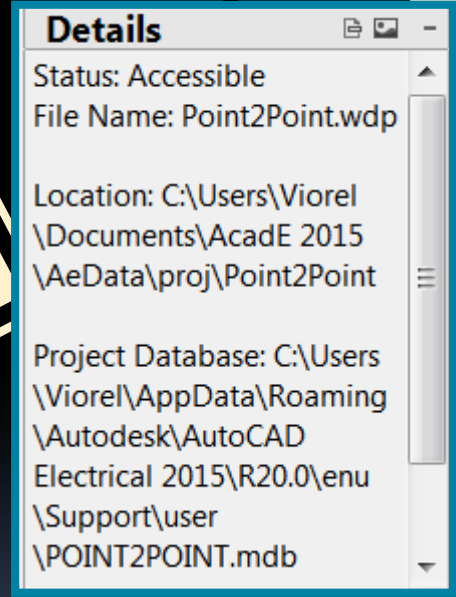
Tabul cu proiectul current



Fereastra de preview



Fereastra cu detalii



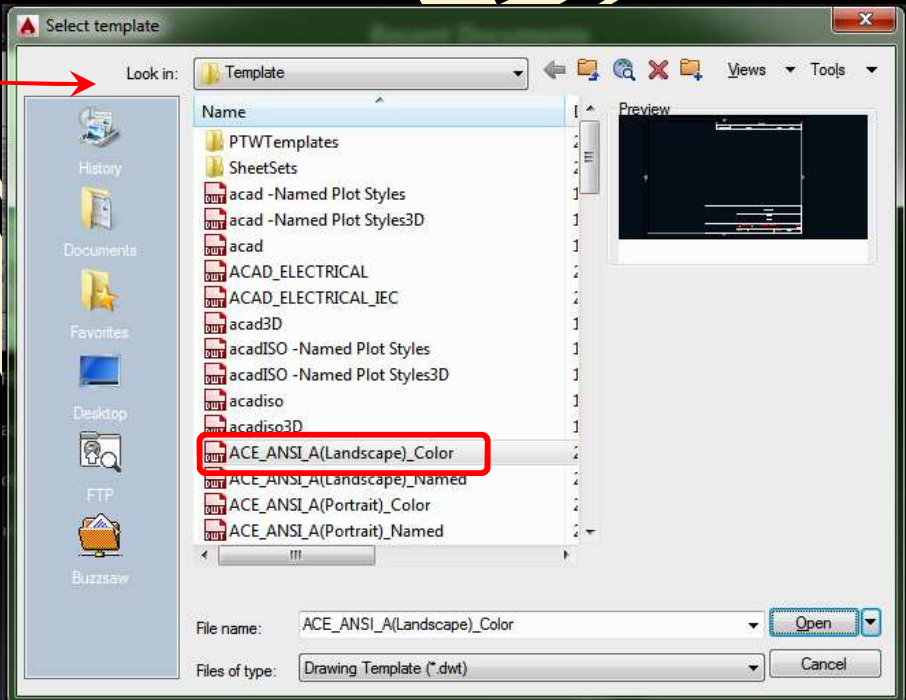
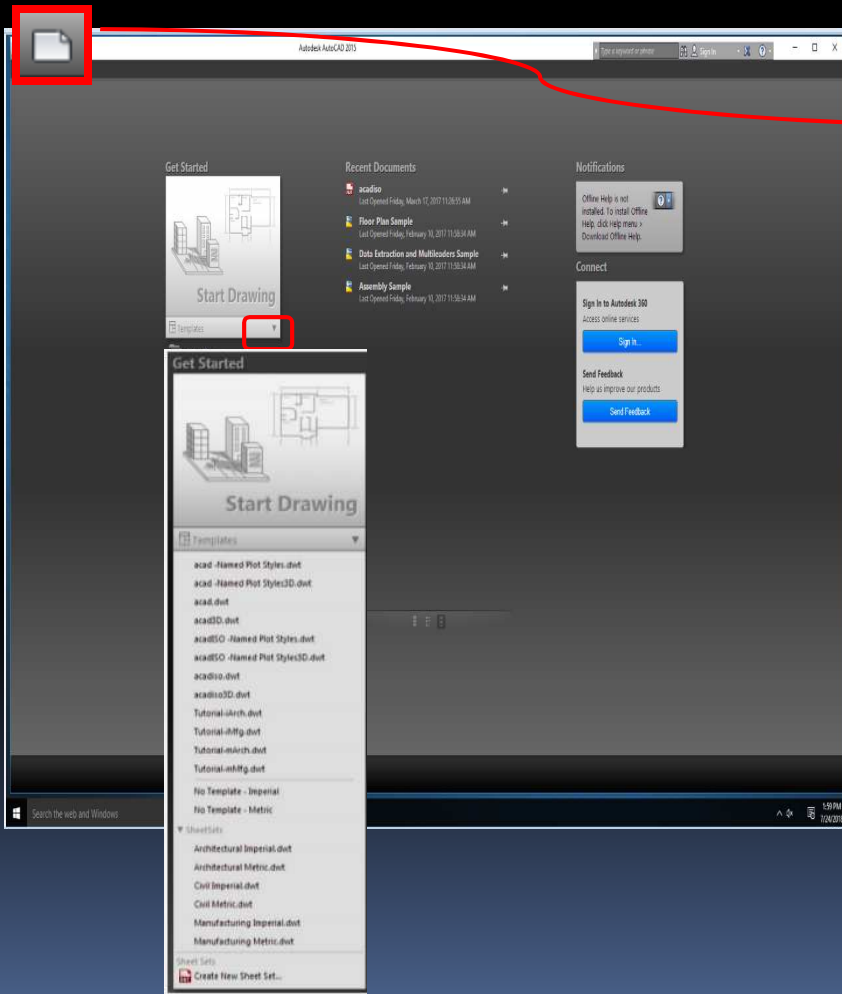
Opțiuni de lucru

The background features large, stylized, golden-yellow letters 'C' and 'P' with black outlines, set against a black background. The 'C' is on the left and the 'P' is on the right, both partially obscured by the text boxes.

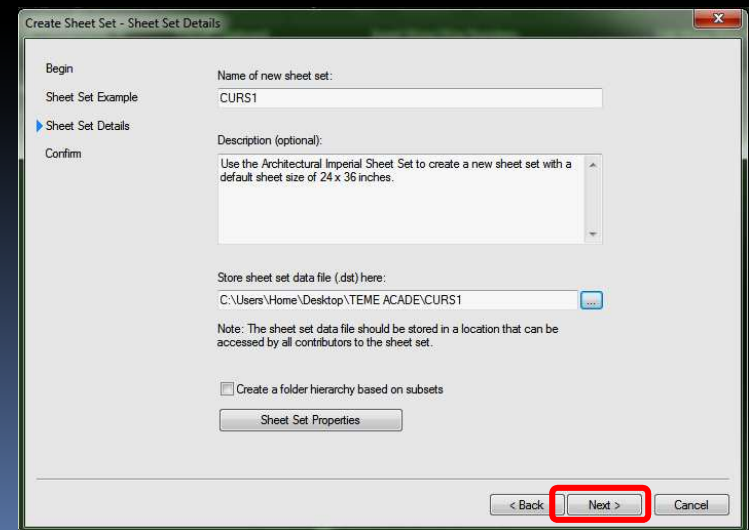
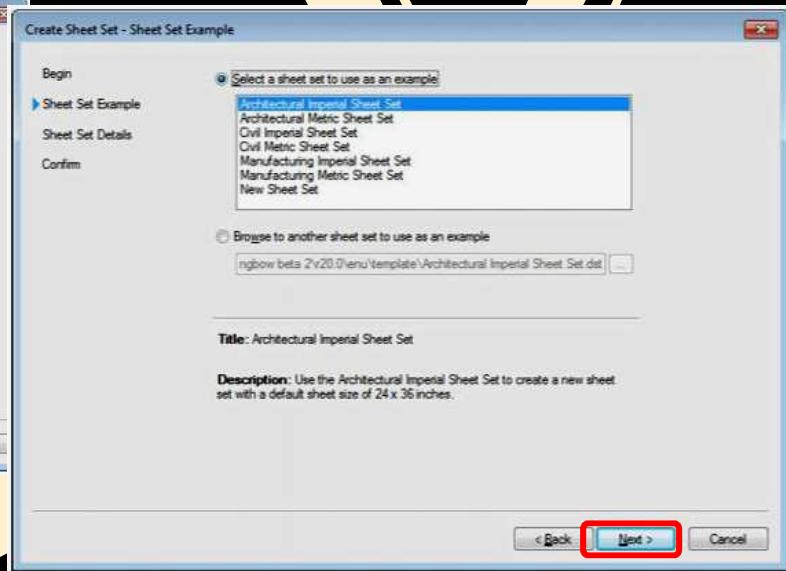
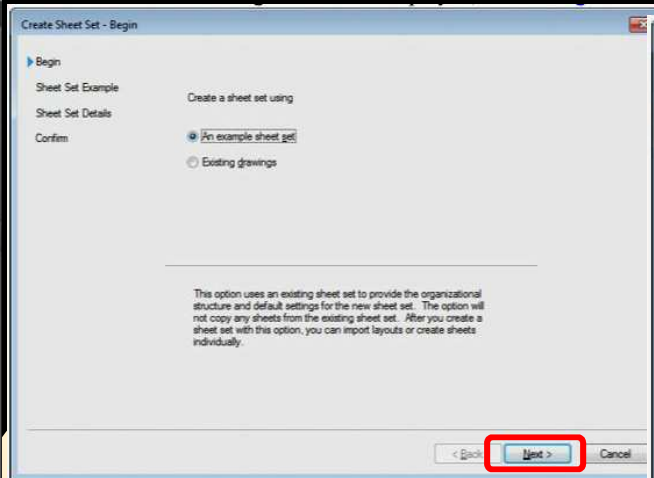
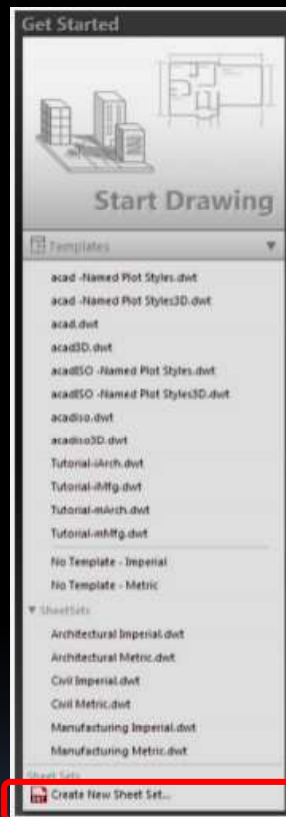
Crearea unui document individual

Crearea de grupuri de fișiere

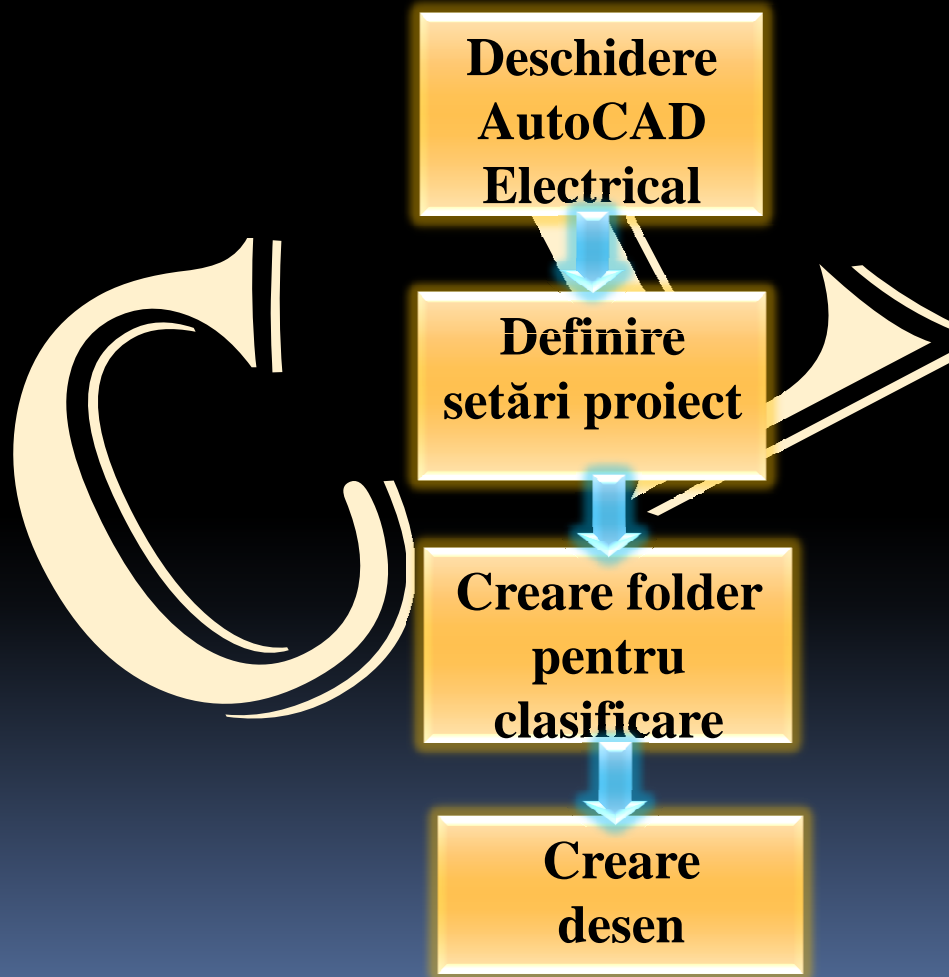
Crearea unui document individual



Crearea de grupuri de fișiere



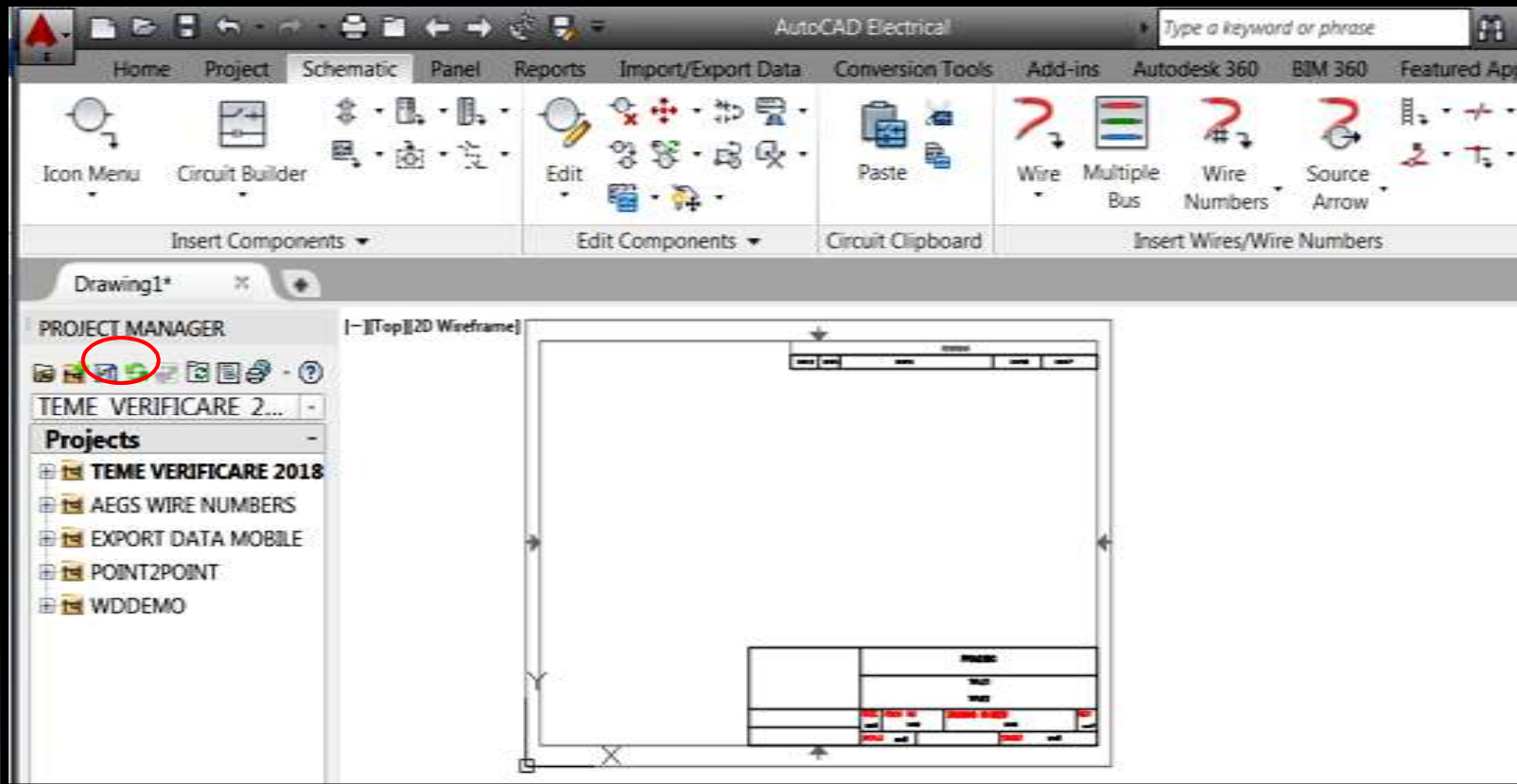
II. Fluxul de lucru în AutoCAD Electrical





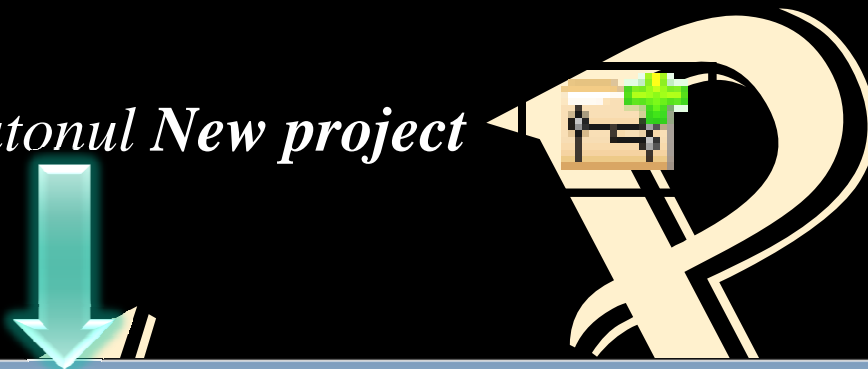
1. Inițializarea unui proiect nou

*1. Deschideți un desen nou:
ACE_ANSI_A (Landscape)_Color.dwt!!!*



*2. Creați pe desktop un folder nou:
NUME PROPRIU*

3. Click pe butonul New project



Create New Project

Name:
CURS01

Location:
C:\Users\Viorel\Desktop\CURS ACADE Browse...

Create Folder with Project Name
C:\Users\Viorel\Desktop\CURS ACADE\CURS01\CURS01.wdp

Copy Settings from Project File:
C:\Users\Viorel\Documents\AcadE 2015\AeData\proj\Demo\wddemo.wdp Browse...

Descriptions...

OK - Properties... OK Cancel Help

4. Click pe butonul Description

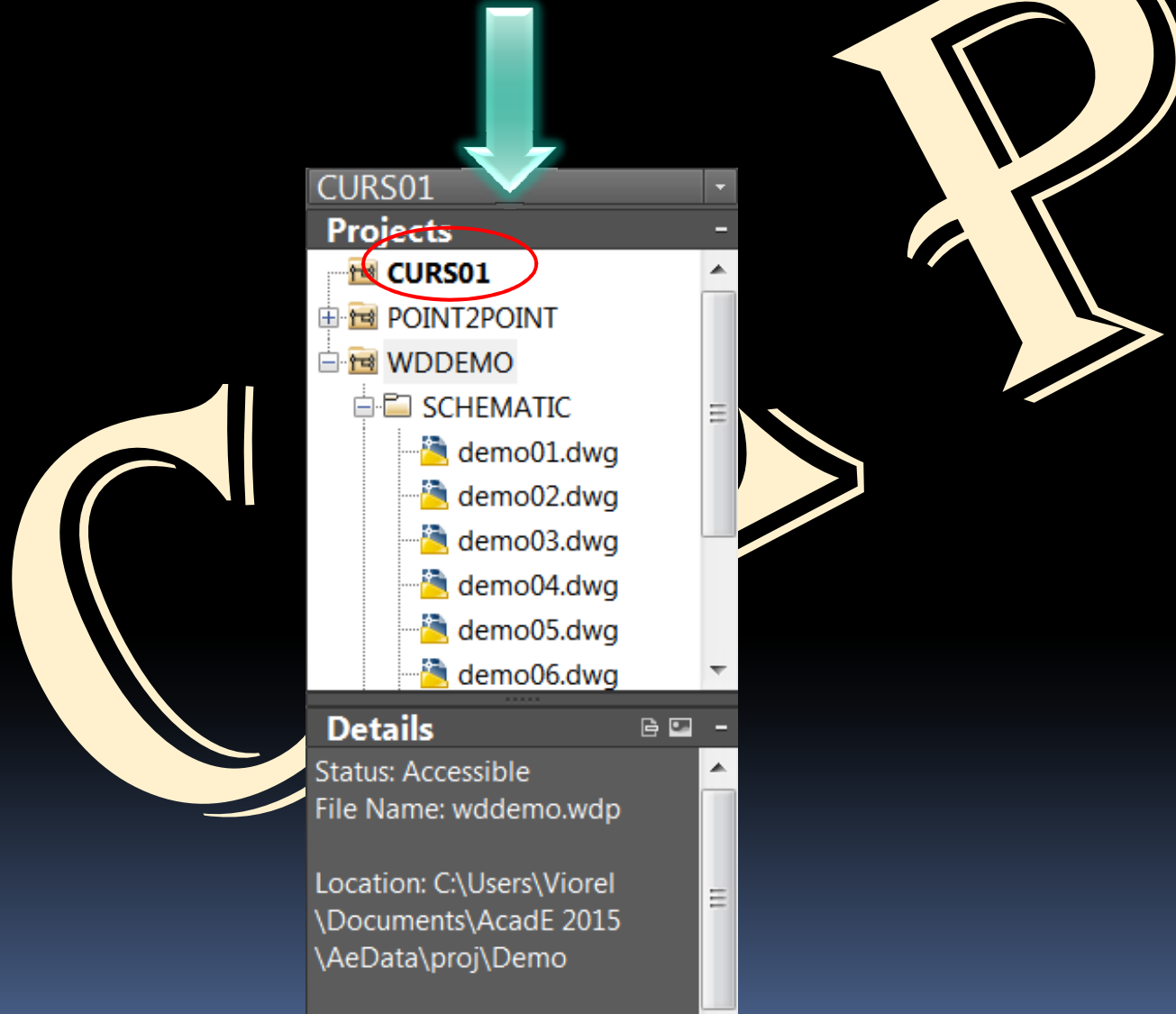


Project Description (for report headers and title block update)

Line1	<input type="text"/>	<input checked="" type="checkbox"/> in reports
Line2	<input type="text"/>	<input checked="" type="checkbox"/> in reports
Line3	<input type="text"/>	<input checked="" type="checkbox"/> in reports
Line4	<input type="text"/>	<input type="checkbox"/> in reports
Line5	<input type="text"/>	<input type="checkbox"/> in reports
Line6	<input type="text"/>	<input type="checkbox"/> in reports
Line7	<input type="text"/>	<input type="checkbox"/> in reports
Line8	<input type="text"/>	<input type="checkbox"/> in reports
Line9	<input type="text"/>	<input type="checkbox"/> in reports
Line10	<input type="text"/>	<input type="checkbox"/> in reports
Line11	<input type="text"/>	<input type="checkbox"/> in reports
Line12	<input type="text"/>	<input type="checkbox"/> in reports

OK Cancel < >

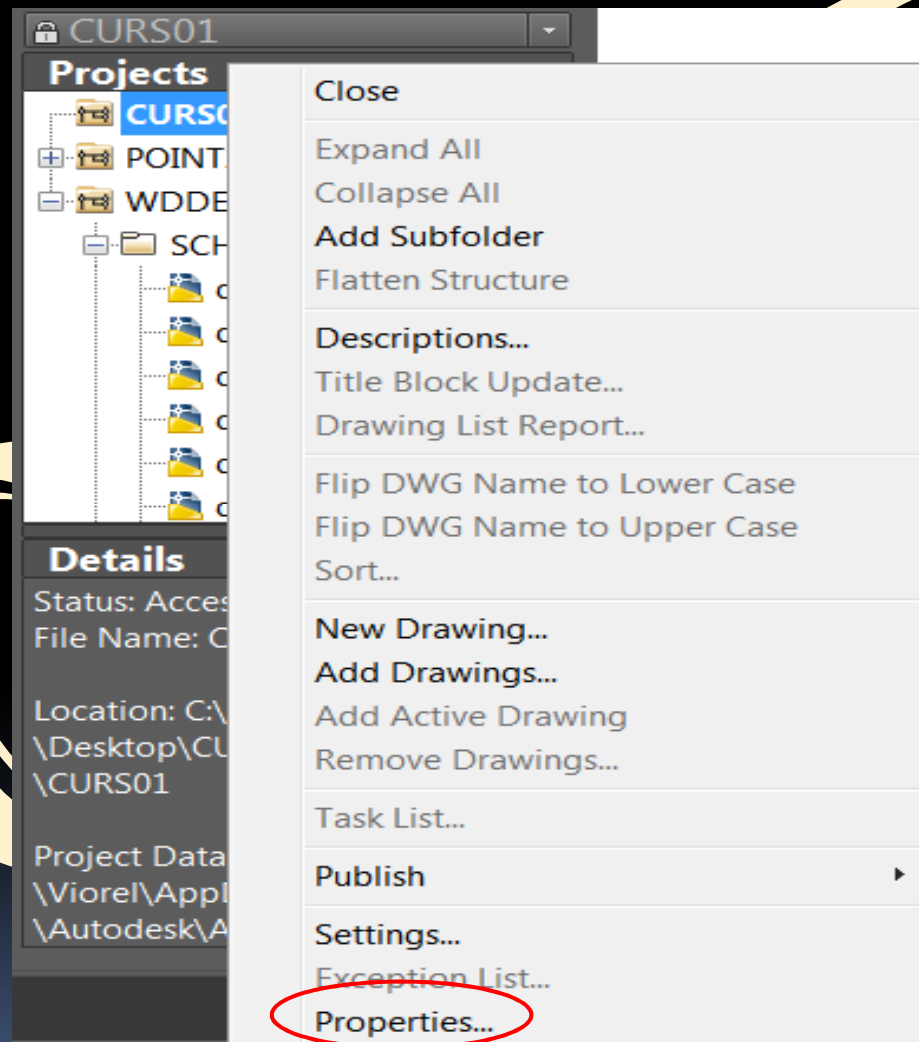
5. Click pe butonul **OK** pentru a crea proiectul





2. Definierea proprietăților unui proiect

1. Click dreapta pe numele proiectului din Project Manager



2. Caseta de dialog Project Properties va fi afișată

Project Settings

Project Properties

Project Settings Components Wire Numbers Cross-References Styles Drawing Format

Project C:\Users\Viore\Deskto\CURS ACADE\CURS01\CURS01.wdp

Library and Icon Menu Paths

- Schematic Libraries
- Schematic Icon Menu File
- Panel Footprint Libraries
- Panel Icon Menu File

Add

Browse...

Remove

Move Up

Move Down

Default

Catalog Lookup File Preference

Use component specific tables Always use MISC_CAT table Use MISC_CAT table only if component specific table does not exist

Other File... (none defined)

Options

Real time error checking

Tag / Wire Number Sort Order: No Project Default

Electrical Code Standard: Default (National Electrical Code)

OK Cancel Help

**Project
Settings**

Library Icon Menu Path specifică directoarele care conțin simboluri și pictograme

Catalog Lookup File Preference setează baza de date a fișierelor care urmează să fie incluse în bibliotecă

Options prin **Real time error checking** permite sistemului să verifice existența unor erori de bază în desen

Components

Project Properties

Project Settings Components Wire Numbers Cross-References Styles Drawing Format

Project C:\Users\Vinod\Desktop\CURS ACADE\CURS01\CURS01.wdp

Component TAG Format

Tag Format

Sequential:

Search for PLC I/O address on insert Line Reference

Component TAG Options

Combined Installation/Location tag mode

Suppress dash when first character of tag

Format Installation/Location into tag

Suppress Installation/Location in tag when match drawing default

Suppress Installation/Location in tag on reports

Upon insert: automatic fill Installation/Location with drawing default or last used

Component Options

Description text upper case

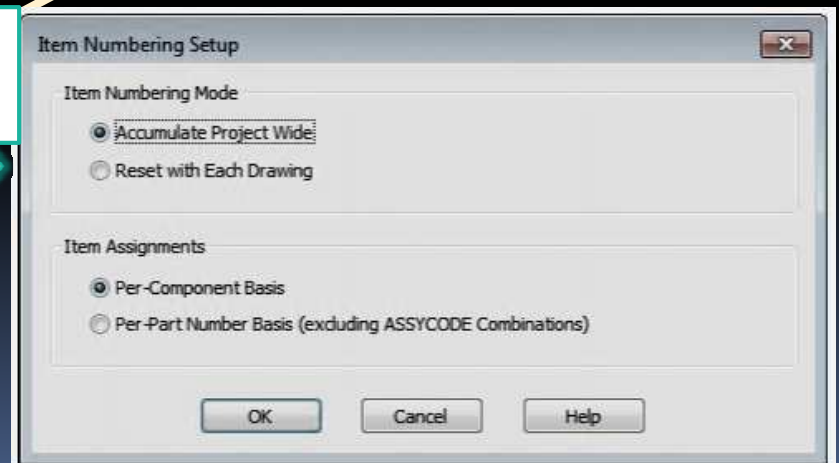
Component TAG Format setează formatul pentru eticheta atribuită diverselor componente

Components

Component TAG Options utilizează eticheta creată prin combinarea codului de instalare/locatie

Component Options

Click
*Item
Numbering*



Wire Numbers

Project Properties

Project Settings | Components | **Wire Numbers** | Cross-References | Styles | Drawing Format

Project C:\Users\Viorel\Desktop\CURS ACADE\CURS01\CURS01.wdp

Wire Number Format

Format: Sequential:
Increment:
 Search for PLC I/O address on insert Line Reference

Wire Number Options

Based on Wire Layer
 Based on Terminal Symbol Location
 Hidden on Wire Network with Terminal Displaying Wire Number
 On per Wire Basis
 Exclude:

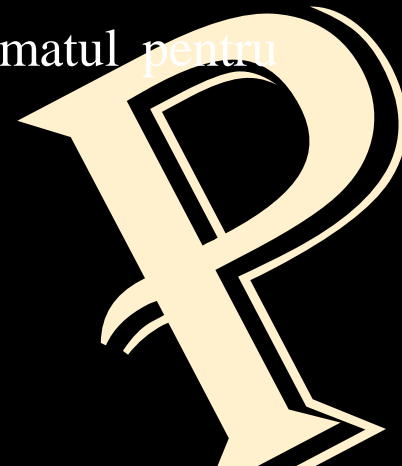
New Wire Number Placement

Above Wire Centered
 In-Line Offset
 Below Wire
Leaders:

Wire Type

OK Cancel Help

Wire Number Format setează formatul pentru numerotarea firelor în proiect

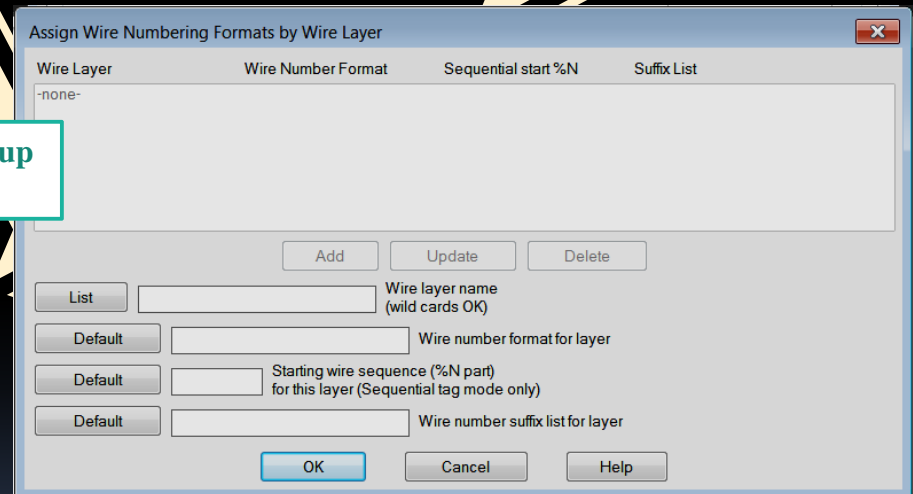


Wire Numbers

Wire Number Options

Click Based on Wire Layer

Layer Setup activ



New Wire Numbering Placement definește poziția de plasare la numerotarea firului

Cross-References

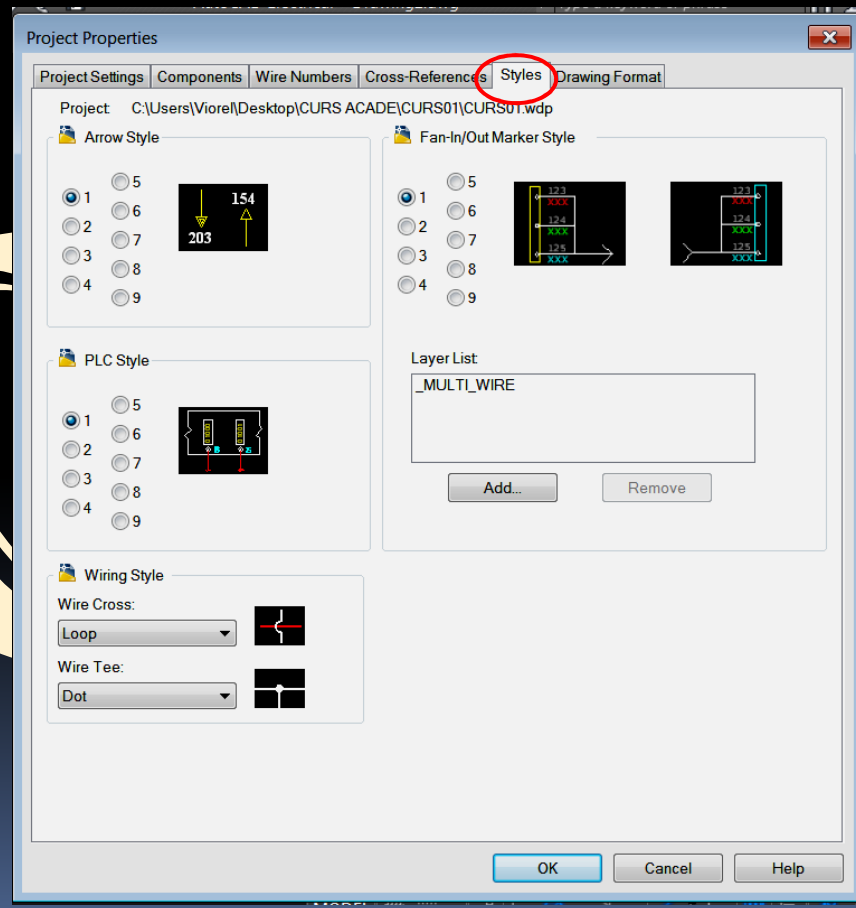
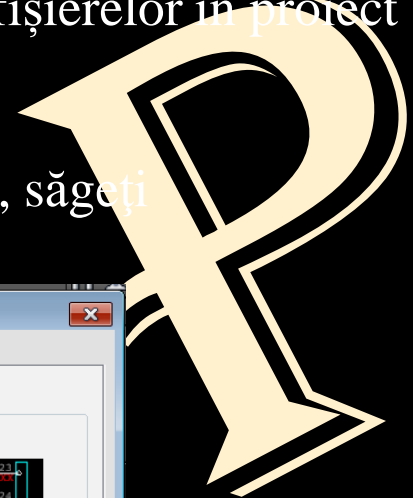


gestionează relaționarea fișierelor în proiect

Styles



schimbă stilurile legate de cablare, săgeți





3. Deschiderea unui desen nou în proiect

1. *Selectați butonul*



din Project Manager

Create New Drawing [X]

Drawing File
Name:

Template:

For Reference Only

Location:
C:\Users\Viorel\Desktop\2018\CURS 01\Tema01.dwg

Description 1: ▼

Description 2: ▼

Description 3: ▼

IEC - Style Designators

Project Code:

Installation Code:

Location Code:

Sheet Values

Sheet: Section:

Drawing: Sub-Section:

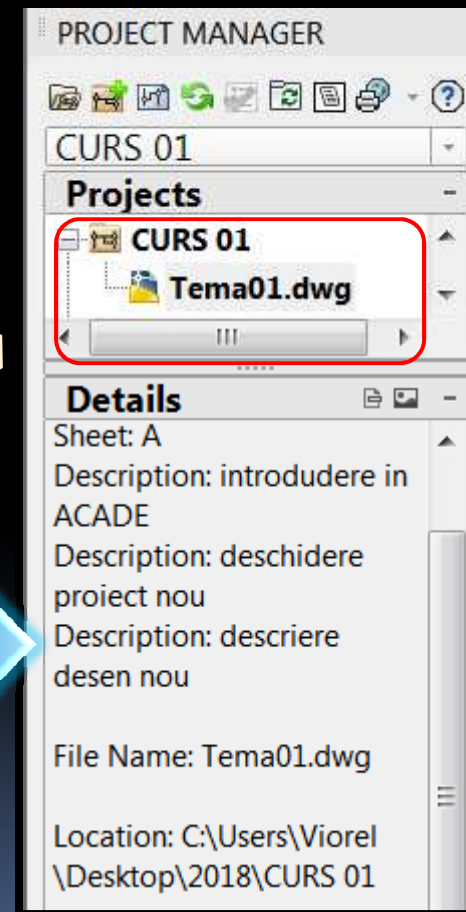
2. *Specificați numele fișierului desen*

3. *Browse pentru a selecta un fișier ca șablon*

4. *Browse pentru a selecta un fișier de lucru*

5. *Completați celelalte tab-uri*

6. *Desenul este inserat în Project manager*



The background features large, stylized yellow letters 'C', 'L', and 'P' with a white outline, arranged in a diagonal pattern. A blue horizontal bar with a white border is positioned across the middle of the image.

4. Modificarea proiectelor

◆ Actualizarea sau reetichetarea firelor sau numerelor de fire

Project Wide Update or Retag



Project-Wide Update or Retag

Select options to run on selected drawings from active project.

<input type="checkbox"/> Component Retag	
<input type="checkbox"/> Component Cross-Reference Update	
<input checked="" type="checkbox"/> Wire Number and Signal Tag/Retag	<input type="button" value="Setup"/>

<input checked="" type="checkbox"/> Ladder References	
<input type="radio"/> Resequence	<input type="button" value="Setup"/>
<input checked="" type="radio"/> Bump - Up/Down by	<input type="text" value="0"/> (Use "-" to bump down)

<input type="checkbox"/> Sheet (%S value)	
<input type="radio"/> Resequence - Start with	<input type="text" value="1"/>
<input type="radio"/> Bump - Up/Down by	<input type="text" value="0"/> (Use "-" to bump down)
<input type="checkbox"/> Drawing (%D value)	<input type="text"/>
<input type="checkbox"/> Other Configuration Settings	<input type="button" value="Setup"/>
<input type="checkbox"/> Title Block Update	<input type="button" value="Setup"/>

Tabul Wire Number and Signal Tag/Retag



◆ Configurarea detaliilor desenelor

Drawing List Display Configuration



Drawing List Display Configuration

Display Options:

- File Name
- Installation Code (%I)
- Location Code (%L)
- Section
- Sub Section
- Sheet Number (%S)
- Drawing Description 1
- Drawing Description 2

Current Display Order:

Drawing Number (%D)

- Separator Value

Always show selection highlight

Show selection highlight only when active

OK Cancel Help

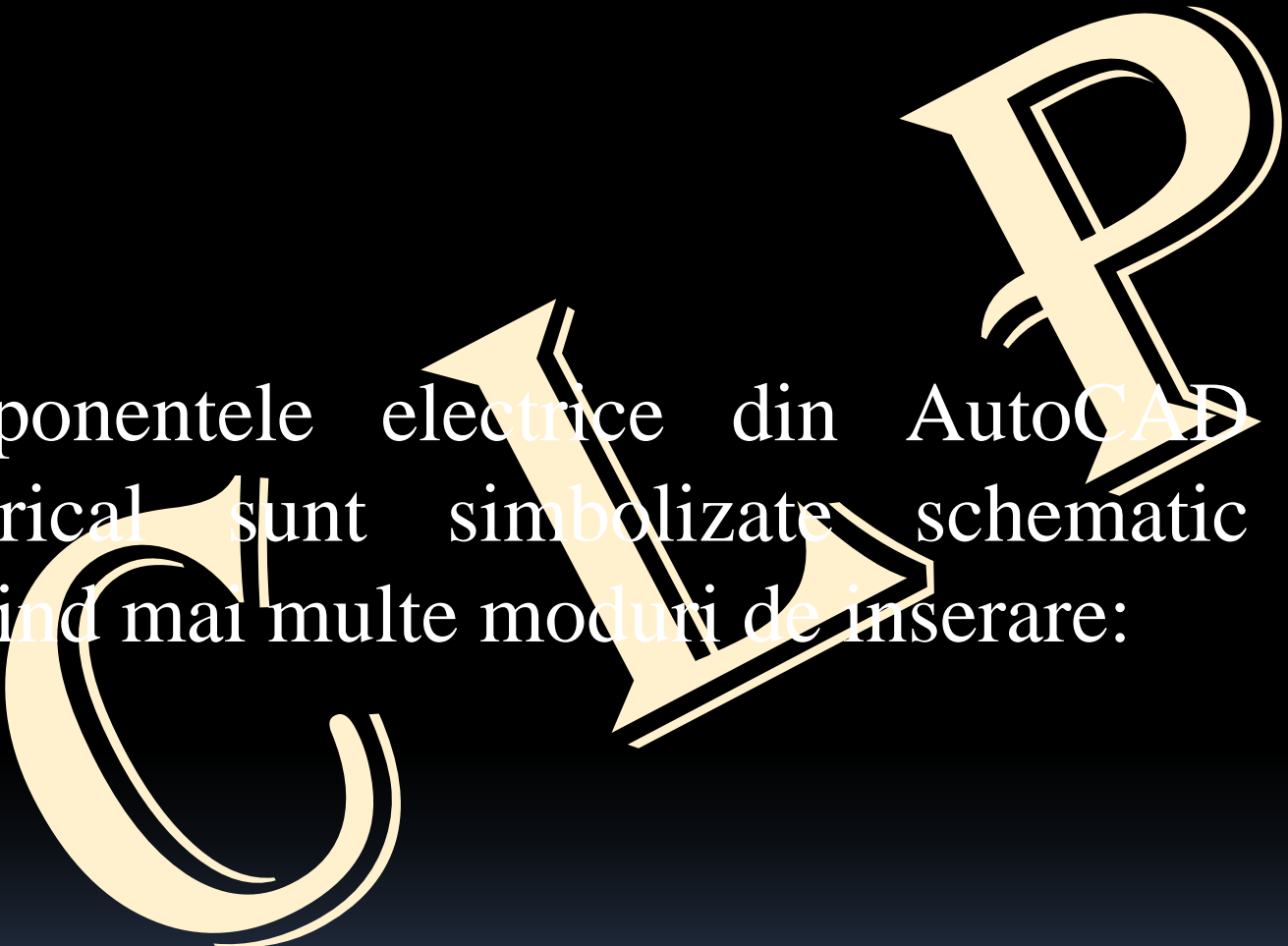
Move Up Move Down

>> All >> << << All

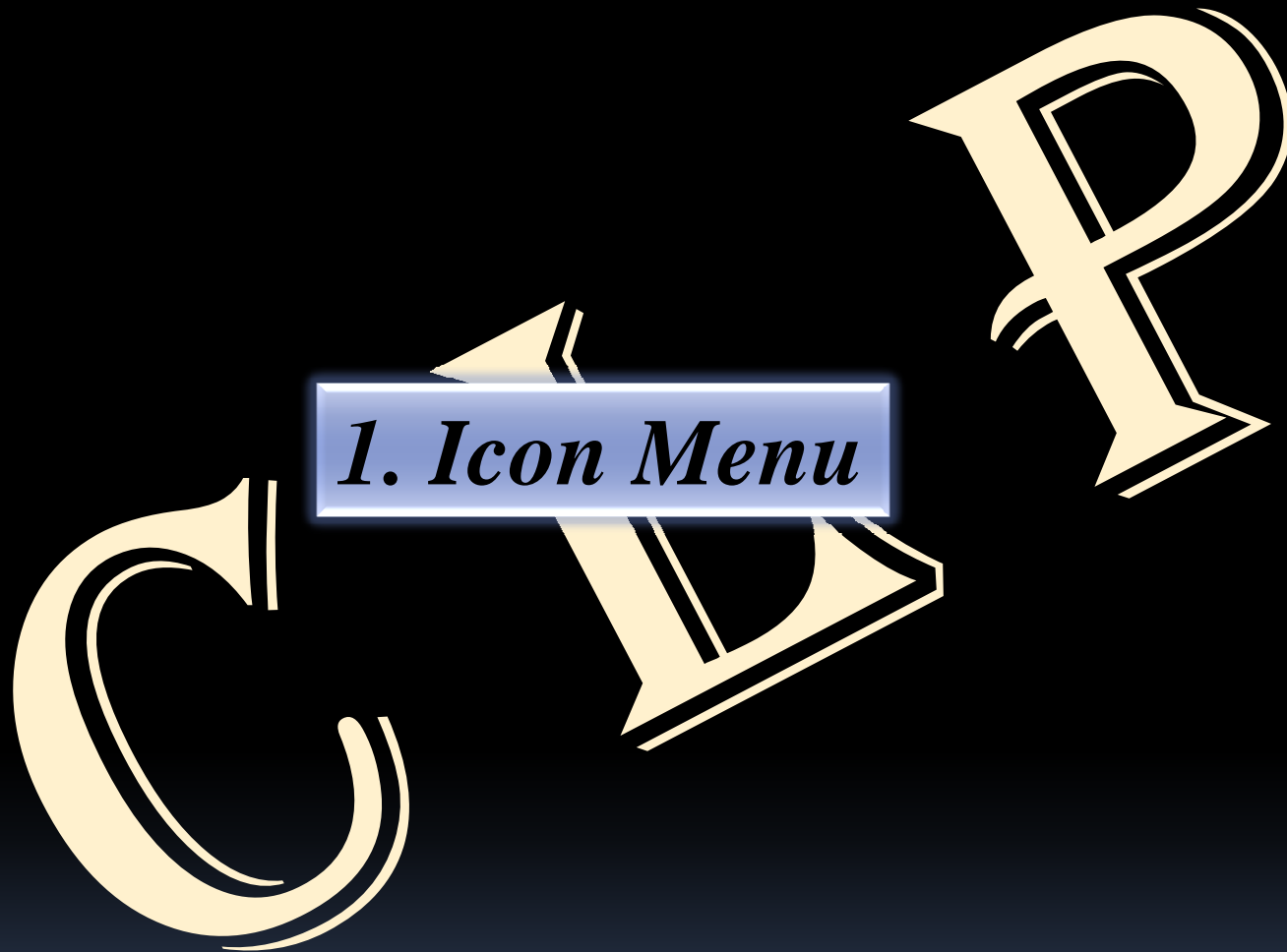
The image features large, stylized, yellow-outlined letters 'C', 'R', and 'P' in a decorative, calligraphic font. The letters are positioned across the top and middle of the slide, with the 'C' on the left, the 'R' in the center, and the 'P' on the right. The background is black, and a horizontal pink band with white text is overlaid across the middle of the letters.

C R P

III. Inserarea componentelor electrice

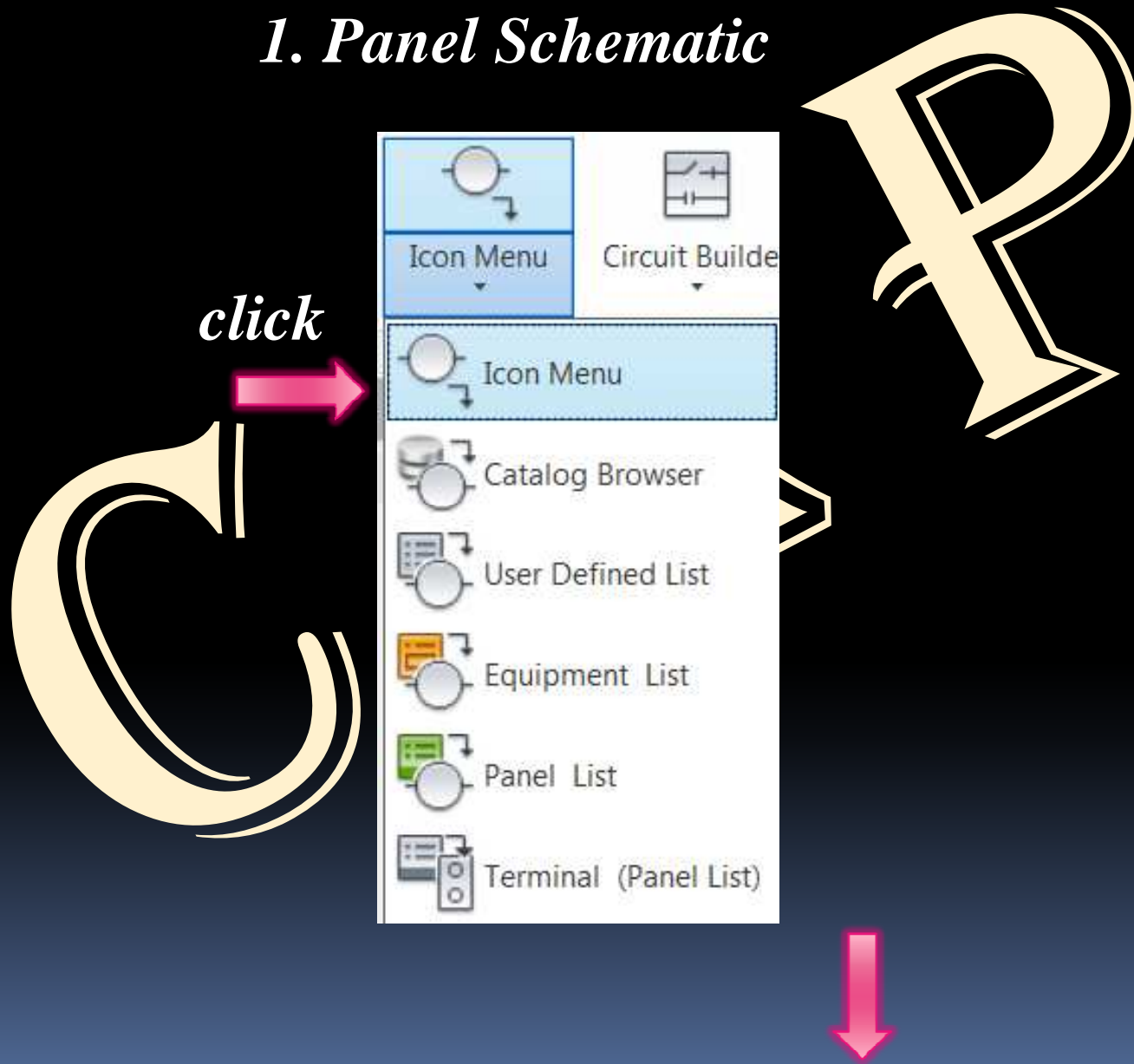
The image features three large, stylized yellow letters: 'C', 'L', and 'P'. The 'C' is on the left, the 'L' is in the center, and the 'P' is on the right. They are rendered in a bold, serif font with a slight shadow effect. The background is a gradient from black at the top to blue at the bottom.

Componentele electrice din AutoCAD
Electrical sunt simbolizate schematic
folosind mai multe moduri de inserare:

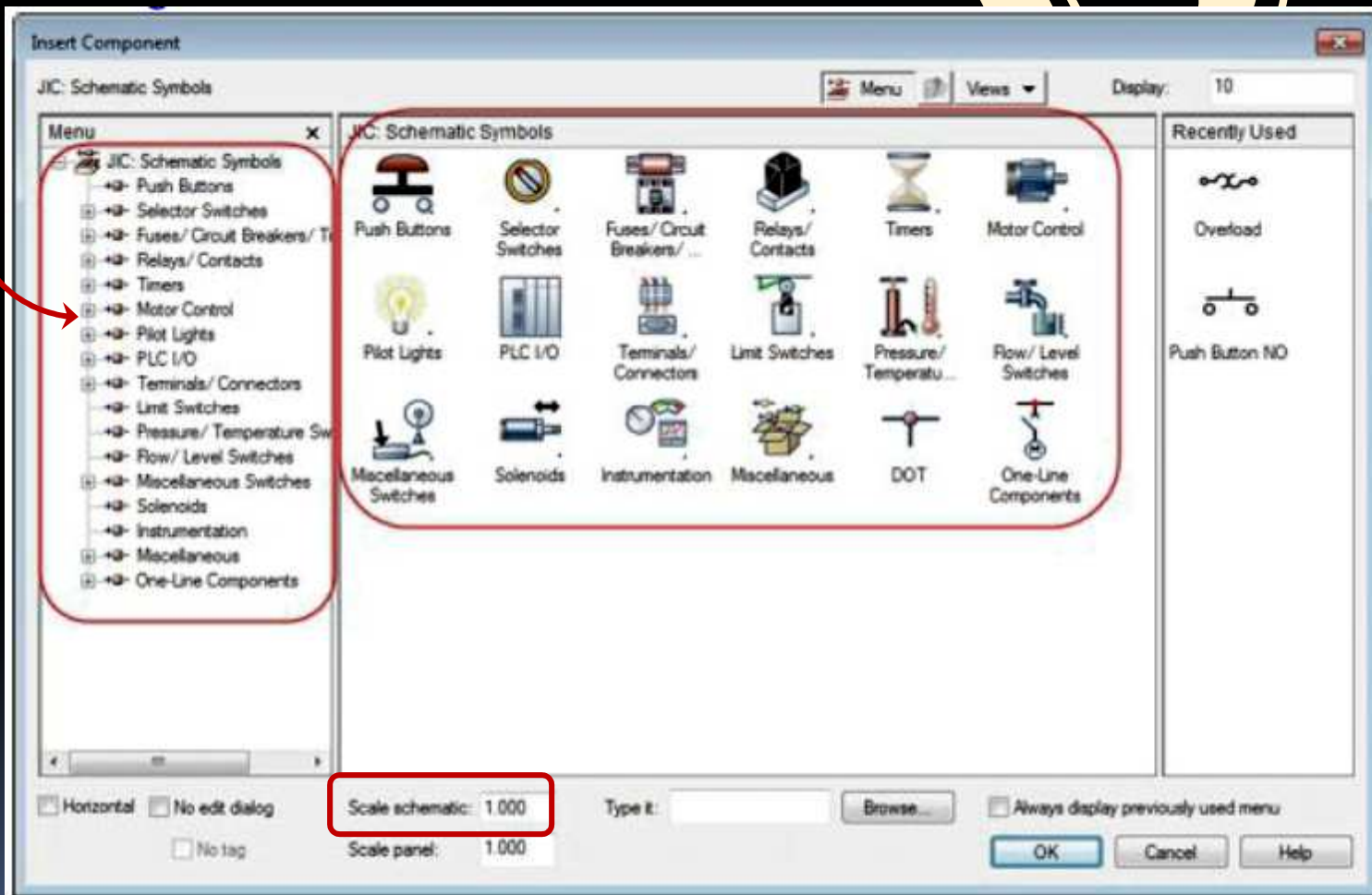


1. Icon Menu

1. Panel Schematic



2. Caseta Insert Component





*3. Selectarea categoriei și a subcategoriei;
click pe simbolul dorit.*



*4. Click pe zona de desen, pentru a plasa
simbolul componentei.*

5. Caseta Insert/Edit Component

Insert / Edit Component [X]

Component Tag
PB1 fixed
Use PLC Address
Schematic
Tags Used: Panel
External List
Options

Description
Line 1
Line 2
Line 3
List: Drawing Project Defaults Pick

Catalog Data
Manufacturer
Catalog
Assembly
Item Count
Next>> Lookup Previous
Drawing Project
Multiple Catalog
Catalog Check

Ratings
Rating
Show All Ratings

Cross-Reference
 Component override Setup
Reference NO NO/NC Setup
Reference NC

Installation code
Drawing Project

Location code
Drawing Project
Show/Edit Miscellaneous

Pins
1 < >
2 < >
X < >
< >

OK-Repeat OK Cancel Help

◆ Component Tag



Component Tag

PB1 fixed

Use PLC Address

Tags Used: **Schematic**

Panel

External List

Options

Connected PLC Address Not Found

This component is not tied directly to a PLC I/O point. You can manually select the PLC address text value to be used for the component tag.

Select List or Manual and pick the address text.

List I/O in use: **Drawing** **Project**

Manual Pick **Cancel**

All Tags in Use

Sort: Tag Drawing Sequence Description Family

Tag	Installation	Location	Sheet	FAM	Description 1,2,3
DS219		MCAB5	2	DS	
DS304		MCAB5	3	DS	
FS608		MACHINE	6	FS	
FS614		MACHINE	6	FS	TUB INLET TO HEATER FLOW N
FU207		MCAB5	2	FU	PURGE AIR FLOW NOT LOW (PA
FU211		MCAB5	2	FU	
FU215		MCAB5	2	FU	
FU221		MCAB5	2	FU	
FU222A		MCAB5	2	FU	
FU224		MCAB5	2	FU	
FU307		MCAB5	3	FU	

Show

- Show parent / stand-alone references
- Show child references
- Show all components for all families
- Show all panel components (#)
- Show one-line components (1-*)

Copy

- Descriptions
- Catalog values
- Installation/Location
- Ratings
- All of the above

Freshen **Copy Tag** **Calculate Next** **Cancel** **Help**

◆ *Catalog Data*

The image shows a software dialog box titled "Catalog Data". It contains several input fields and buttons. Red arrows point from the text on the right to the "Manufacturer", "Catalog", "Assembly", and "Count" fields. A yellow watermark with the letter "P" is overlaid on the right side of the dialog box.

Manufacturer	<input type="text"/>
Catalog	<input type="text"/>
Assembly	<input type="text"/>
Item	<input type="text"/>
Count	<input type="text"/>

Buttons: Next>>, Lookup, Previous, Drawing, Project, Multiple Catalog, Catalog Check

Ratings section:
Rating:
Show All Ratings

specificați numele producătorului

specificați valoarea numărului de catalog

specificați valoarea pentru codul de asamblare

specificați numărul elementului

specificați numărul de componente

Catalog Data

Manufacturer

Catalog

Assembly

Item Count

Next>>

Lookup Previous

Drawing Project

Multiple Catalog

Catalog Check

Ratings

Rating

Show All Ratings

Catalog Browser

Category: PB (Push buttons)

Search: AB "30.5mm EXTENDED" RED

Results

CATALOG	MANUFACTURER	DESCRIPTION	TYPE	STYLE	CONTACTS
800H-BR6A	AB	PUSH BUTTON - MOMENTARY, NEMA 4/4X	30.5mm EXTENDED	RED	1 NO 1 NC
800H-BR6A2	AB	PUSH BUTTON - MOMENTARY, NEMA 4/4X	30.5mm EXTENDED	RED	2 NO
800H-BR6B	AB	PUSH BUTTON - MOMENTARY, NEMA 4/4X	30.5mm EXTENDED	RED	2 NO 2 NC
800H-BR6C	AB	PUSH BUTTON - MOMENTARY, NEMA 4/4X	30.5mm EXTENDED	RED	4 NO 4 NC
800H-BR6D1	AB	PUSH BUTTON - MOMENTARY, NEMA 4/4X	30.5mm EXTENDED	RED	1 NO
800H-BR6H	AB	PUSH BUTTON - MOMENTARY, NEMA 4/4X	30.5mm EXTENDED	RED	3 NO 3 NC

Record Count: 9 Filter by WDBLKNAM value: PB11 Search Database: Primary

OK Cancel

Multiple Bill of Material Information

Sequential Code: D1 List

Catalog Data

Manufacturer: AB

Catalog: 871A-TS4-D1

Assembly:

Count Unit

Parts Catalog Lookup
c:\users\gau...\catalogs\default_cat.mdb

Table: CD List

Miscellaneous

Catalog Lookup

Catalog Check

Item Number:

Drawing: Find List

Project: Find List

OK Cancel Help

◆ Descriptions

Description

Line 1

Line 2

Line 3

List

desenul curent

proiectul curent

lista implicită

Descriptions (general): c:/users/viorel/appdata/roa.../r20.0/.../support/wd_desc.wdd

. Generic description file WD_DESC.WDD. Create project-specific
. description files using project name with ".WDD" extension
. This file is accessed from the "Defaults" radio button under EDIT or
. INSERT component "ListDescriptions" when project-specific WDD
. file is not found.
. Use the "|" character to show the break point for a second line
. Anything after "|" character is ignored

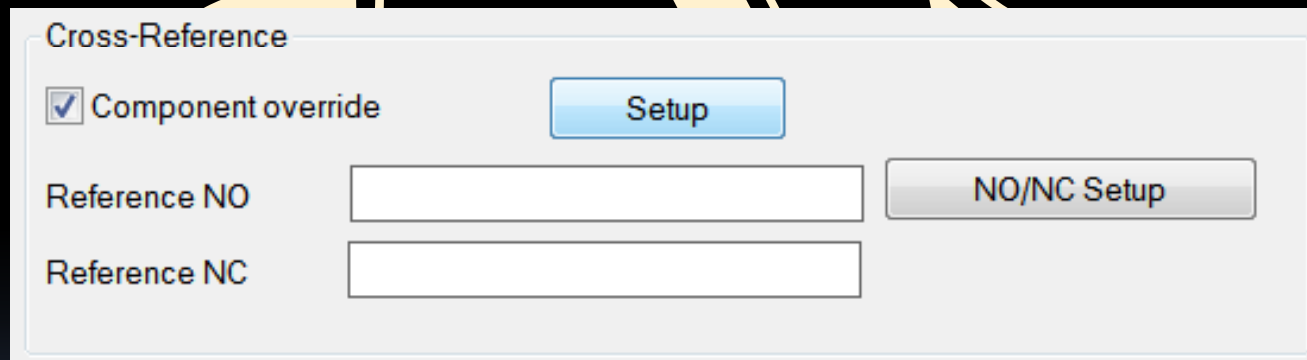
. Generic descriptions for RELAYS
MASTER CONTROL|RELAY
SYSTEM|ENABLED

. Generic descriptions for PUSH BUTTONS
START
STOP
RESET
SYSTEM|RESET
EMERGENCY|STOP
RUN
JOG
OPEN
CLOSE

Highlight entry above and pick OK,
or select Pick File to select a different file and description list

◆ *Cross-Reference*

Realizează conectarea pinilor componenteii curente, cu pinii celorlalte componente ale desenului/proiectului.



The image shows a software dialog box titled "Cross-Reference". It features a checked checkbox labeled "Component override" with a blue "Setup" button to its right. Below this, there are two text input fields: "Reference NO" and "Reference NC". To the right of these fields is a grey "NO/NC Setup" button.

Field	Value
Component override	<input checked="" type="checkbox"/>
Reference NO	<input type="text"/>
Reference NC	<input type="text"/>

Component override Setup

Cross-Reference Component Override

Cross-Reference Format
Same Drawing
%N Default "%N" "%S-%N"

Between Drawings
%N Same "%S-%N"

Component Cross-Reference Display

Text Format Setup
 Graphical Format
 Table Format

OK Cancel Help

NO/NC Setup

Maximum NO/NC counts and/or allowed Pin numbers

Set value to 0 for none allowed, blank for undefined

Maximum NO contact count [] [←] [→]

Maximum NC contact count [] [←] [→]

Maximum convertible NO/NC [] [←] [→]

Maximum undefined type "4" [] [←] [→]

Option: Pre-defined Pin number combinations

Contact and pin number format is: [Retrieve from peer]
type.pin.pin.type.pin.pin.type.pin.pin....
where type 1 = NO, 2 = NC, 0 = convertible,
3 = Form-C, 4 = Other

ex: 2NO (pins 2,3 and 4,5) and 1NC (pins 6,8) would be
1,2,3;1,4,5;2,6,8

ex: 2 Form-C, pins 7com,4no,1nc and 8com,5no,2nc would be
3,7,4,1;3,8,5,2 (Form-C order is 3.COM,NO,NC)

Pin List []

OK Cancel

◆ *Installation Code and Location Code*

Installation code

linia circuitului pentru componentă

Drawing Project

Location code

locația fizică a componentei

Drawing Project

Show/Edit Miscellaneous

OK-Repeat OK

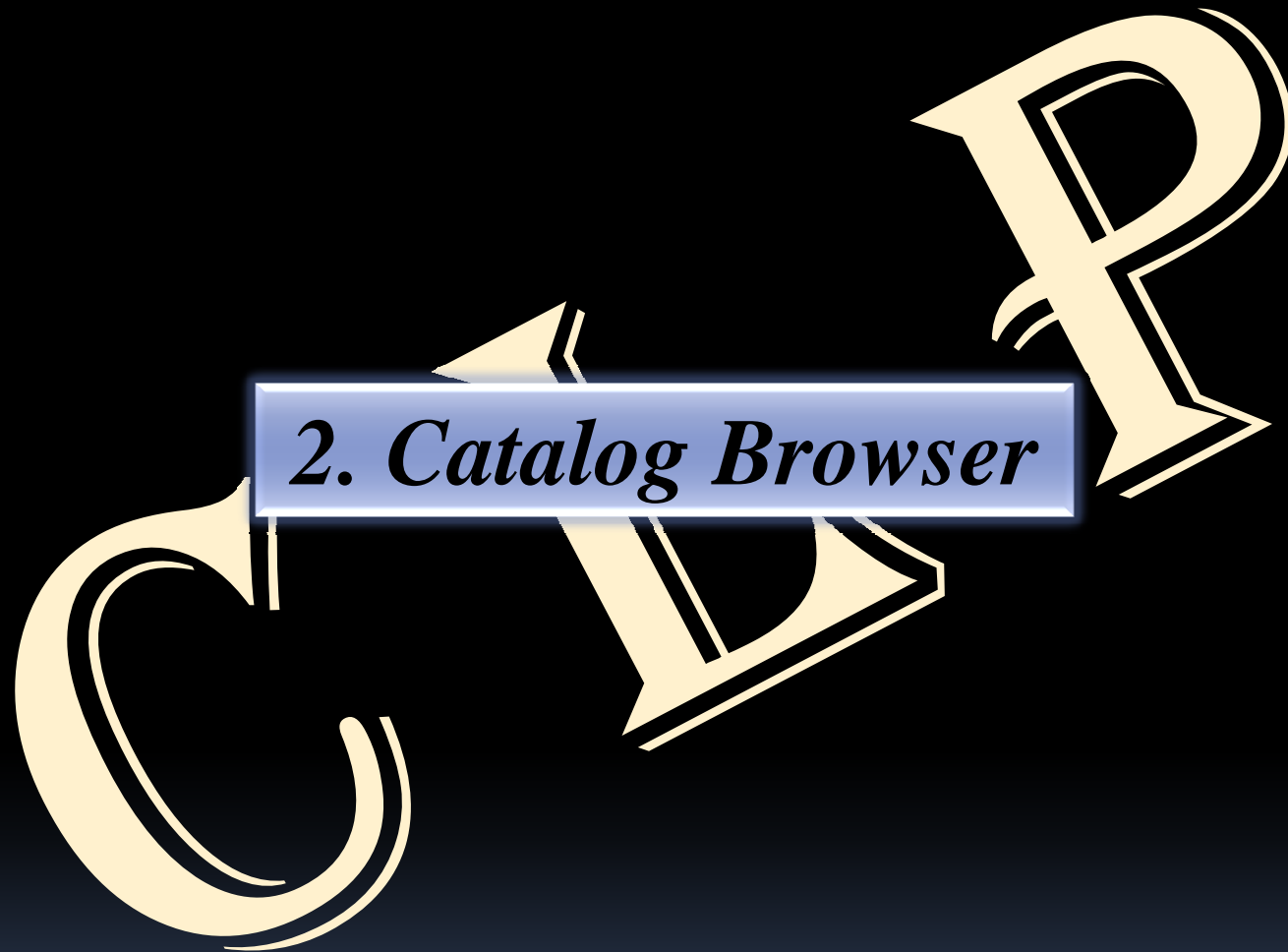
◆Pins

Se numerotează pinii componentei,
pentru a putea fi conectați cu alte componente.

Pin Label	Input Field	Left Arrow	Right Arrow
1	<input type="text"/>	<	>
2	<input type="text"/>	<	>
X	<input type="text"/>	<	>

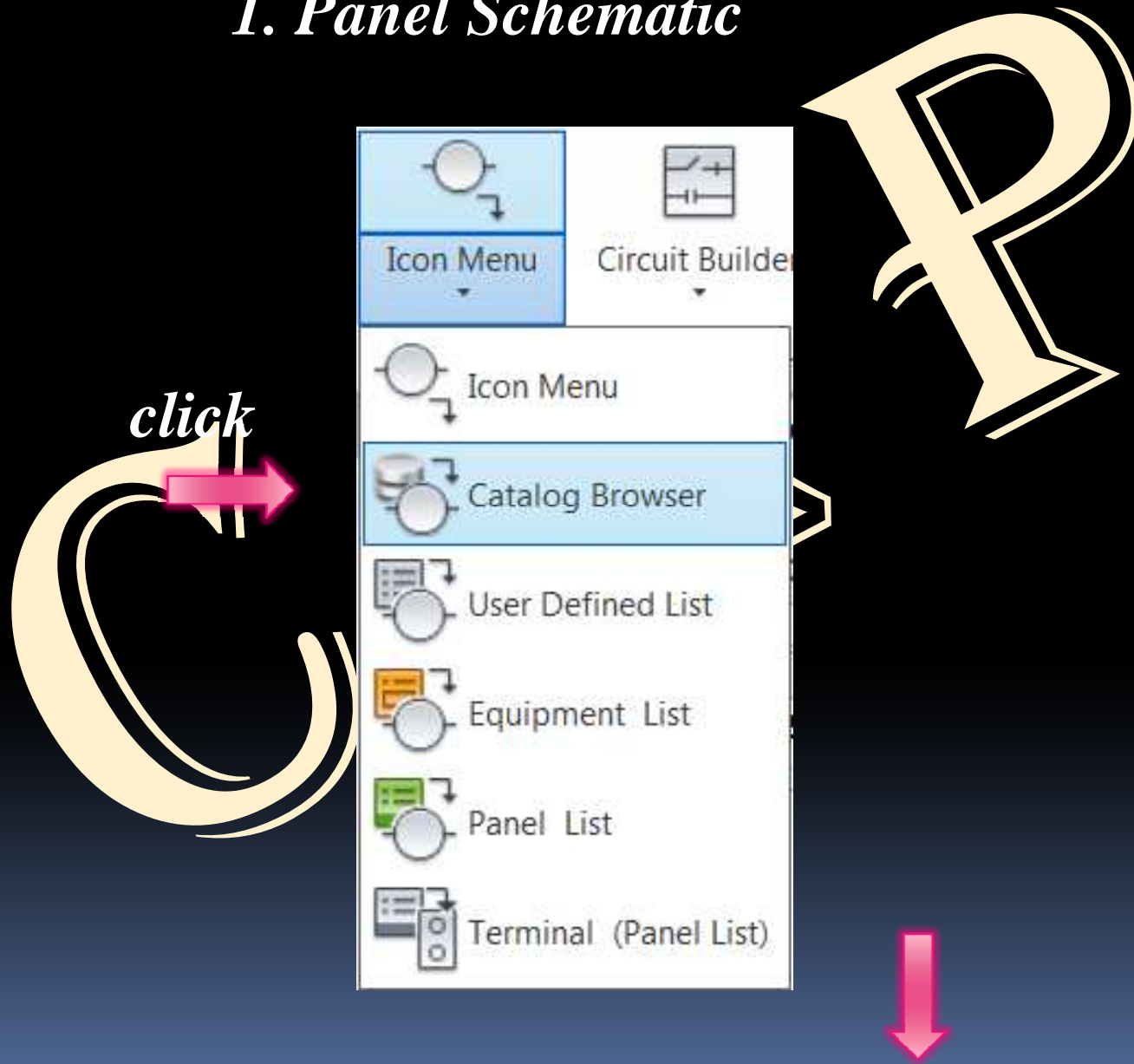
< >

Cancel Help

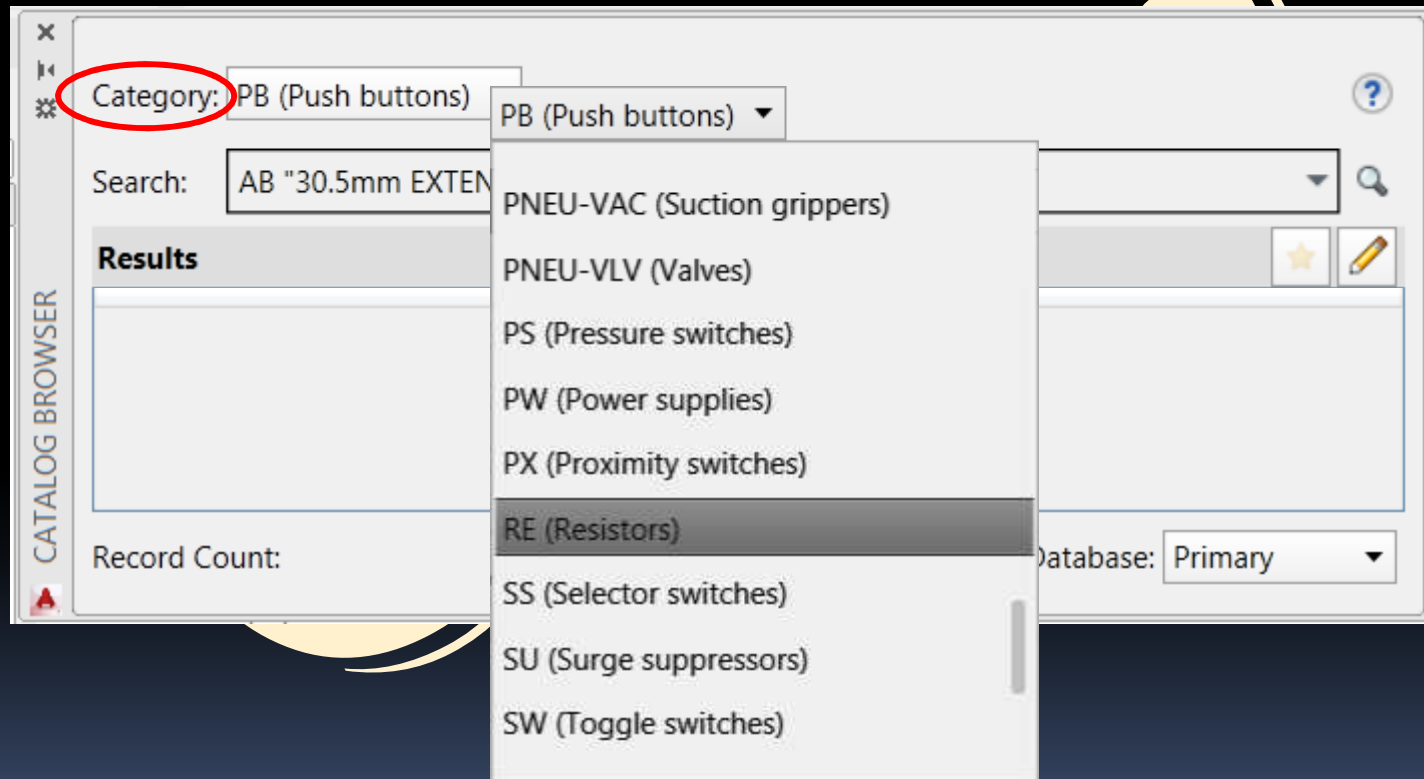


2. Catalog Browser

1. Panel Schematic



2. Catalog Browser



3. Category

The screenshot shows a 'CATALOG BROWSER' window with the following details:

- Category: RE (Resistors)
- Search: TELEMECANIQUE PROTECTED
- Record Count: (blank)

The search results table is as follows:

LOG	MANUFACTURER	DESCRIPTION
68711	TELEMECANIQUE	BRAKING RESISTOR - ENCLOSED - IP 23 - 5.74 ohm - 28.4 KV
68712	TELEMECANIQUE	BRAKING RESISTOR - ENCLOSED - IP 23 - 3.8 ohm - 32 KW
68713	TELEMECANIQUE	BRAKING RESISTOR - ENCLOSED - IP 23 - 2.6 ohm - 47 KW
68714	TELEMECANIQUE	BRAKING RESISTOR - ENCLOSED - IP 23 - 2.44 ohm - 59 KW
68715	TELEMECANIQUE	BRAKING RESISTOR - ENCLOSED - IP 23 - 4.5 ohm - 50 KW
68716	TELEMECANIQUE	BRAKING RESISTOR - ENCLOSED - IP 23 - 3.6 ohm - 70 KW
68717	TELEMECANIQUE	BRAKING RESISTOR - ENCLOSED - IP 23 - 2.87 ohm - 88 KW
68718	TELEMECANIQUE	BRAKING RESISTOR - ENCLOSED - IP 23 - 2.48 ohm - 100 KW
687537	TELEMECANIQUE	BRAKING RESISTOR - ENCLOSED - IP 20 - 4.2 ohm - 37 KW
687575	TELEMECANIQUE	BRAKING RESISTOR - ENCLOSED - IP 20 - 4.2 ohm - 75 KW

Record Count: 30

Search Database: Primary

4. Selectare componentă

CATALOG BROWSER

Category: RE (Resistors) ?

Search: TELEMECANIQUE PROTECTED 🔍

Results ★ ✎

LOG	MANUFACTURER	DESCRIPTION
68711	TELEMECANIQUE	RESISTOR - ENCLOSED - IP 23 - 5.74 ohm - 28.4 KW
68712	TELEMECANIQUE	BRAKING RESISTOR - ENCLOSED - IP 23 - 3.8 ohm - 32 KW
68713	TELEMECANIQUE	BRAKING RESISTOR - ENCLOSED - IP 23 - 2.6 ohm - 47 KW
68714	TELEMECANIQUE	BRAKING RESISTOR - ENCLOSED - IP 23 - 2.44 ohm - 59 KW
68715	TELEMECANIQUE	BRAKING RESISTOR - ENCLOSED - IP 23 - 4.5 ohm - 50 KW
68716	TELEMECANIQUE	BRAKING RESISTOR - ENCLOSED - IP 23 - 3.6 ohm - 70 KW
68717	TELEMECANIQUE	BRAKING RESISTOR - ENCLOSED - IP 23 - 2.87 ohm - 88 KW
68718	TELEMECANIQUE	BRAKING RESISTOR - ENCLOSED - IP 23 - 2.48 ohm - 100 KW
687537	TELEMECANIQUE	BRAKING RESISTOR - ENCLOSED - IP 20 - 4.2 ohm - 37 KW
687575	TELEMECANIQUE	BRAKING RESISTOR - ENCLOSED - IP 20 - 4.2 ohm - 75 KW

Record Count: 30

Search Database: Primary ▼

Click



Insert Component

JIC: Schematic Symbols

Menu

- Motor Control
- Pilot Lights
- PLC I/O
- Terminals/ Connectors
 - Limit Switches
 - Pressure/ Temperature Switches
 - Flow/ Level Switches
- Miscellaneous Switches
 - SPDT Switches
- Solenoids
- Instrumentation
- Miscellaneous
 - Electronics**
 - Cable Markers
 - Receptacles
 - Shields (Non Intelligent)
 - Splice symbols
 - User Circuits
 - Generic Boxes
 - Stand-Alone Cross-Reference Symbols
 - Wire Arrows Reference only
- One-Line Components
 - Connector
 - Motor Control
 - Transformer
 - Terminal
 - Cable Marker
 - Bus Tap

JIC: Electronics

Fixed Resistor	Fixed Resistor (Box)	Fixed Resistor with Pins	Fixed Resistor (Box) with Pins	Variable Resistor	Variable Resistor
Variable Resistor	Variable Resistor	Diode	Diode	Diode with Pins	Diode with Pins
Zener Diode	Zener Diode	Zener Diode with Pins	Zener Diode with Pins	Capacitor	Capacitor with Pins

Recently Used

- Push Button NO

Scale schematic: 3.000 Type it:

Assign Symbol To Catalog Number

The selected symbol is not mapped to the selected catalog number.

→ Map symbol to catalog number

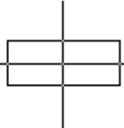
→ Do not map symbol to catalog number

Do not show me this again

5. Plasare componentă

[-] [Top] [2D Wireframe]

REVISIONS				
XXX13	XXX14	XXX15	XXX16	XXX17



		PROJDESC		
		TITLE1		
		TITLE2		
	SIZE XXX4	FSCM NO XXX5	DRAWING NUMBER XXX6	REV XXX7
	SCALE XXX8		SHEET XXX9	

Command: nil
Command:

6. Caseta Insert/Edit Component



Insert / Edit Component

Component Tag: R1 fixed

Use PLC Address

Tags Used: Schematic Panel External List Options

Catalog Data: Manufacturer: TELEMECANIQUE Catalog: VW3A58703 Assembly: Item: Count: Next>> Lookup Previous Drawing Project Multiple Catalog Catalog Check

Ratings: Rating: 40W Show All Ratings

Description: Line 1: 300 Line 2: Line 3: List: Drawing Project Defaults Pick

Cross-Reference: Component override Setup Reference NO: Reference NC: NO/NC Setup

Installation code: Drawing Project

Location code: Drawing Project

Show/Edit Miscellaneous

OK-Repeat OK Cancel He

[Top] 2D Wireframe

300

R1

1 40W 2

REVISIONS				
XXXX12	XXXX14	XXXX15	XXXX16	XXXX17

PROJDESC				
TITLE1				
TITLE2				
SIZE	FSCM NO	DRAWING NUMBER	REV	
xxx#	xxx#	xxx#	xxx#	xxx#
SCALE	xxx#	SHEET	xxx#	

Select a block: 3.740058,5.971601,0.000000

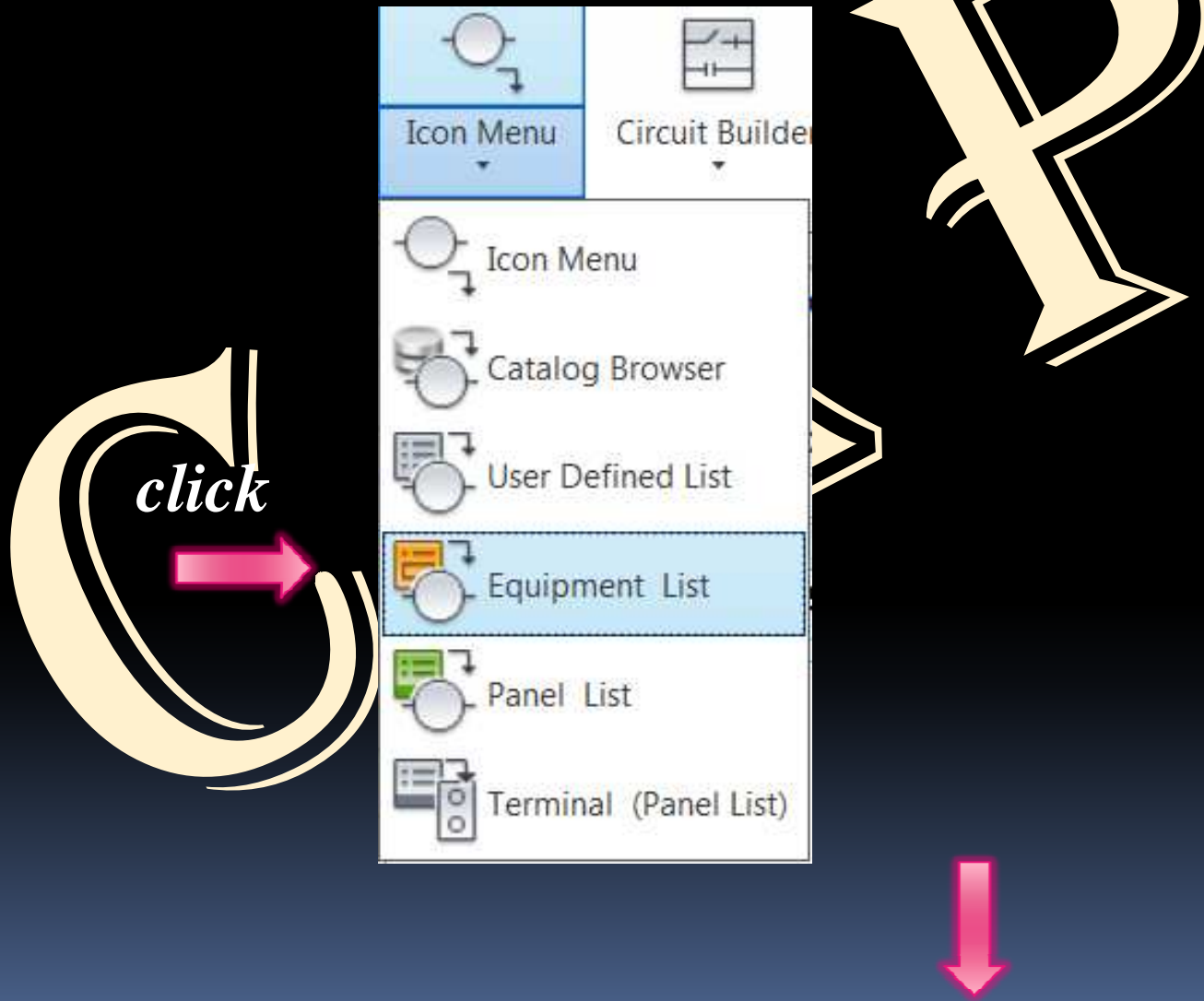
Command: *Cancel*

Type a command

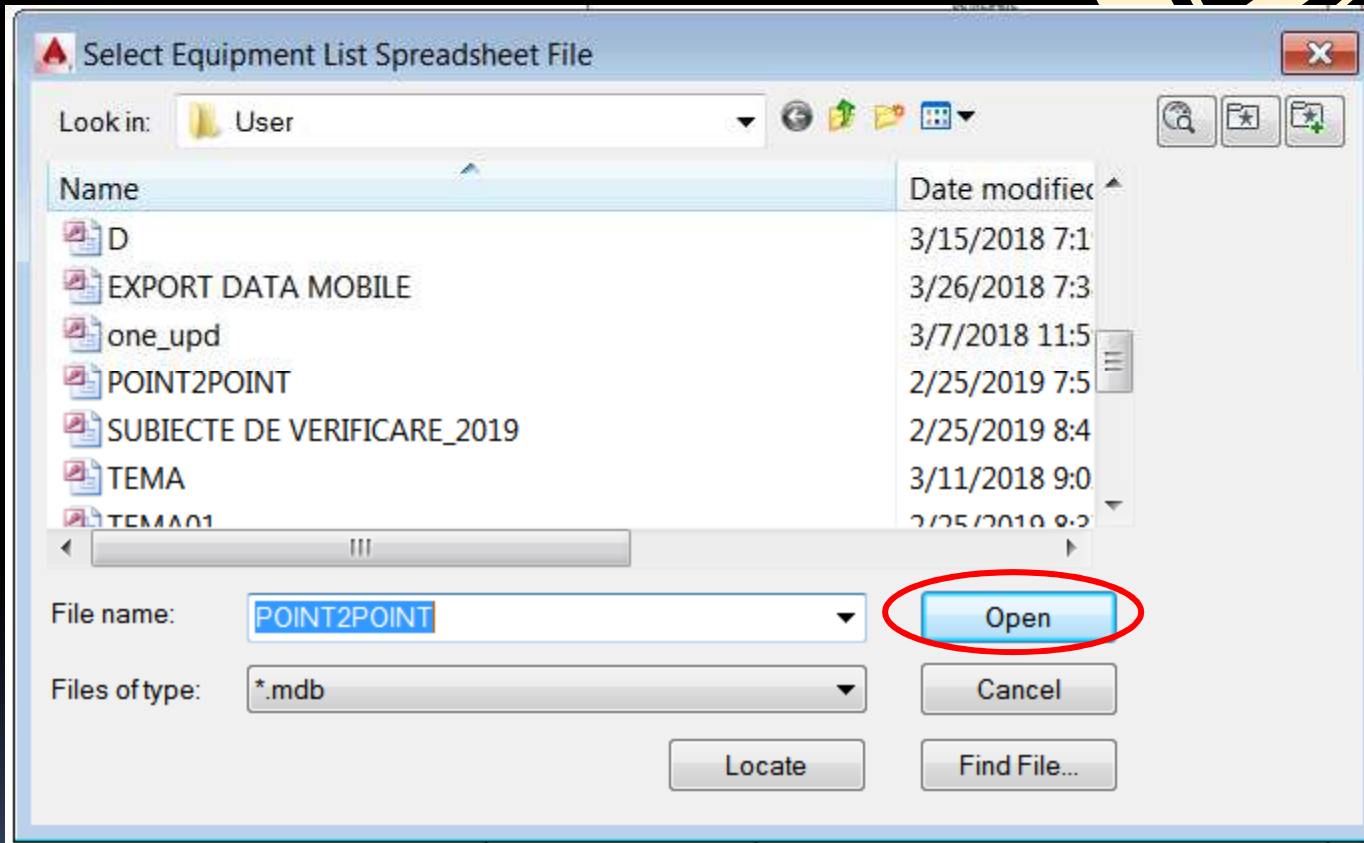
The image features a large, stylized graphic of the letters 'C', 'L', and 'P' in a bright yellow color with a white outline. The letters are arranged in a diagonal line from the bottom-left to the top-right. A semi-transparent blue rectangular box with a white border is positioned over the middle of the 'L' and 'P'. Inside this box, the text '3. Equipment List' is written in a black, italicized serif font. The background is a dark blue gradient that transitions from black at the top to a lighter blue at the bottom.

3. Equipment List

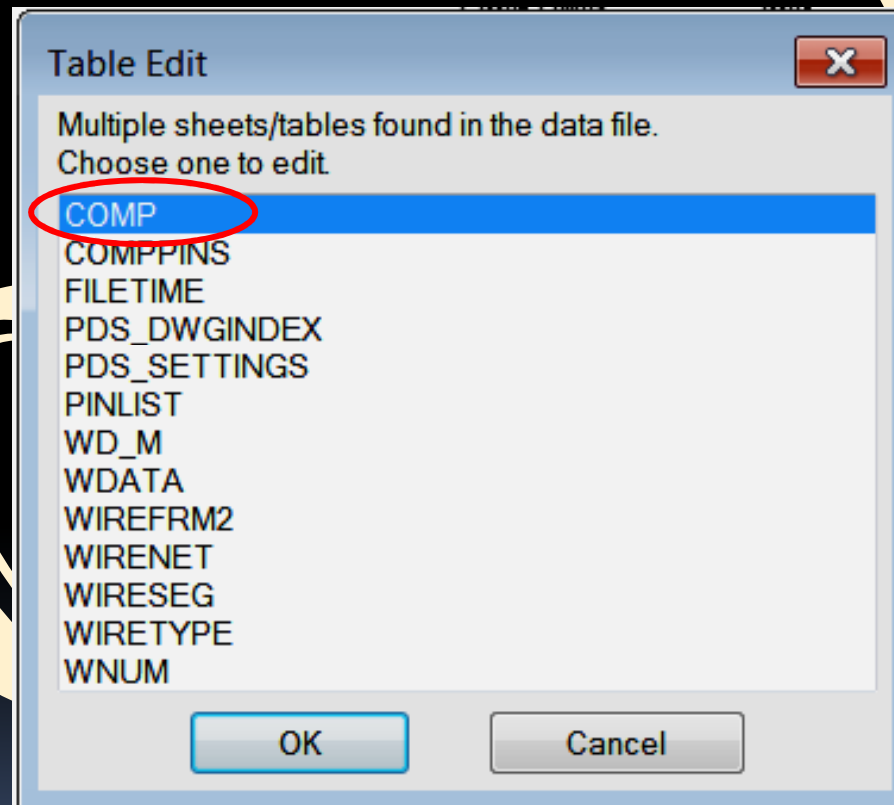
1. Panel Schematic



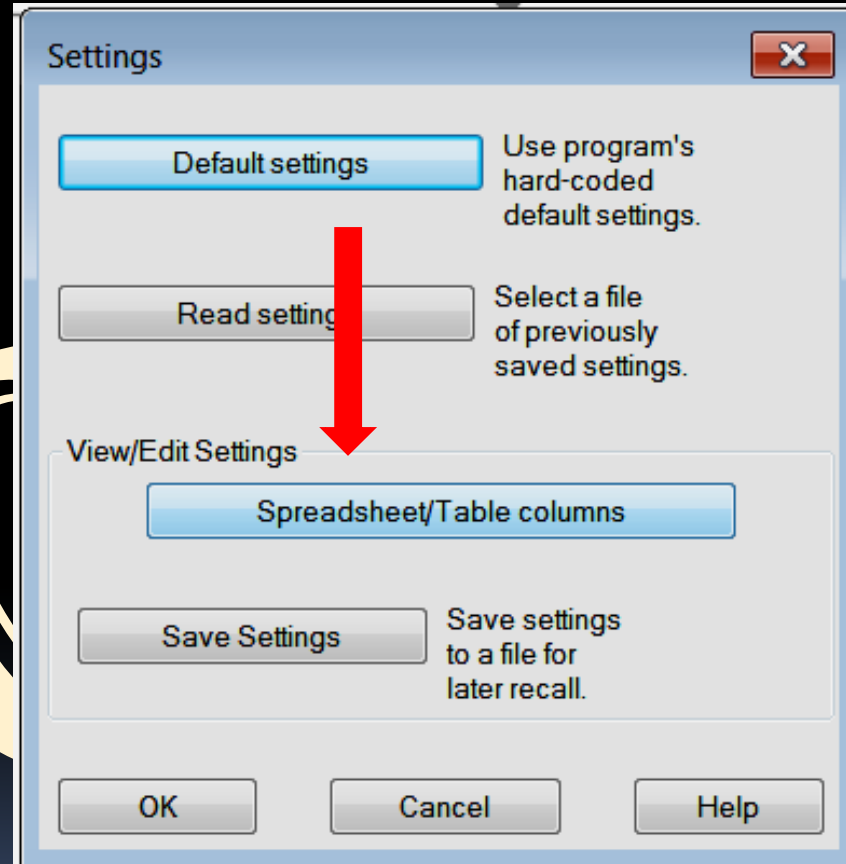
2. Select Equipment List Spreadsheet File



3. Table Edit



4. Settings



5. Equipment List Spread Settings



Equipment List Spreadsheet Settings

Col1	Col2	Col3	Col4	Col5	Col6	Col7	Col8	Col9
DWGIX	PAR1_CHL	FAMILY	TAGNAME	DESC1	DESC2	DESC3	REF	MFG
1	1	PJ	J5	MOLEX_SL_				
1	1	CBL	DS1	D SUB CAB				ALPH
1	2	CBL	DS1	D SUB CAB				
1	2	CBL	DS1	D SUB CAB				
1	2	CBL	DS1	D SUB CAB				

Assign column numbers to data categories below

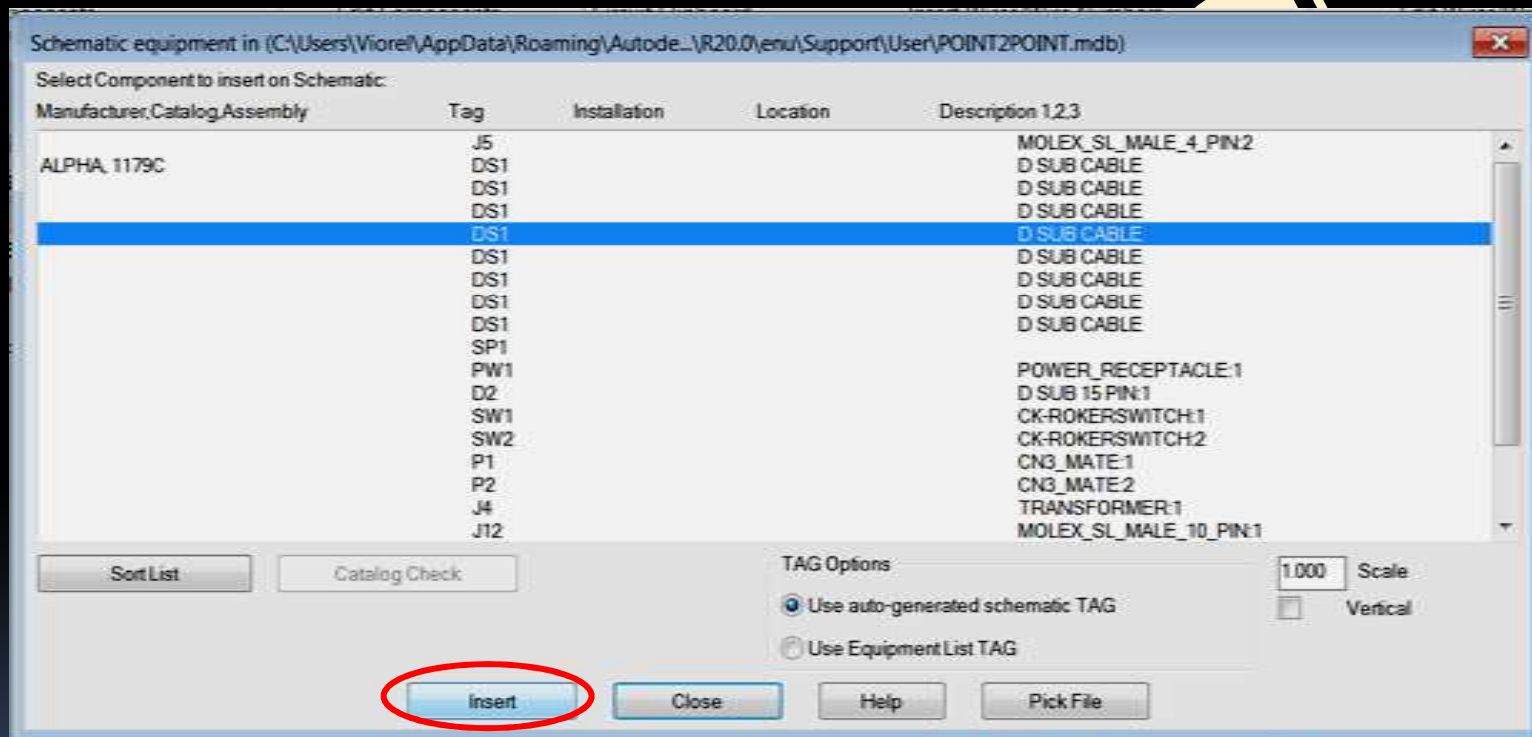
9	Manufacturer	36	Item number	5	Description 1
10	Catalog	4	Component Tag	6	Description 2
11	Assembly Code	16	Installation	7	Description 3
35	Catalog lookup table name	17	Location		

OK Cancel More

După modificări ale datelor pe coloane



6. Schematic equipment in



7. Insert

Insert PW1

Block Name	Comment
none found	

Icon Menu Select comp

Copy Component Insert "just li

OK Cancel

Insert Component

JIC: Schematic Symbols

Menu

- JIC: Schematic Symbols
 - Push Buttons
 - Selector Switches
 - Fuses/ Circuit Breakers/ Transformers
 - Circuit Breakers/ Disconnects
 - Relays/ Contacts
 - Timers
 - Motor Control
 - Pilot Lights
 - PLC I/O
 - Terminals/ Connectors
 - Limit Switches
 - Pressure/ Temperature Switches
 - Flow/ Level Switches
 - Miscellaneous Switches
 - Solenoids
 - Instrumentation
 - Miscellaneous
 - One-Line Components

JIC: Time Delay Relays

ON Delay Coil	ON Delay Coil with Pins	ON Delay Starter	ON Delay NO - TC	ON Delay NC - TO	ON Delay NO - TC with Pins
ON Delay NC - TO with Pins	Instantaneous NO	Instantaneous NC	Instantaneous NO with Pins	Instantaneous NC with Pins	OFF Delay Timers

Recently Used

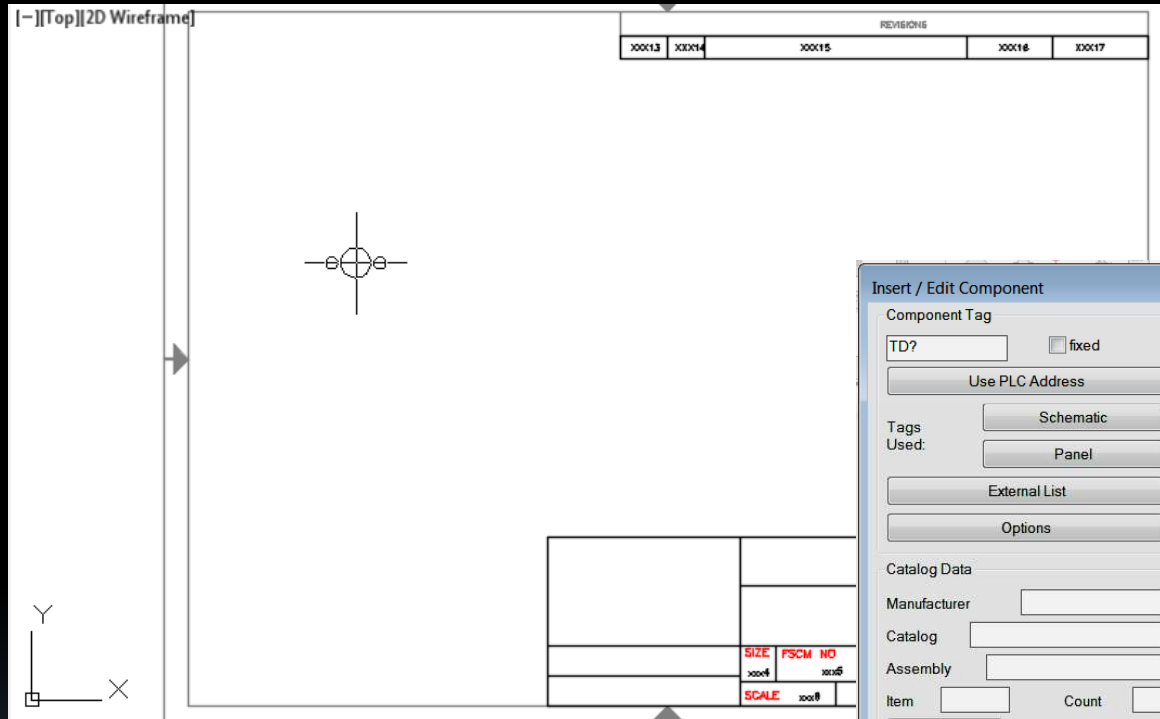
- Circuit Breaker 1 Pole

Vertical No edit dialog Scale schematic: 1.000 Type it: Browse... Always display previously used menu

No tag Scale panel: 1.000

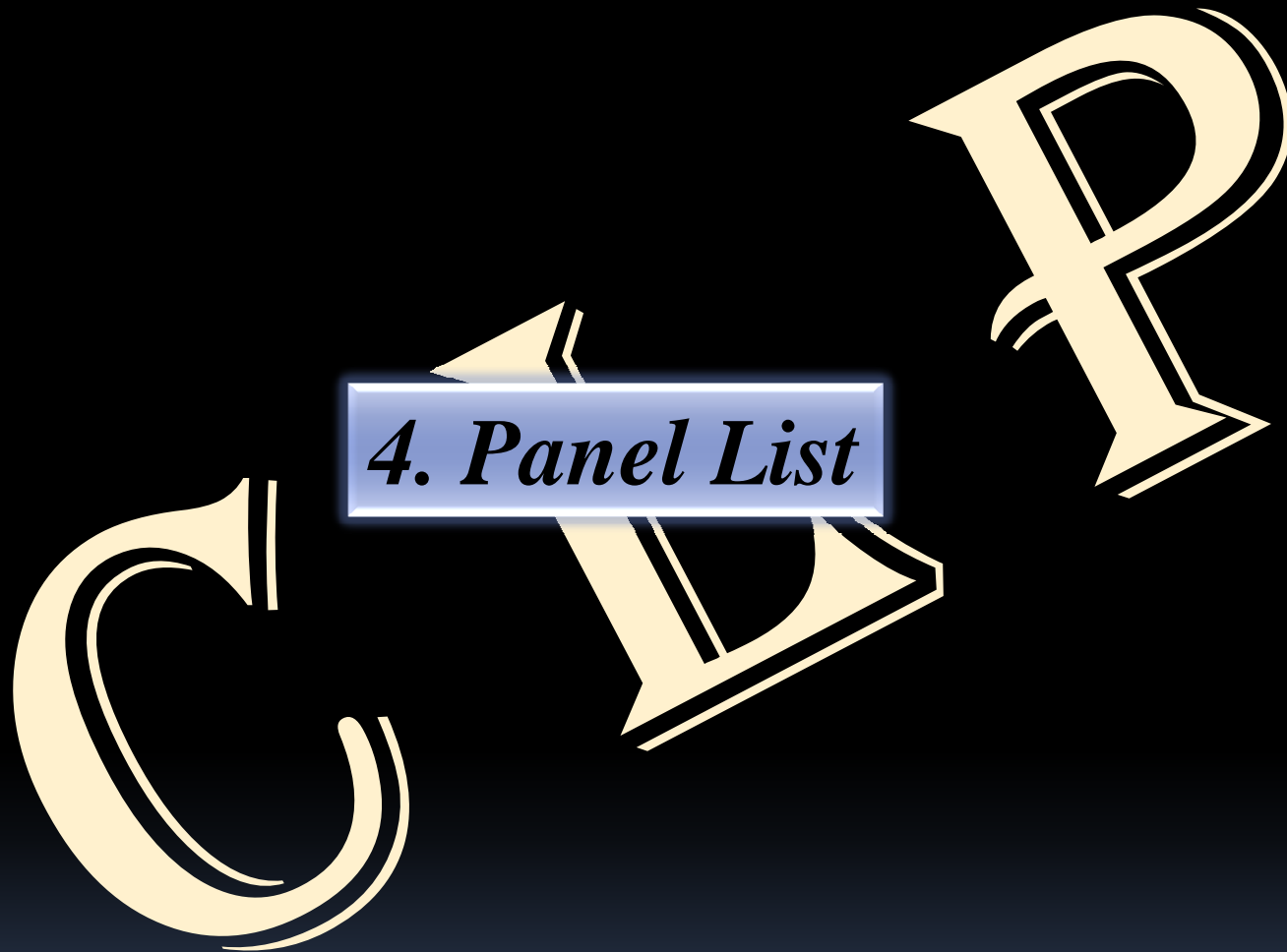
OK Cancel Help

8. Plasarea componentei



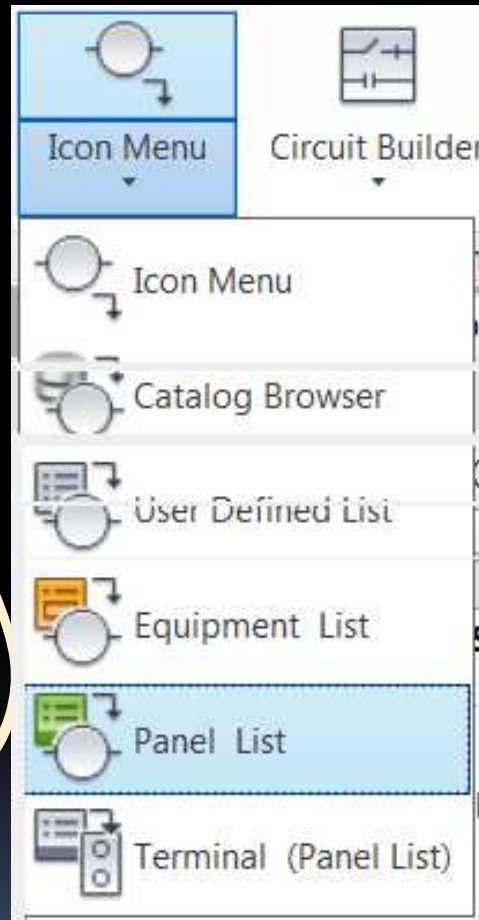
The 'Insert / Edit Component' dialog box is shown, containing the following sections:

- Component Tag:** TD? fixed, Use PLC Address, Tags Used: Schematic, Panel, External List, Options.
- Description:** Line 1, Line 2, Line 3, List: Drawing, Project, Defaults, Pick.
- Cross-Reference:** Component override Setup, Reference NO, NO/NC Setup, Reference NC.
- Installation code:** Drawing, Project, Pins (1, 2, X) with navigation arrows.
- Location code:** Drawing, Project, Show/Edit Miscellaneous.
- Ratings:** Rating, Show All Ratings.
- Buttons:** OK-Repeat, OK, Cancel, Help.

The image features three large, stylized letters 'C', 'L', and 'P' in a decorative, calligraphic font. The letters are yellow with a white outline and are set against a black background. The 'C' is on the left, the 'L' is in the center, and the 'P' is on the right. A blue rectangular box with a white border is positioned over the 'L', containing the text '4. Panel List' in a black, italicized serif font.

4. Panel List

1. Panel Schematic



2. Panel Layout List



The screenshot displays a software interface with a 'Panel Layout List --> Schematic Components Insert' dialog box. The dialog box is divided into several sections:

- Extract component list for:**
 - Project
 - Active drawing
 - save list to external file
- Installation Codes to extract:**
 - All
 - Blank
 - Named Installation
 - Installation:
 - List:
- Location Codes to extract:**
 - All
 - Blank
 - Named Location
 - Location:
 - List:

At the bottom of the dialog box are buttons for **OK**, **Cancel**, and **Help**. A **Browse** button is also present with the text 'use external file' next to it.

The background shows a schematic drawing with various components and a table of wire numbers. The table has columns for 'WIRE NO.', 'WIRE NO.', 'WIRE NO.', and 'WIRE NO.'. The schematic includes components like 'CONVEYOR MOTOR', 'HYDRAULIC MOTOR', and 'REWORKING MOTOR'. The Autodesk logo is visible in the bottom right corner of the drawing area.

Active drawing

Panel Components (Drawing - for all installations, for all locations)

Select Panel footprint reference to insert on Schematic:

x	Tag	Installation	Location	Manufacturer,Catalog,Assembly	Description 1,2,3	Block	Sheet
-	207CBL					MMPTAG	1
-	211CBL					MMPTAG	1
-	215CBL					MMPTAG	1
-	DS207					MMPTAG	1
-	DS211					MMPTAG	1
-	DS214					MMPTAG	1
-	MOT212				HYDRAULIC MOTOR	MMPTAG	1
-	MOT215				INDEXING MOTOR	MMPTAG	1
-	RECPT229					MMPTAG	1
-	MOT208		FLOOR		CONVEYOR MOTOR	MMPTAG	1
-	RECPT226		MCAB5			WD_PTAG	1
-	RECPT227		MCAB5			WD_PTAG	1
-	LS406	MACHINE		MS, MS-50L	RAM 01 RETRACTED	WD_PTAG	1
-	LS407	MACHINE		MS, MS-50L	RAM 02 RETRACTED	WD_PTAG	1
-	LS408	MACHINE		MS, MS-50L	RAM 03 RETRACTED	WD_PTAG	1
-	TAS607			SQD, 9025-GXW5A12	TUB OUTLET TEMPERATURE NOT L...	E1310001	1
-	TAS623	MACHINE		SQD, 9025-GXW5A12	FLAME DETECTED NO 1	E1310001	1

Sort List Reload Mark Existing Display: Show All Hide Existing Catalog Check TAG Options: Use auto-generated schematic TAG Use panel footprint TAG Scale: 1.000 Vertical:

Insert Close Help Pick File

Project

Select Drawings to Process

Drawing Description

Ref	Subfolder	Sect...	Sub-Se...	Project Drawing List
				C:\Users\Viorel\Documents\...utorial\Aegs\Wire numbers\AEGS...
				C:\Users\Viorel\Documents\AcadE 2015\AeData\proj\Aegs\AE...
				C:\Users\Viorel\Documents\AcadE 2015\AeData\proj\Aegs\AE...
				C:\Users\Viorel\Documents\AcadE 2015\AeData\proj\Aegs\AE...
				C:\Users\Viorel\Documents\AcadE 2015\AeData\proj\Aegs\AE...
				C:\Users\Viorel\Documents\AcadE 2015\AeData\proj\Aegs\AE...
				C:\Users\Viorel\Documents\AcadE 2015\AeData\proj\Aegs\AE...
				C:\Users\Viorel\Documents\AcadE 2015\AeData\proj\Aegs\AE...
				C:\Users\Viorel\Documents\AcadE 2015\AeData\proj\Aegs\AE...

Do All Process Reset Un-select by Section/sub-section by Subfolder

Ref	Subfolder	Sect...	Sub-Se...	Project Drawing List
-----	-----------	---------	-----------	----------------------

Select from the top list insert into the bottom list OK Cancel Help View

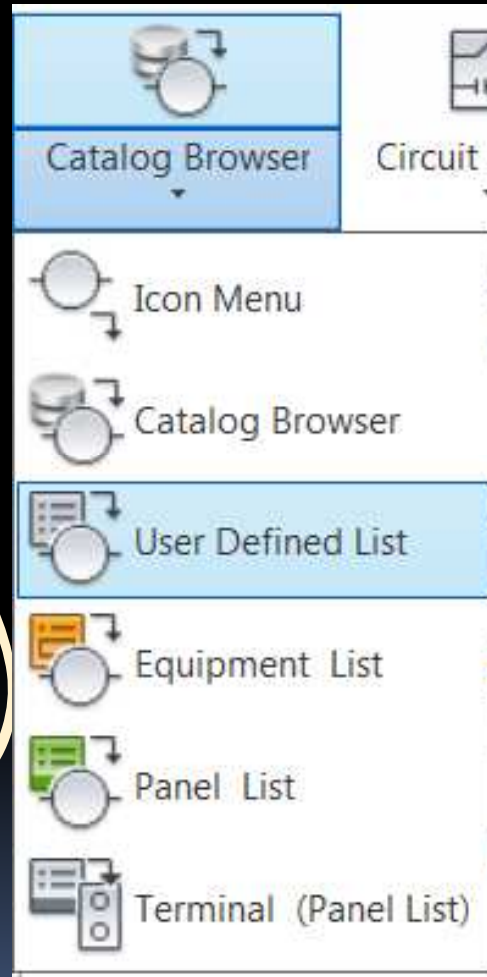
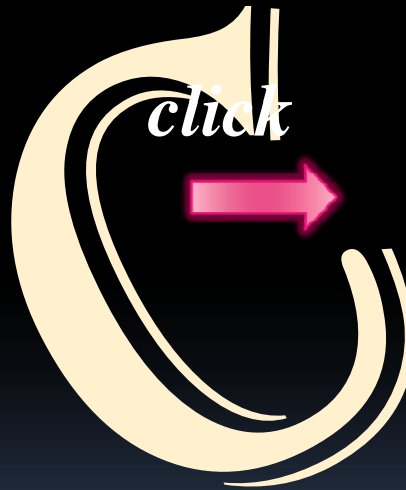


IV. Inserarea componentelor speciale

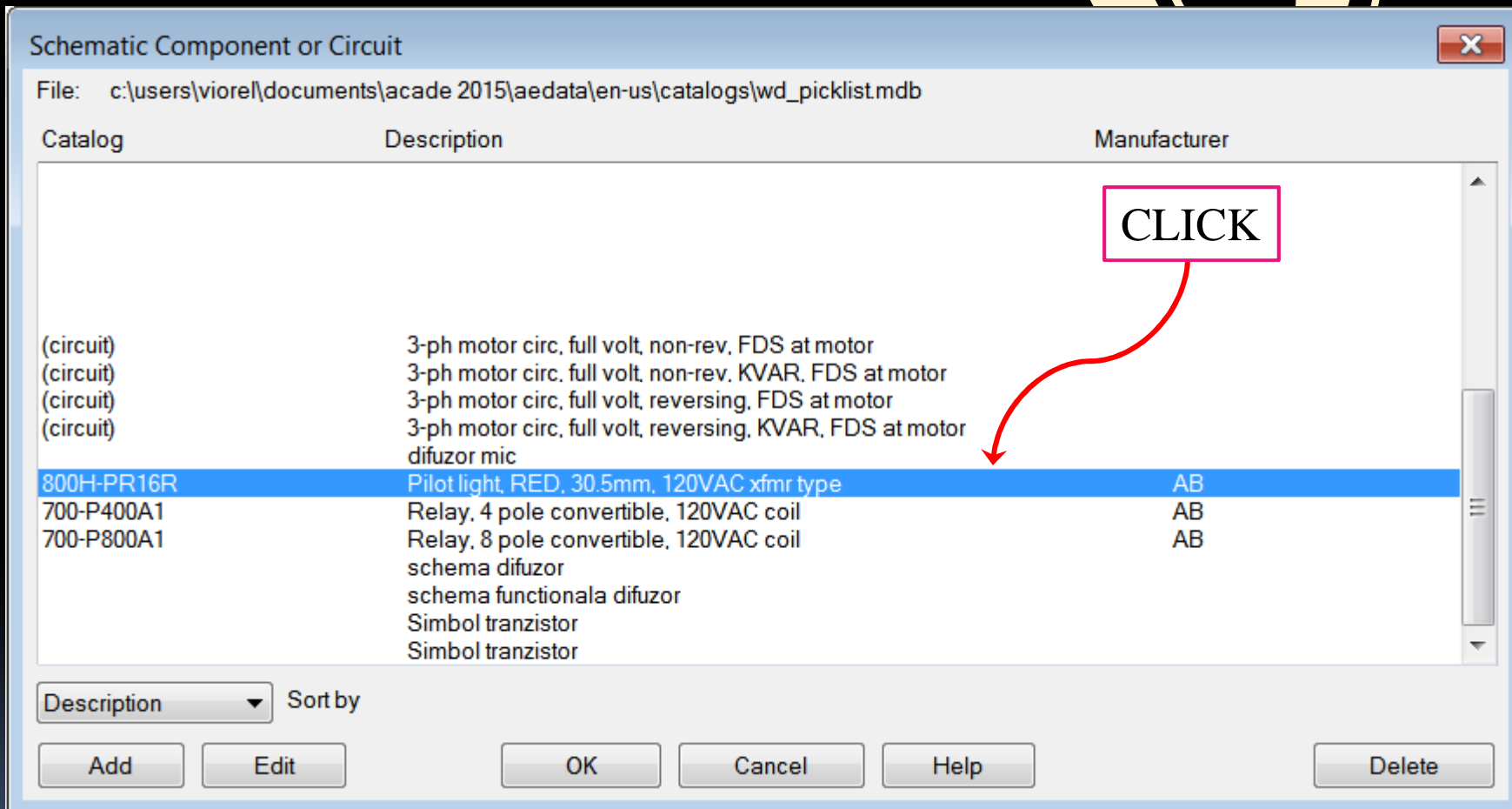
CHAPTER

1. User Defined List

1. Panel Schematic



2. Schematic Component or Circuit



3. Plasare componentă

[-][Top][2D Wireframe]

REVISIONS				
10013	10014	10015	10016	10017

300

R1

40W

1 2

CR1

45

NO NC

PROJDESC			
TITLE1			
TITLE2			
SIZE	FSCM NO	DRAWING NUMBER	REV
10014	10015	10016	10017
SCALE	SHEET		10018

Command:
Command:
Specify insertion point:

4. Caseta Insert/Edit Component



Insert / Edit Component

Component Tag: LT1 fixed

Use PLC Address

Tags Used: Schematic, Panel, External List, Options

Catalog Data: Manufacturer: AB, Catalog: 800H-PR16R, Assembly: , Item: 8, Count:

Description: Line 1: CONTROL, Line 2: POWER ON, Line 3:

List: Drawing, Project, Defaults, Pick

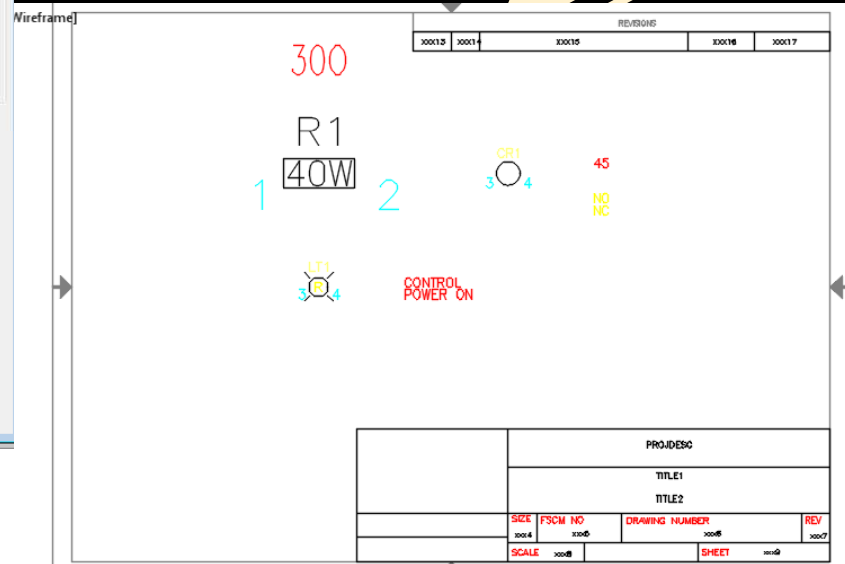
Cross-Reference: Component override, Setup, Reference NO, NO/NC Setup, Reference NC

Installation code: Drawing, Project, Pins: 1: 3, 2: 4, X:

Location code: Drawing, Project, Show/Edit Miscellaneous

Ratings: Rating: , Show All Ratings

Buttons: OK-Repeat, OK, Cancel, Help



Command:
Command:
Type a command

5. Adăugarea componentelor speciale în listă

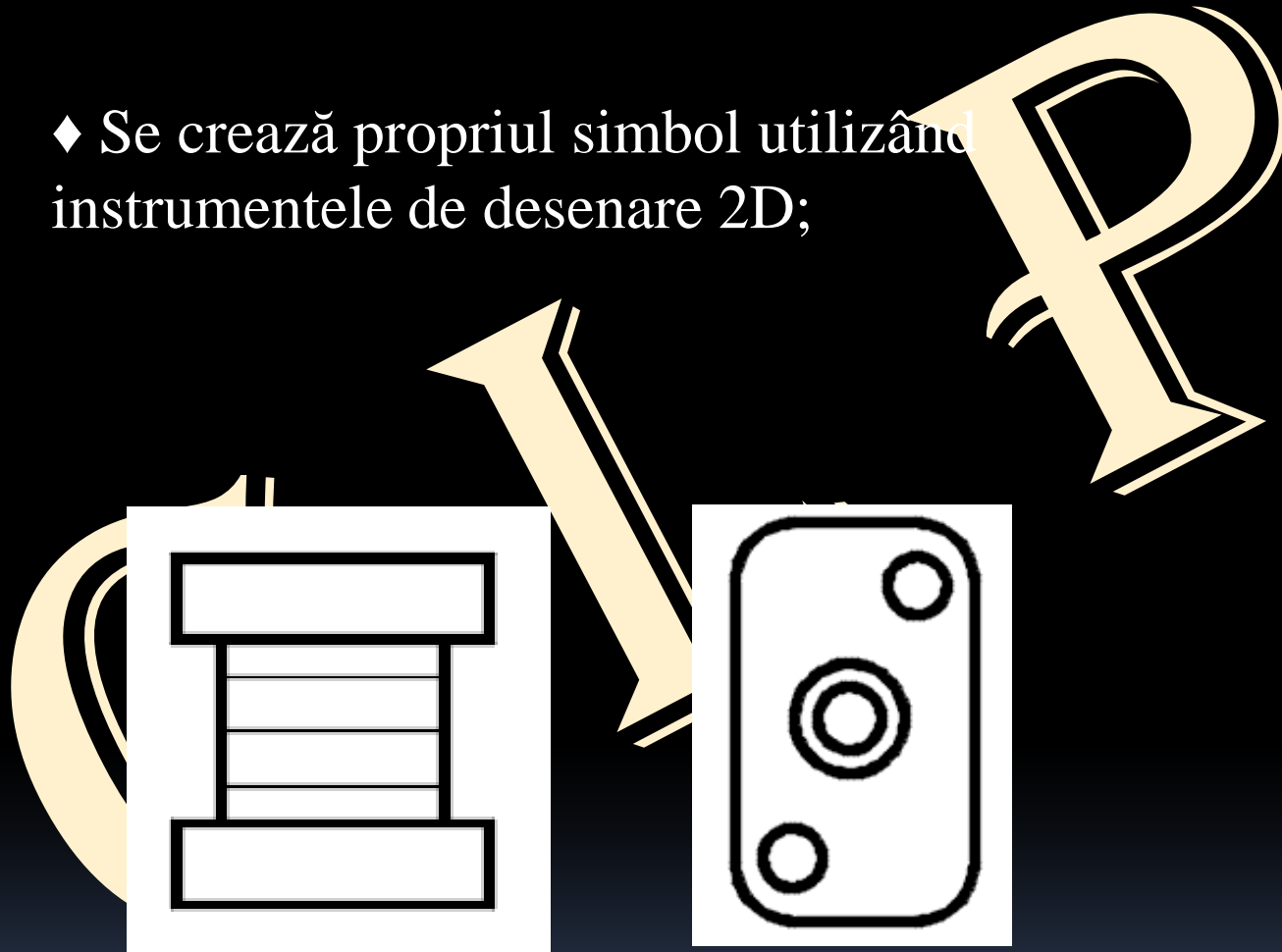


bobine

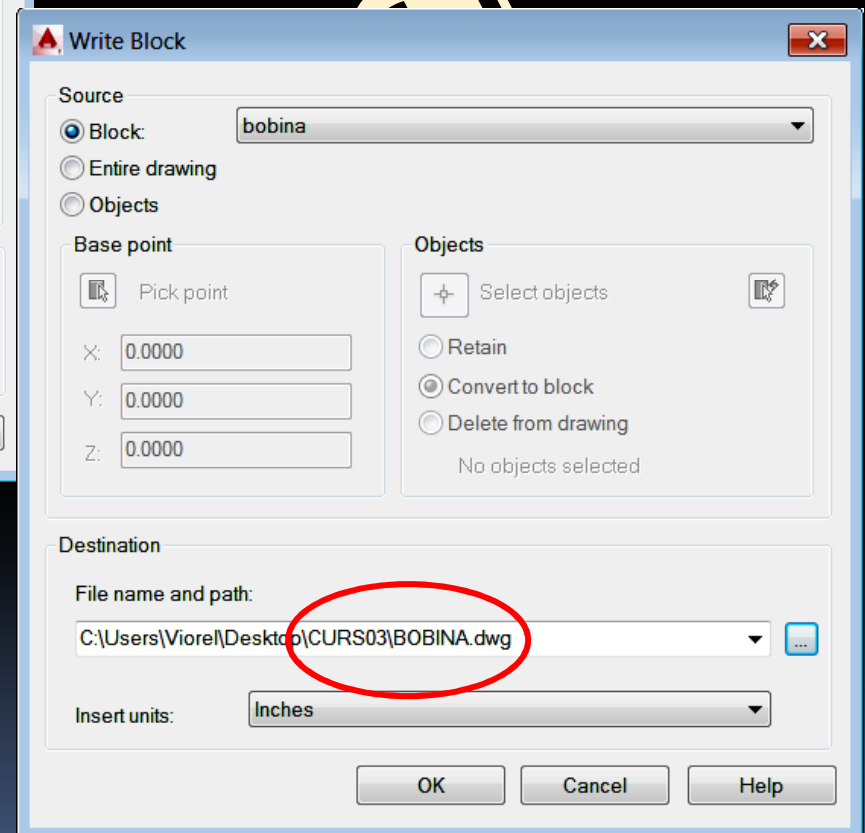
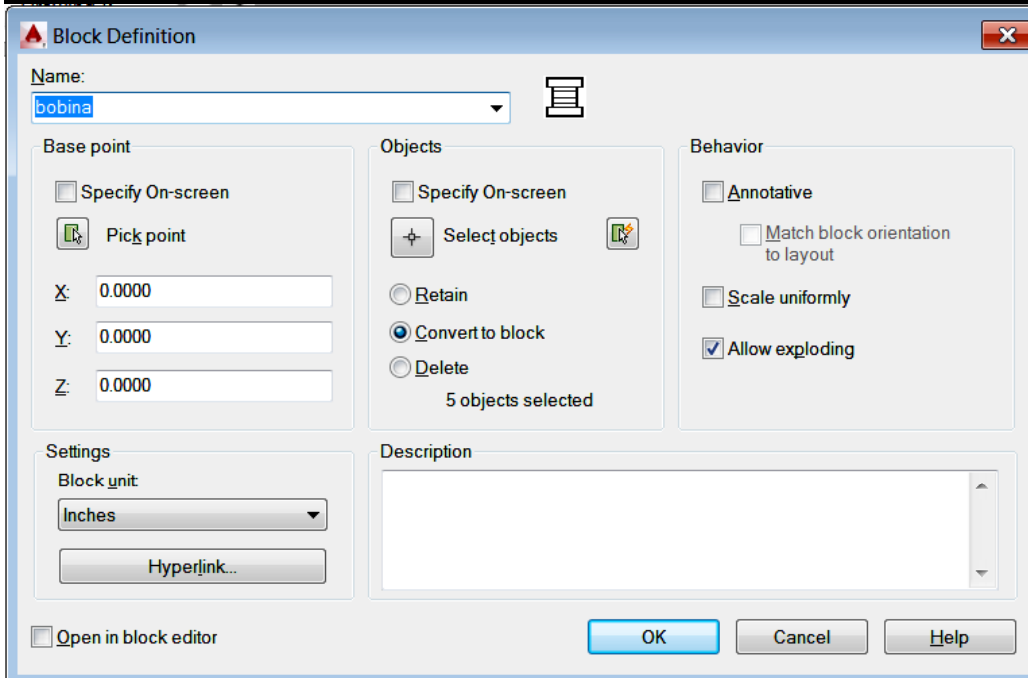


întrerupător

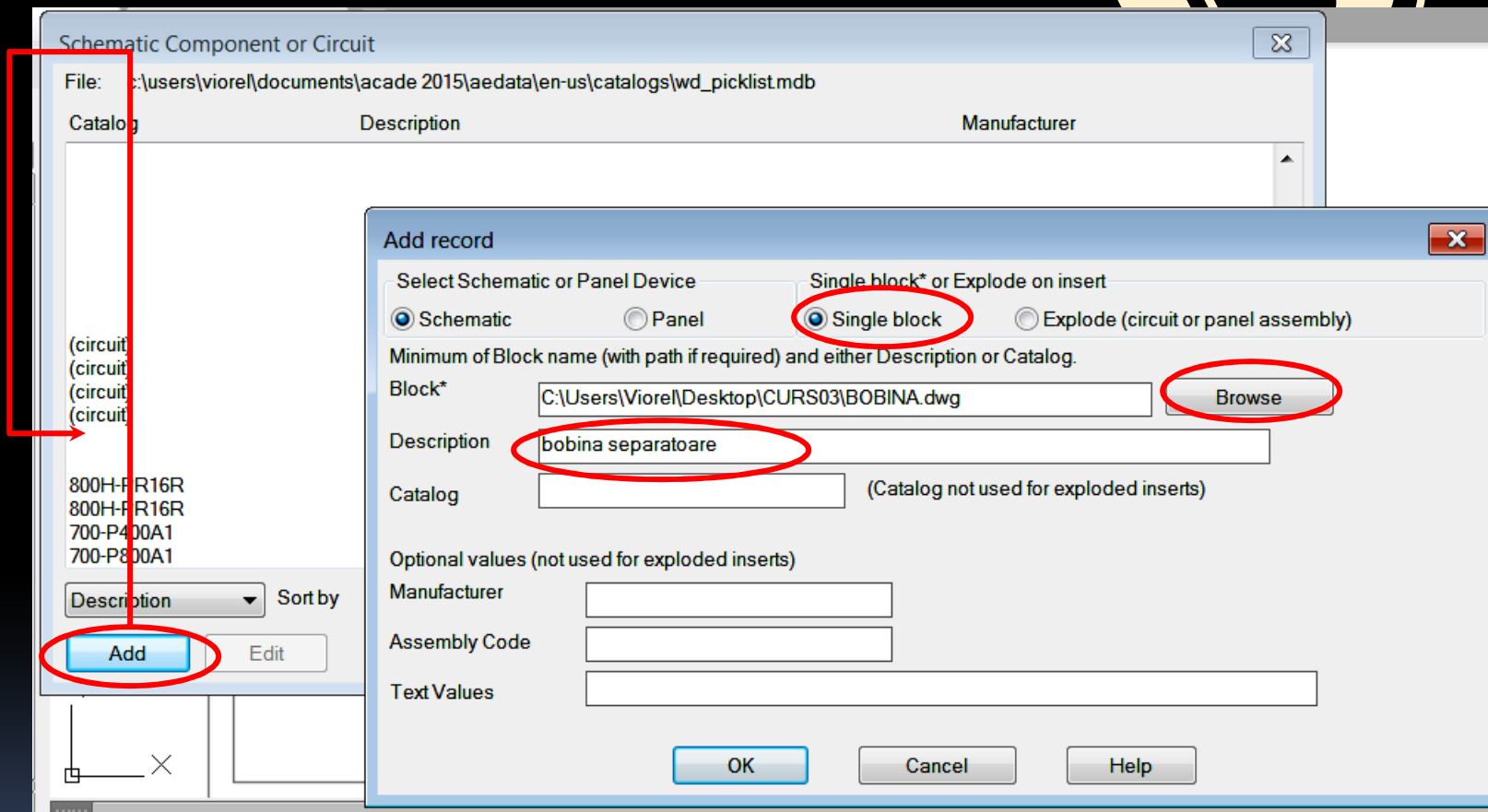
- ◆ Se crează propriul simbol utilizând instrumentele de desenare 2D;



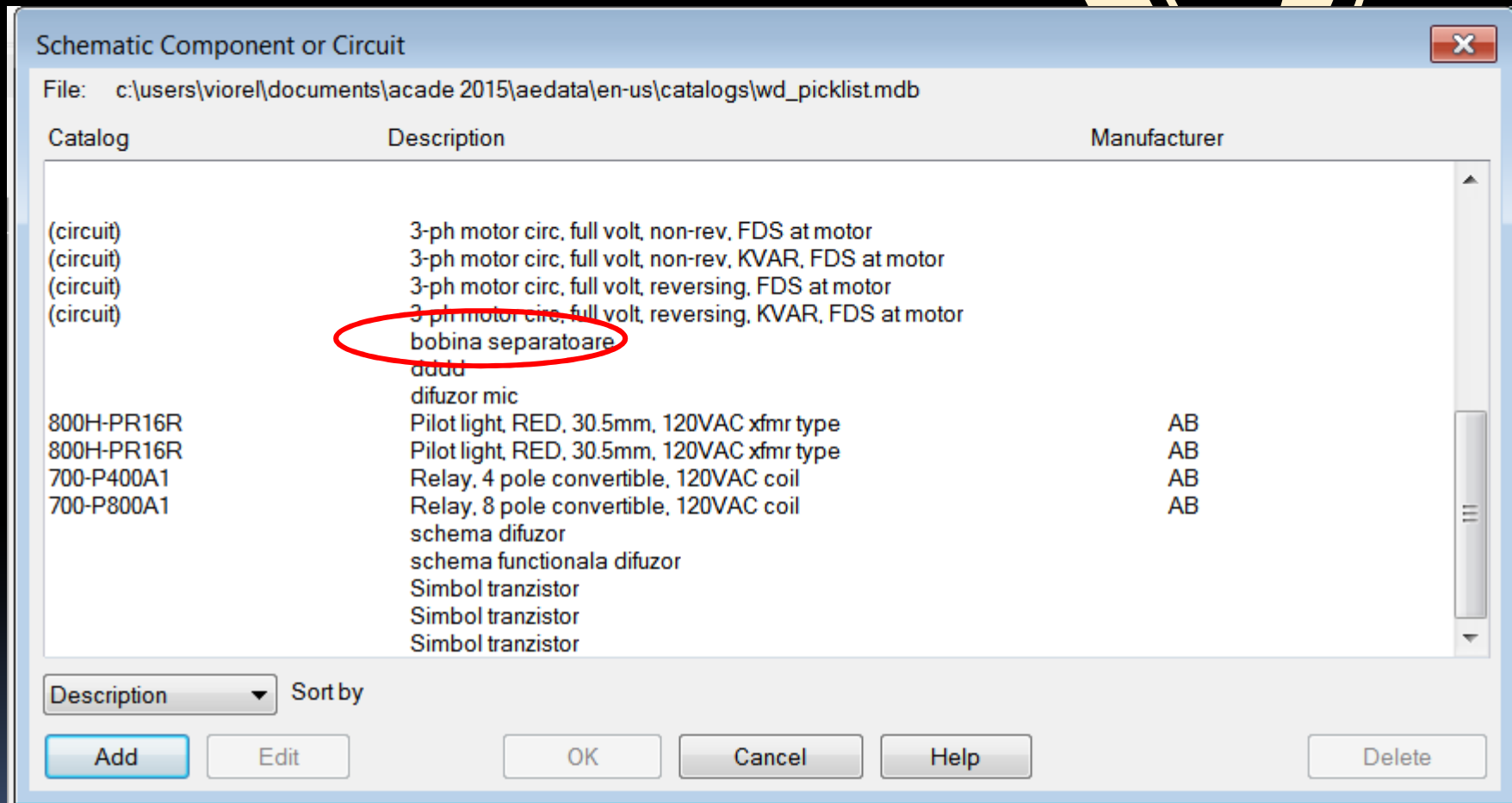
◆ Se crează block → wblock în locația dorită;



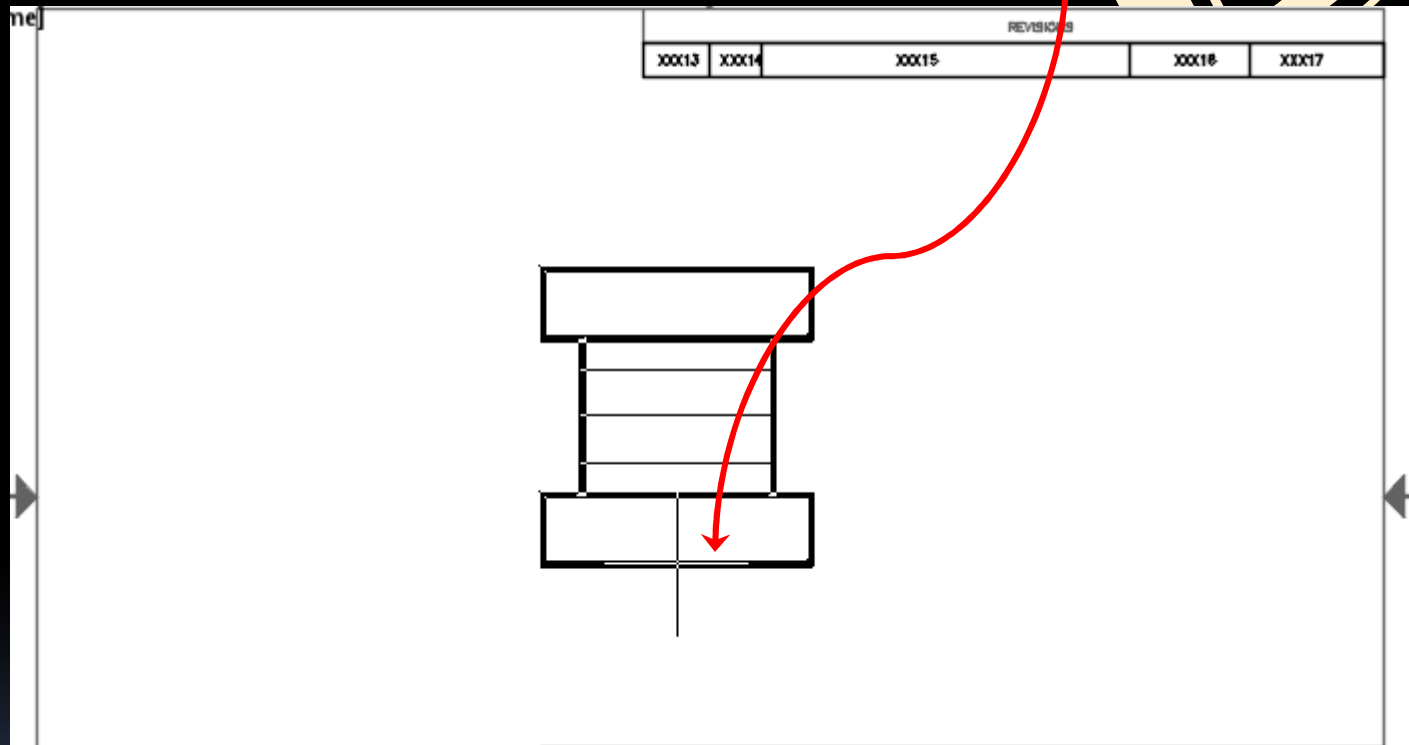
◆ Se adaugă componenta în listă;



◆ Lista este completată cu componenta specială;

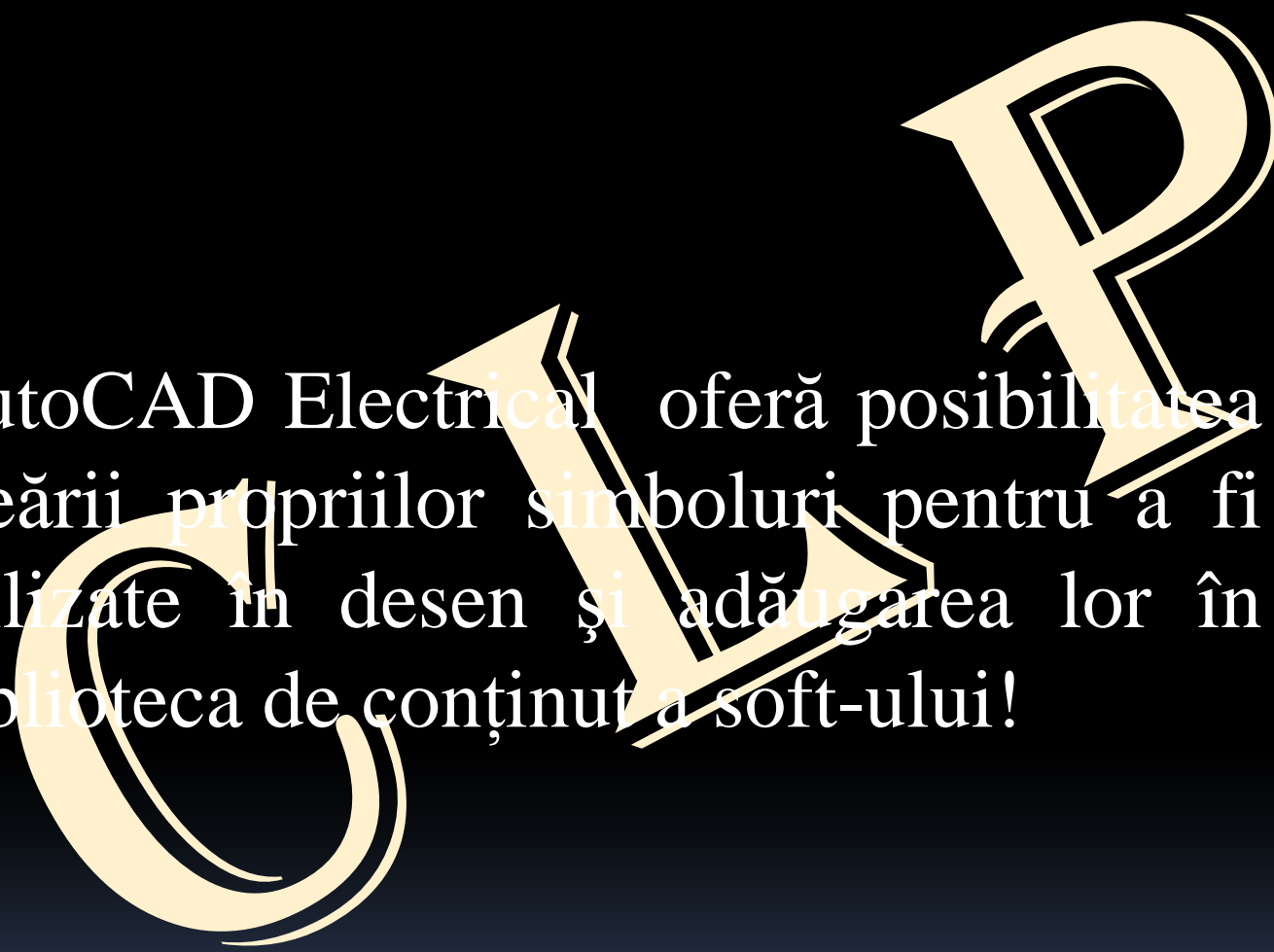


- ◆ Componenta specială se atașează cursorului, la selecție.



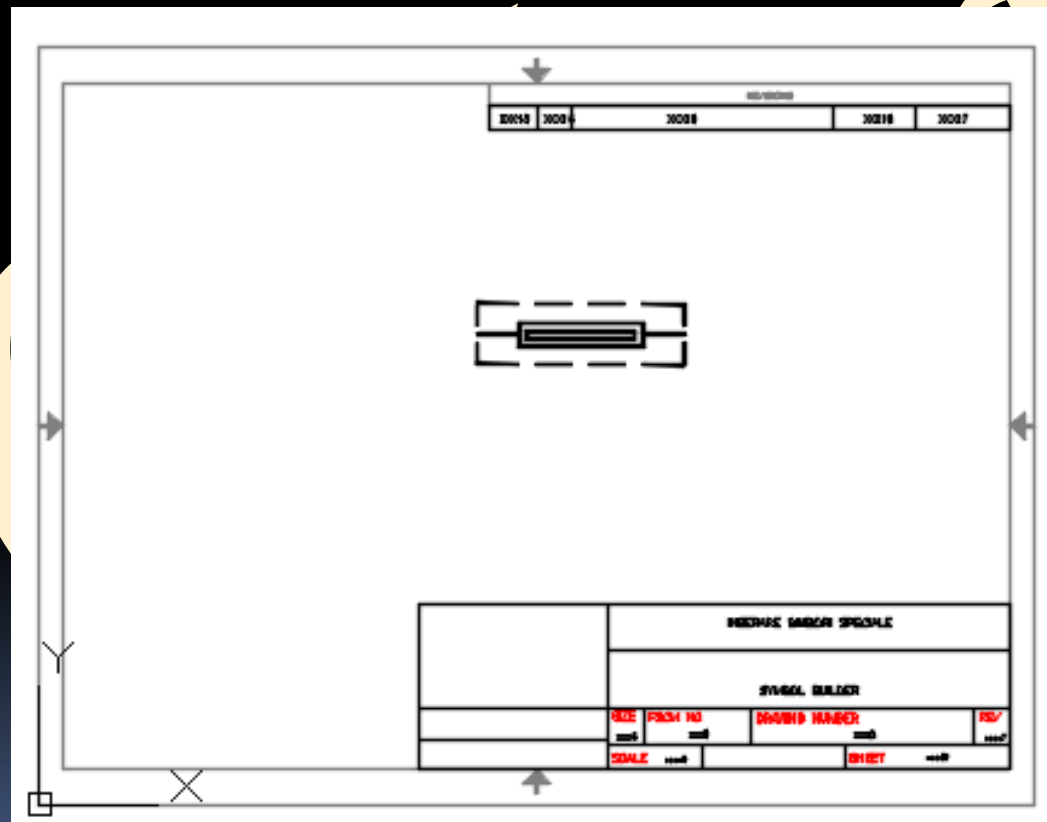


2. Creatorul de simboluri

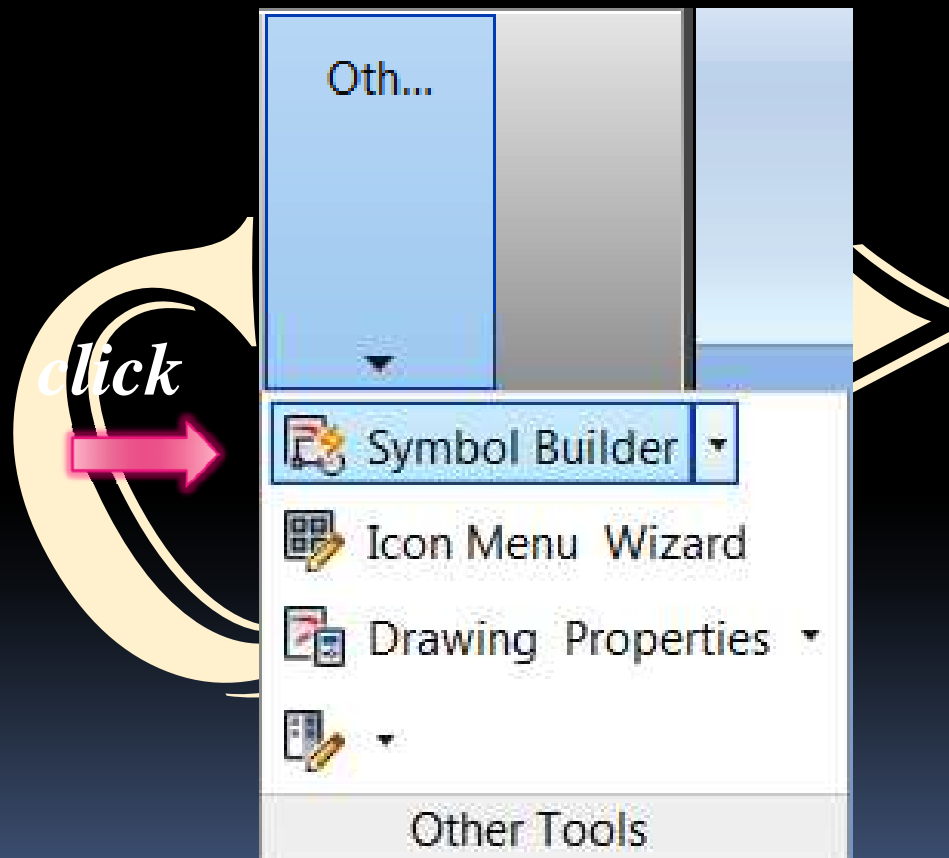
The image features three large, stylized yellow letters: 'C' on the left, 'L' in the center, and 'P' on the right. Each letter has a 3D effect with a dark shadow and a white highlight. The 'L' has a white arrow pointing towards the text. The 'C' has a white arrow pointing towards the text. The 'P' has a white arrow pointing towards the text. The background is a gradient from black at the top to blue at the bottom.

AutoCAD Electrical oferă posibilitatea
creării propriilor simboluri pentru a fi
utilizate în desen și adăugarea lor în
biblioteca de conținut a soft-ului!

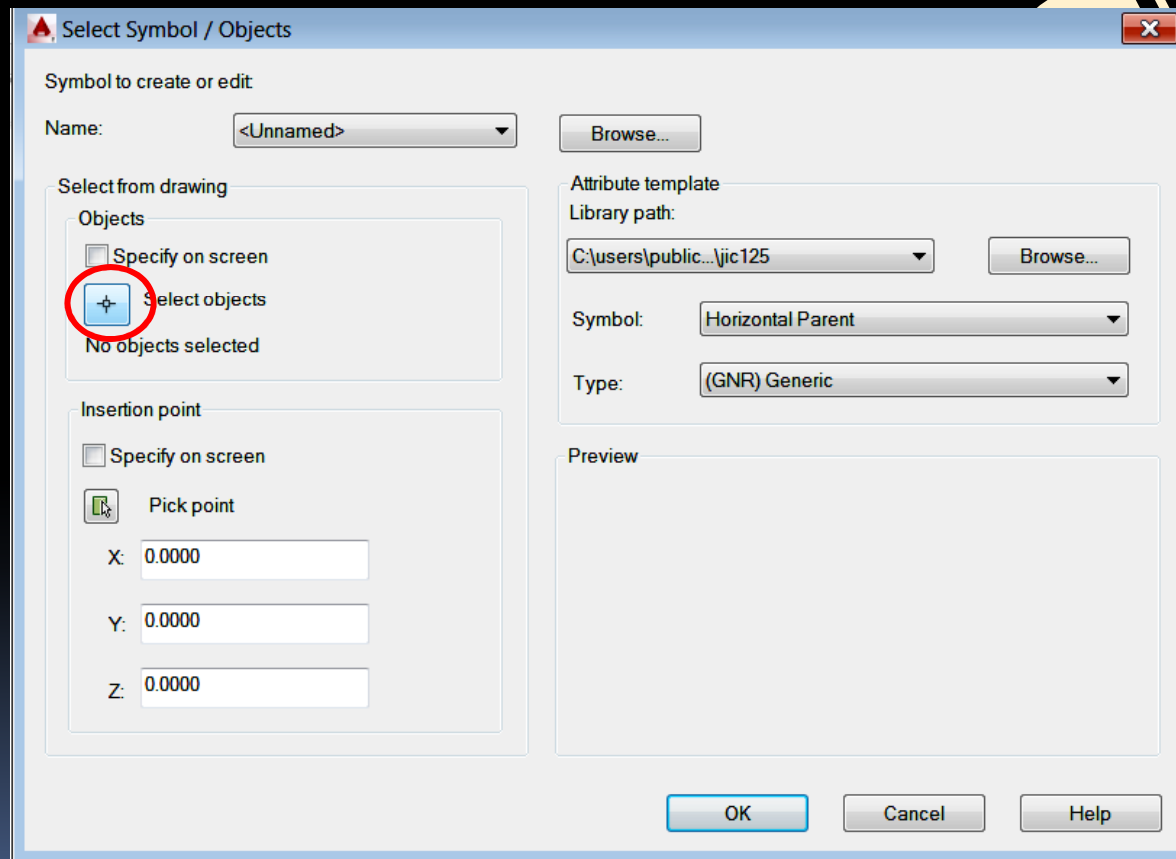
1. Crearea schiței pentru simbolul dorit



*2. Click pe panoul Other Tools,
afișul Symbol Builder*



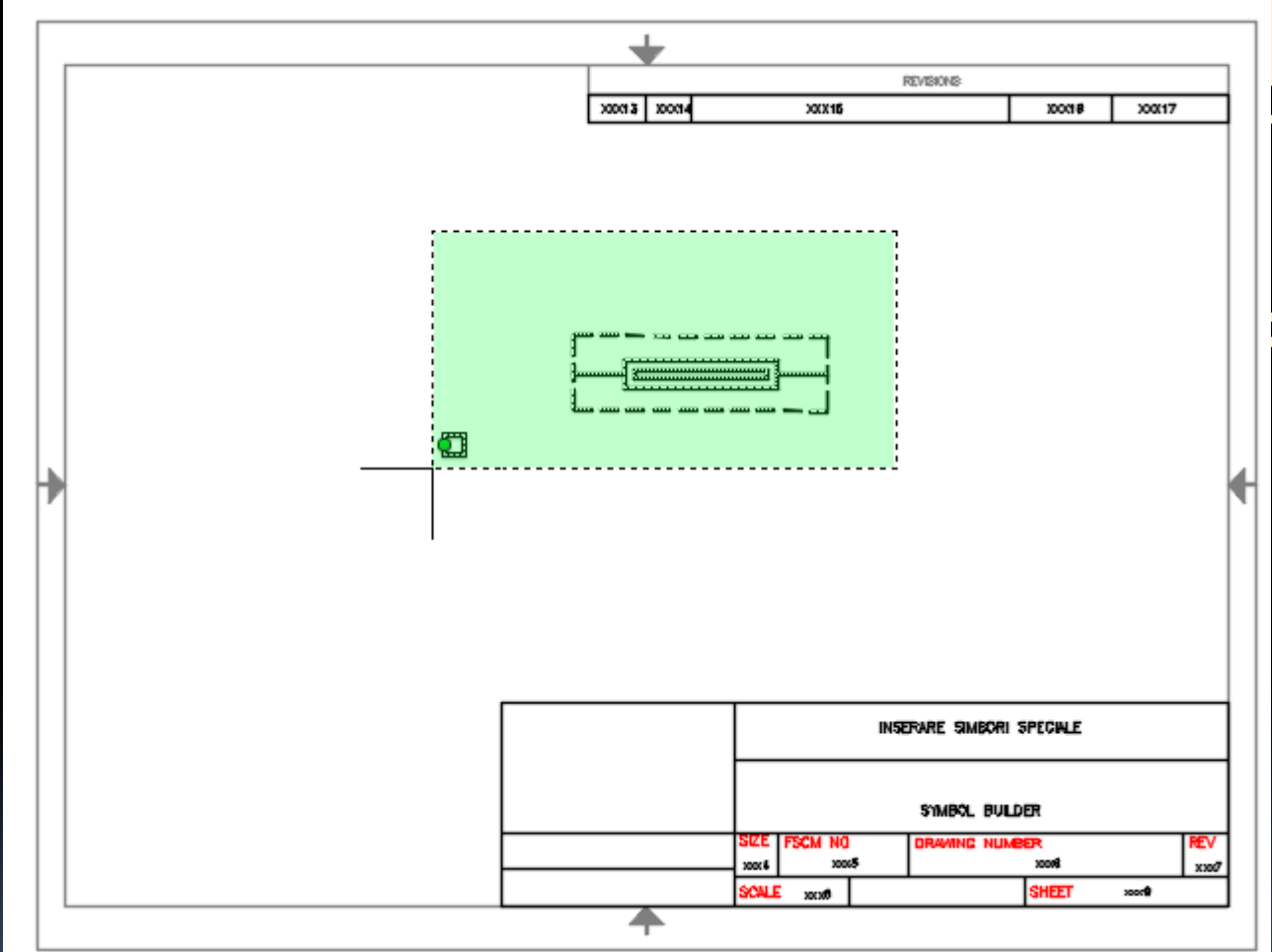
3. Caseta Select Symbol/Objects selecția obiectelor



The image shows a screenshot of the 'Select Symbol / Objects' dialog box in AutoCAD. The dialog is titled 'Select Symbol / Objects' and has a standard Windows window border with a close button (X) in the top right corner. The dialog is divided into several sections:

- Symbol to create or edit:** A 'Name:' dropdown menu is set to '<Unnamed>', and a 'Browse...' button is to its right.
- Select from drawing:** This section contains a sub-section for 'Objects' with a 'Specify on screen' checkbox (unchecked) and a 'Select objects' button (circled in red) with a plus sign icon. Below it, it says 'No objects selected'.
- Insertion point:** This section has a 'Specify on screen' checkbox (unchecked) and a 'Pick point' button with a cursor icon. Below are three input fields for X, Y, and Z coordinates, all set to '0.0000'.
- Attribute template:** A 'Library path:' dropdown menu is set to 'C:\users\public...\jic125', with a 'Browse...' button to its right.
- Symbol:** A dropdown menu is set to 'Horizontal Parent'.
- Type:** A dropdown menu is set to '(GNR) Generic'.
- Preview:** A large empty rectangular area for previewing the symbol.

At the bottom of the dialog, there are three buttons: 'OK', 'Cancel', and 'Help'.



REVISIONS				
XXXX3	XXXX4	XXXX16	XXXX8	XXXX17

				INSERIRE SIMBOLI SPECIALE	
				SIMBOLI BULDER	
	SIZE	FSCM NO	DRAWING NUMBER		REV
	XXXX6	XXXX5	XXXX9		XXXX7
	SCALE	XXXX10	SHEET		XXXX8

4. Caseta Select Symbol/Objects vizionarea obiectelor

Select Symbol / Objects

Symbol to create or edit

Name:

Select from drawing

Objects

Specify on screen

Select objects

5 object(s) selected

Insertion point

Specify on screen

Pick point

X:

Y:

Z:

Attribute template

Library path:

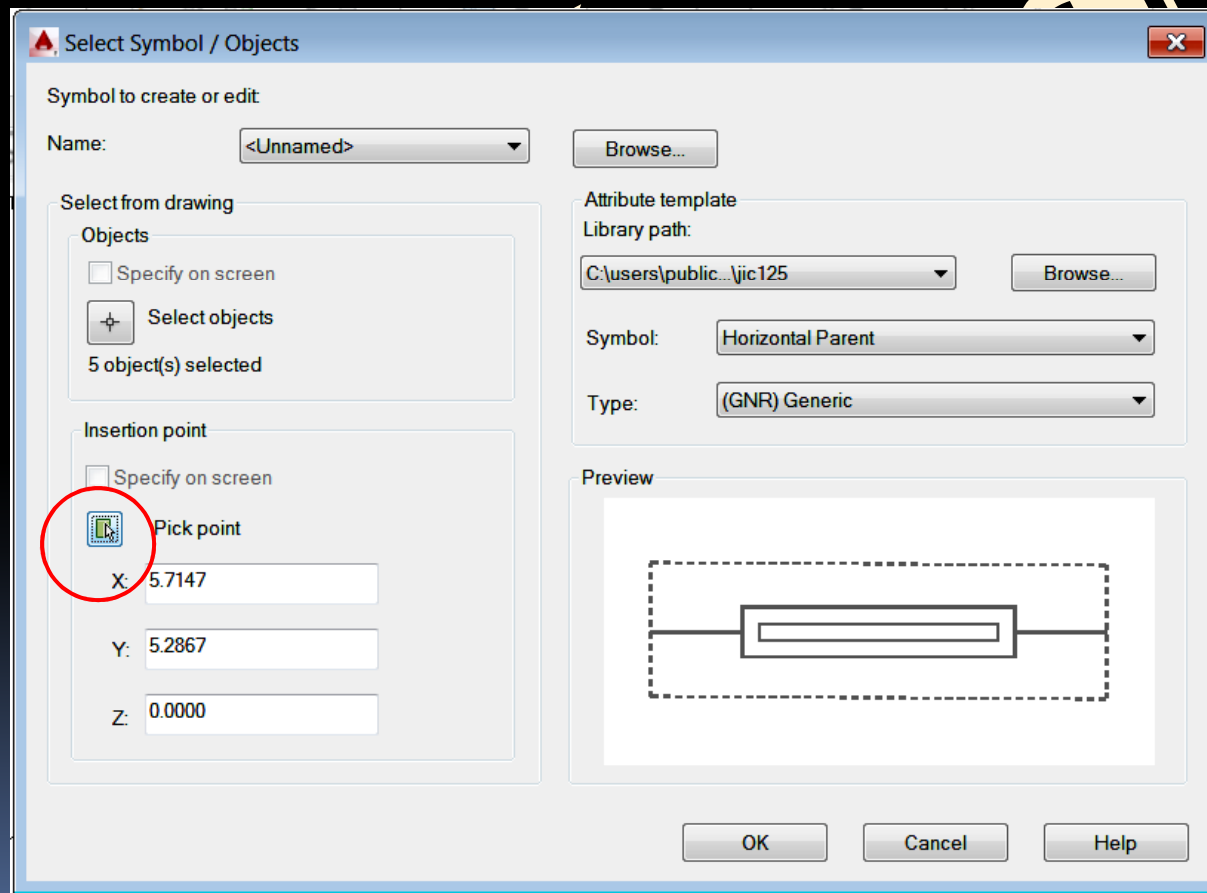
Symbol:

Type:

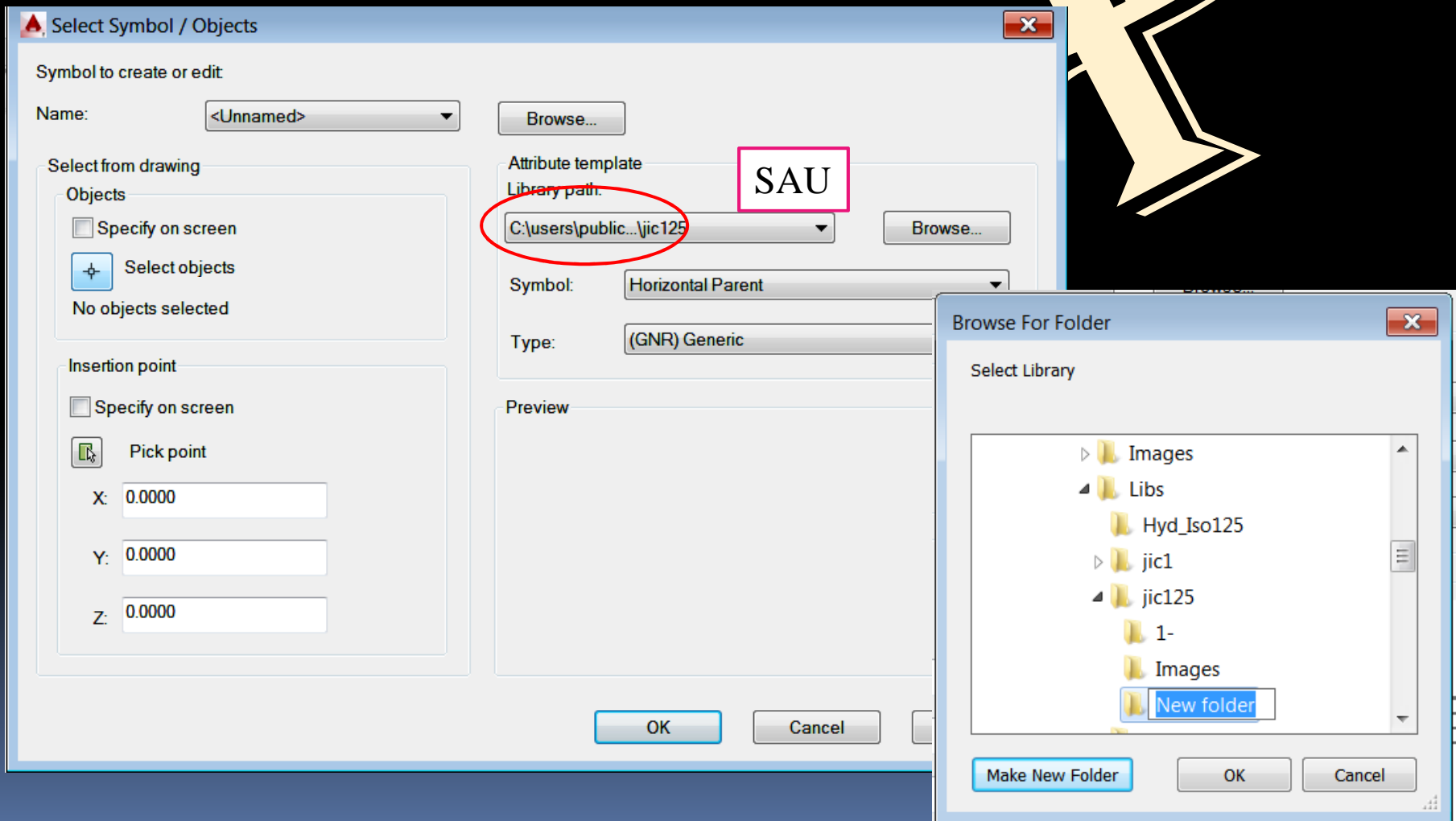
Preview

5. Caseta Select Symbol/Objects

*Pick point pentru specificarea
punctului de inserție pentru simbol*



6. Caseta Select Symbol/Objects alegerea directoarelor



8. Afișarea mediului de creare a simbolurilor

The screenshot displays the AutoCAD Electrical Symbol Builder interface. The left-hand side contains a library panel with the following sections:

- Library:** C:\users\public\...jic125
- Symbol:** Horizontal Parent
- Type:** Generic
- Required:** TAG1, MFG, CAT, ASSYCODE, FAMILY, DESC1, DESC2, DESC3, INST, LOC, XREF
- Optional:** XREFNO, XREFNC, CONTACT (NO), STATE, PINLIST, PEER_PINLIST, WDTAGALT, WDTYPE, WD_WEBLINK, WD_JUMPERS
- POS:** POS1
- RATING:** RATING1
- Wire Connection:** Direction / Style (Left / None)
- Pins:**
- Link Lines:** Direction (Top)
- Inserted Link Lines:**

The main drawing area shows a schematic diagram with a central component symbol (a rectangle with a smaller rectangle inside) connected to a larger dashed rectangular frame. The interface includes a ribbon with tabs like 'Block Editor' and 'Symbol Builder', and a command line at the bottom displaying the message: "Enter block name or [?]: HDV1_001 Regenerating model."

9. Symbol Builder Attribute Editor

Library: C:\users\public\...\jic125
Symbol: Horizontal Parent
Type: Generic

Required

TAG1	
MFG	
CAT	
ASSYCODE	
FAMILY	
DESC1	
DESC2	
DESC3	
INST	
LOC	
XREF	
<input checked="" type="checkbox"/> V1	

Optional

XREFNO	
XREFNC	
CONTACT	NO
STATE	
PINLIST	
PEER_PINLIST	
WDTAGALT	
WDTYPE	
WD_WEBLINK	
WD_JUMPERS	

POS

POS1	
------	--

RATING

RATING1	
---------	--

Wire Connection

Direction / Style | Left / None

Pins

Link Lines

Direction	Top
-----------	-----

Inserted Link Lines

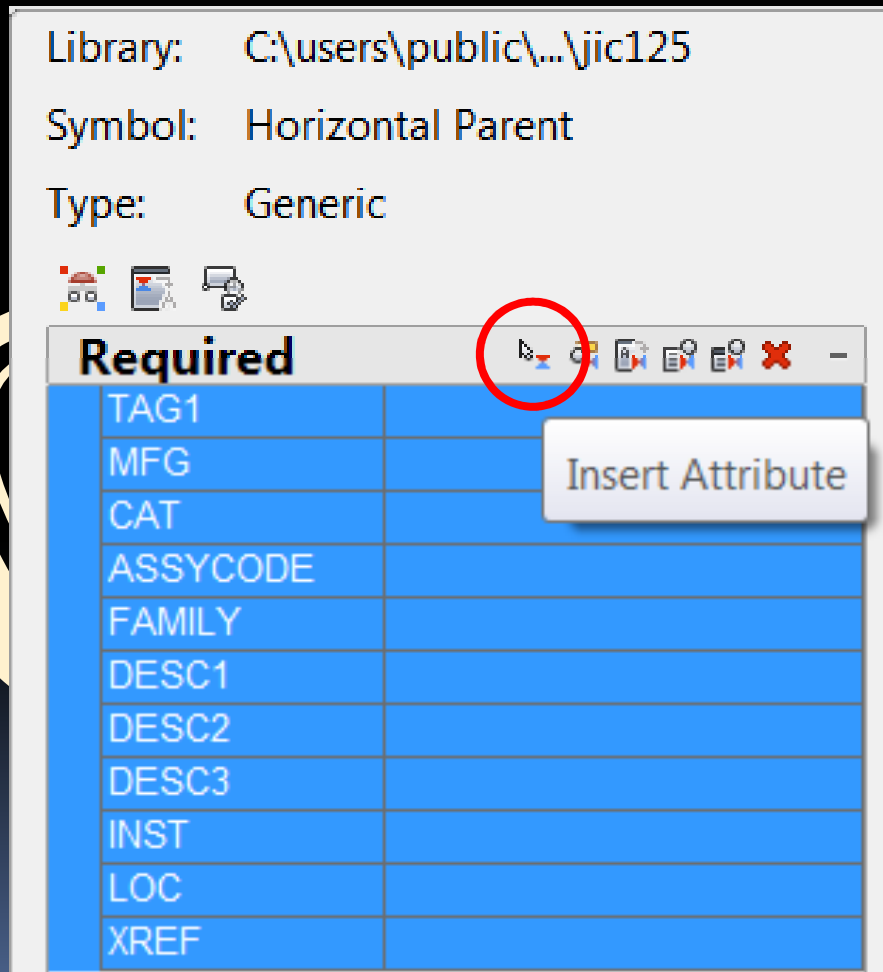
SYMBOL BUILDER ATTRIBUTE EDITOR



10. Completarea Atributelor

◆ Selectarea etichetelor

Library: C:\users\public\...\jic125
Symbol: Horizontal Parent
Type: Generic



Required

TAG1	
MFG	
CAT	
ASSYCODE	
FAMILY	
DESC1	
DESC2	
DESC3	
INST	
LOC	
XREF	

Insert Attribute

◆ Modificarea etichetelor

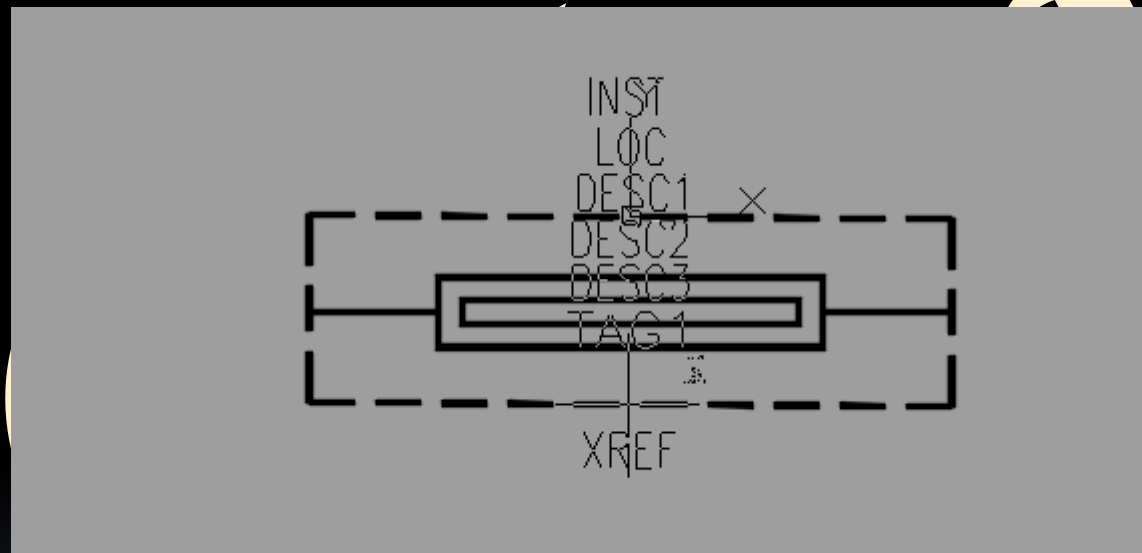
Text	
Tag	V1
Prompt	
Value	
Style	Standard
Justify	Left
Height	0.0500
Rotation	0
Width factor	0.0000
Obliquing	0
Direction	By style
Boundary...	0.0000

Misc	
Upside down	No
Backward	No
Invisible	No
Constant	No
Verify	No
Preset	No
Multiple lines	No
Lock posi...	No

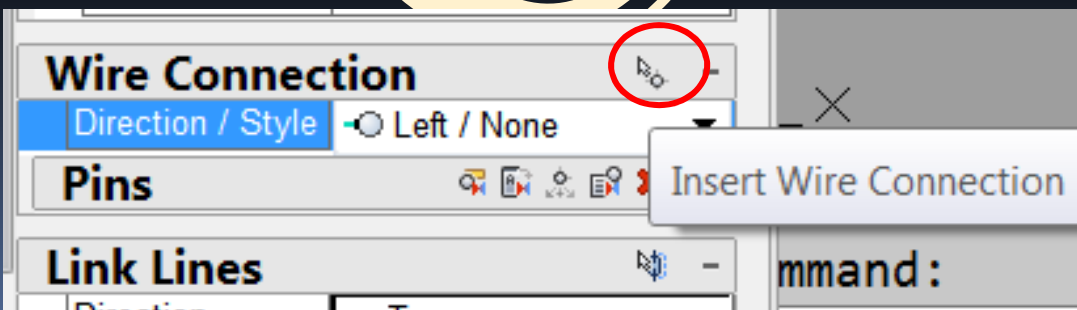
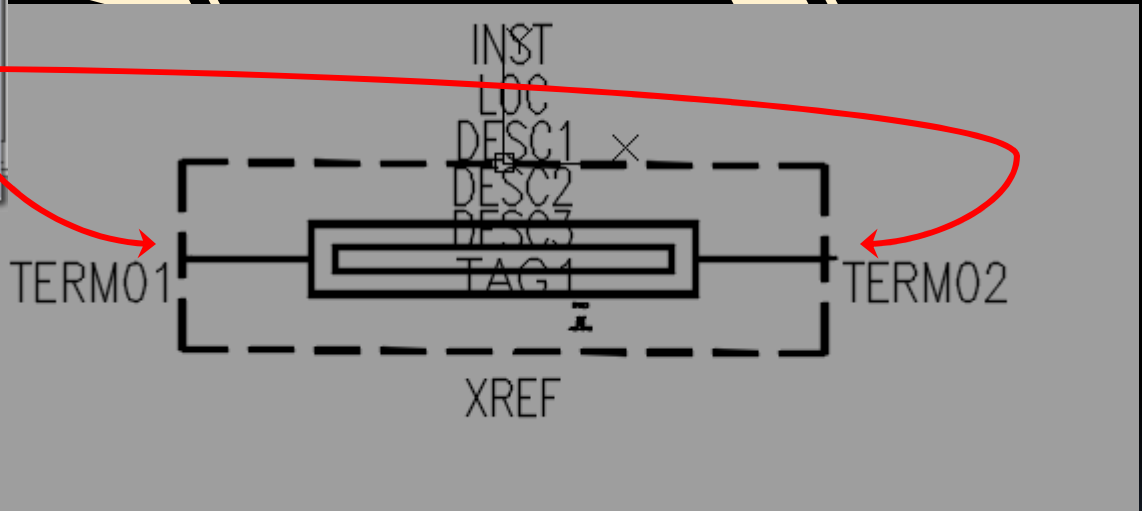
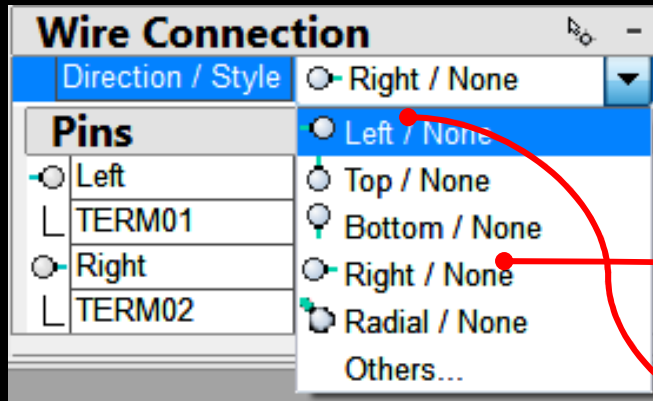
Insert / Edit is valid only for this session

Insert OK Cancel Help

◆ Aplicarea etichetelor pe simbol

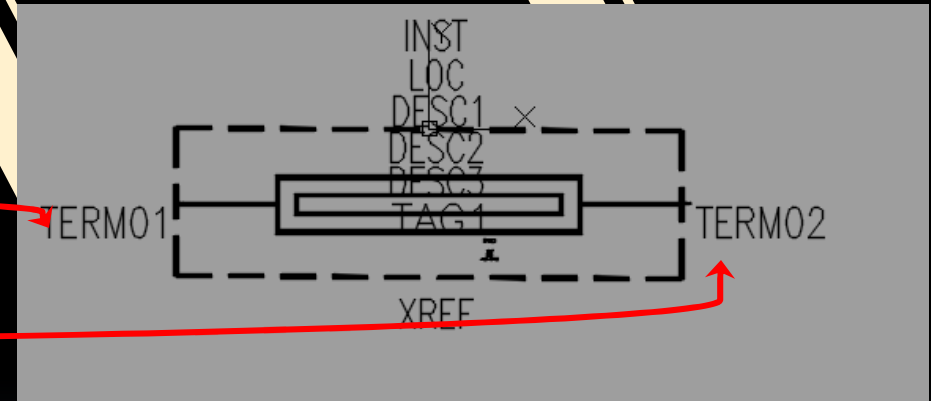


◆ Conectarea firelor – definirea terminalelor componentei



- ◆ Conectarea firelor - numerotarea pinilor componente

Pins	
Left	
TERM01	1
Right	
TERM02	2



- ◆ Finalizarea editărilor → click DONE din tabul **Symbol Builder**

11. Close Block Editor: Save Symbol

Close Block Editor: Save Symbol

Symbol

Block Wblock

Orientation: (H) Horizontal

Catalog lookup

Symbol name: (DV) Generic

WDBLKNAM: HAM

Type: (1) Parent

Contact: <Not Applicable>

Unique identifier: _015

Symbol name: **HDTUB|_015**

File path: C:\users\public\documents\autodesk\acade ;

Base point

Specify on screen

Pick point

X: 0.0000

Y: 0.0000

Z: 0.0000

Image

Icon image

Name (.png): HDV1_015

File path: C:\Users\Viorel\AppData\Roaming\Autodesk\Autc

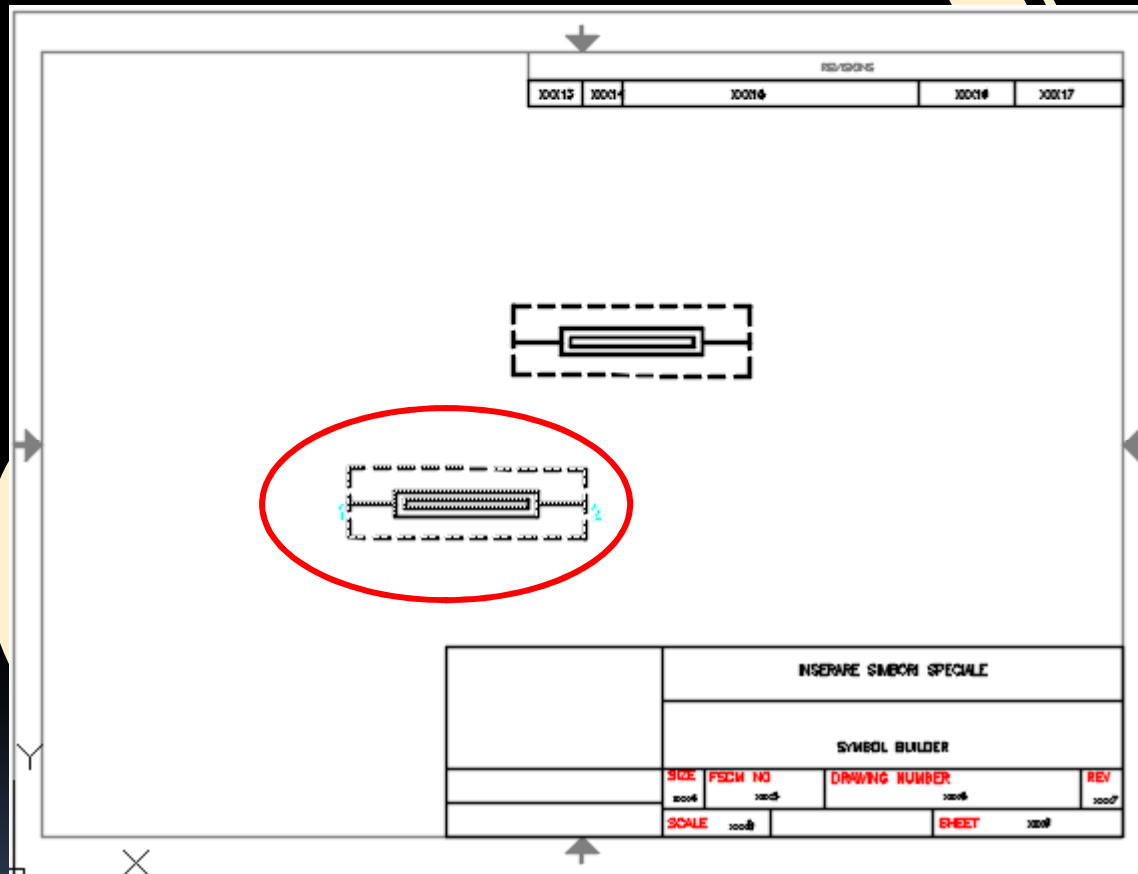
0 error(s) found in the symbol

Details...

OK No Cancel Help

Punctul de inserție se alege în centrul componentei

◆ Simbolul este inserat în desen ca block!



12. Completarea Casetei Insert/Edit Component



Insert / Edit Component

Component Tag: TUB fixed

Use PLC Address

Tags Used: Schematic Panel External List Options

Catalog Data: Manufacturer Catalog Assembly Item Count

Next>> Lookup Previous Drawing Project Multiple Catalog Catalog Check

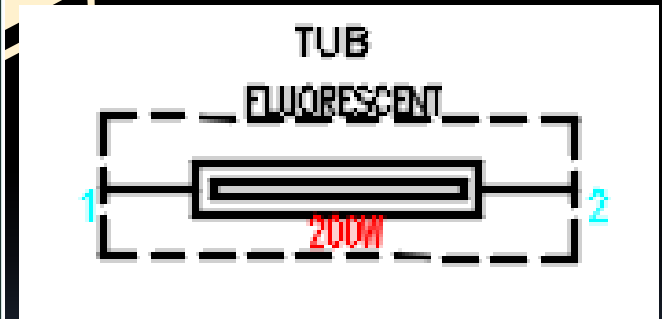
Ratings: Rating Show All Ratings

Description: Line 1: FLUORESCENT Line 2: 200W Line 3: List: Drawing Project Defaults Pick

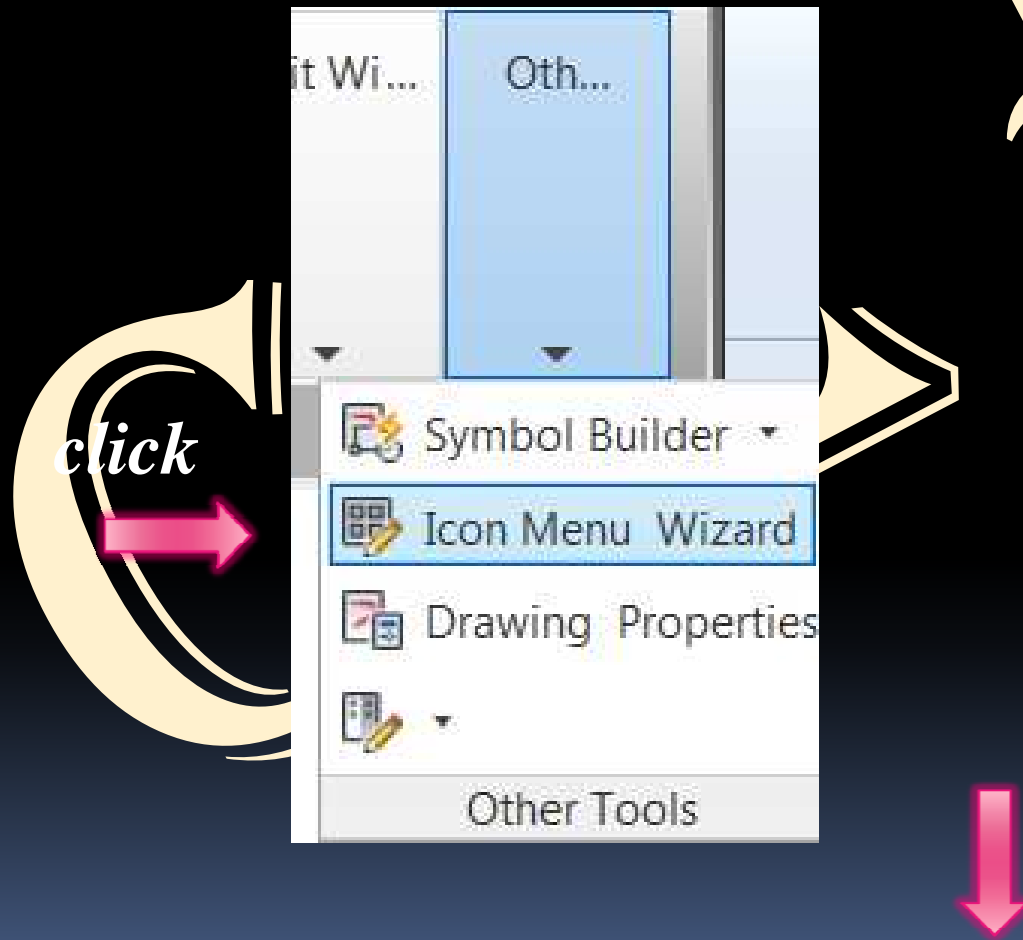
Cross-Reference: Component override Setup Reference NO Reference NC NO/NC Setup

Installation code: Drawing Project Pins: 1 1 2 2 X

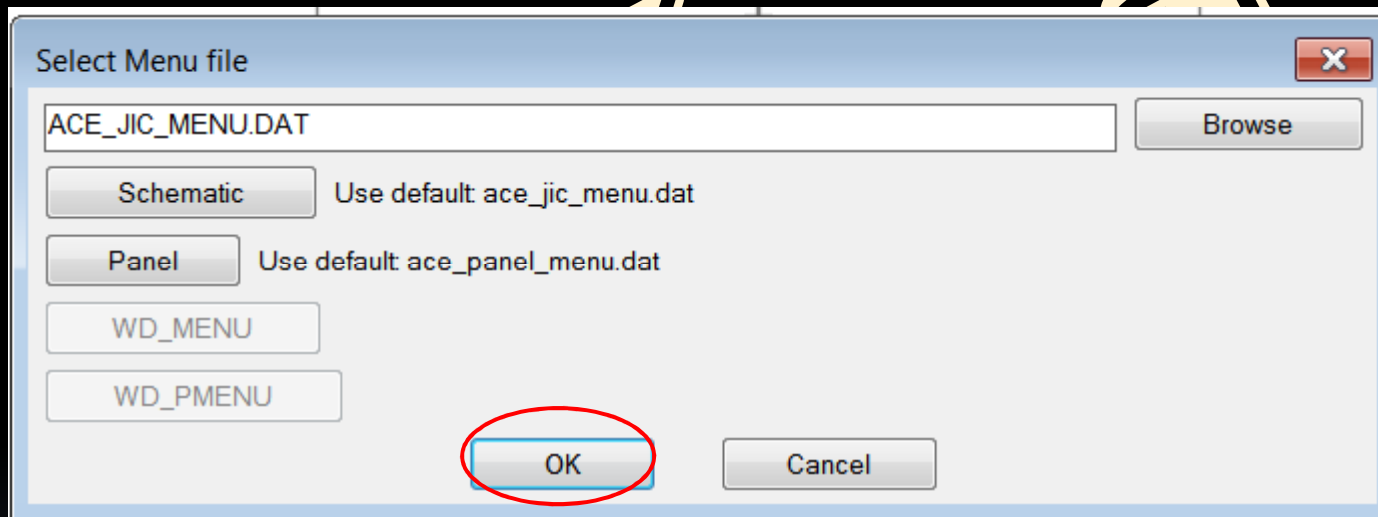
Location code: Drawing Project Show/Edit Miscellaneous OK-Repeat OK Cancel Help



13. Adăugarea unui nou simbol în Icon Menu



◆ Select Menu file



◆ Icon Menu Wizard

Click pe butonul **Add**, apoi pe butonul **Component**



◆ Add Icon-Component

Add Icon - Component

Icon Details

Preview

Name: TUB

Image file: HDTUB1_015

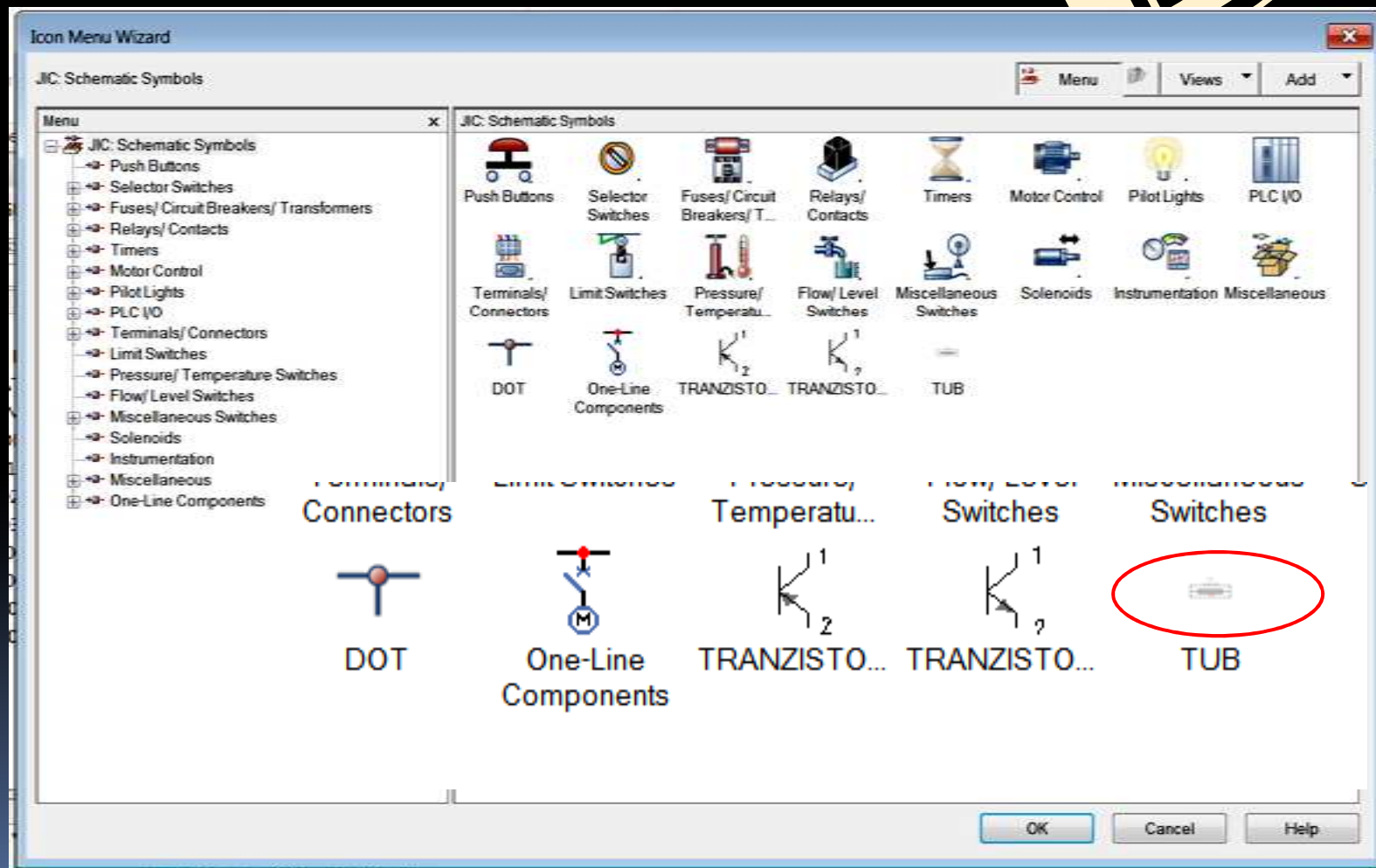
Create PNG from current screen image

Location: C:\Users\Viorel\AppData\Roaming\Autodesk\AutoCAD Elect...\HDTUB1_015.PNG

Block Name to Insert

Block name: HDTUB1_015

14. Pictograma adăugată în Icon Menu



The image features a black background with a blue gradient at the bottom. In the center, the letters 'C', 'V', and 'P' are rendered in a stylized, yellow, outlined font. A semi-transparent blue rectangular box is overlaid on the letters, containing the text '3. Pneumatic components' in a black, italicized serif font.

3. Pneumatic components

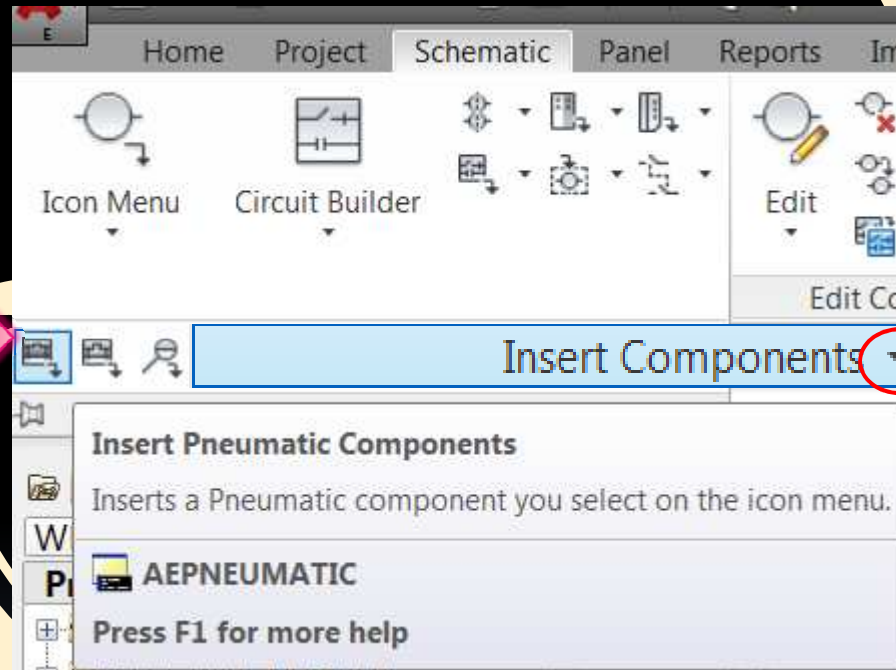
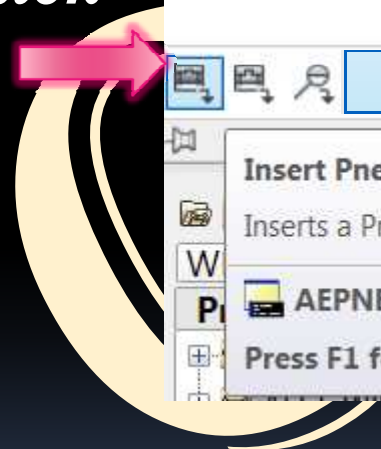


Sunt componente care funcționează
datorită presiunii gazelor care curg prin ele.

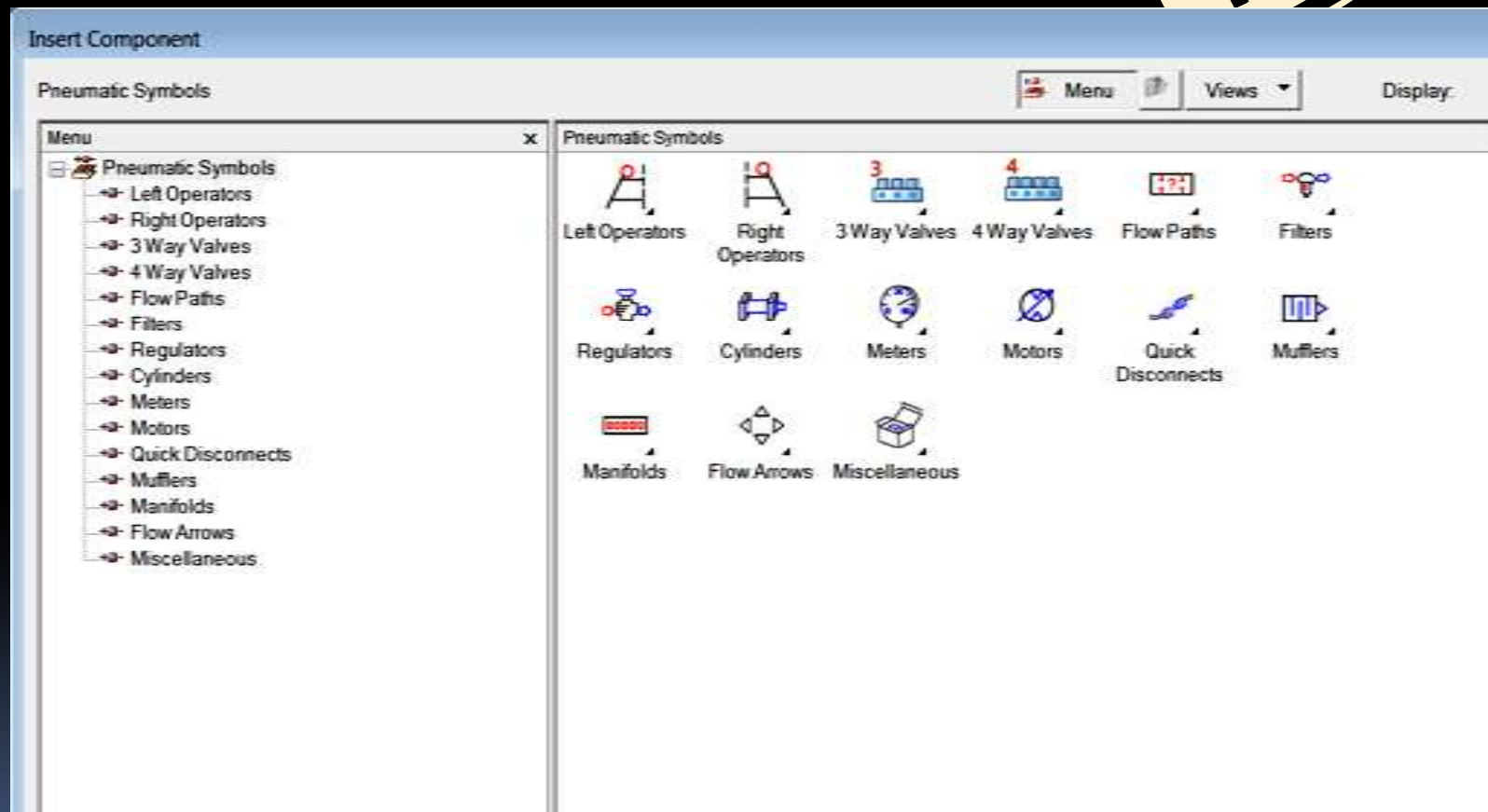
1. Panel Schematic



click



2. Insert Components

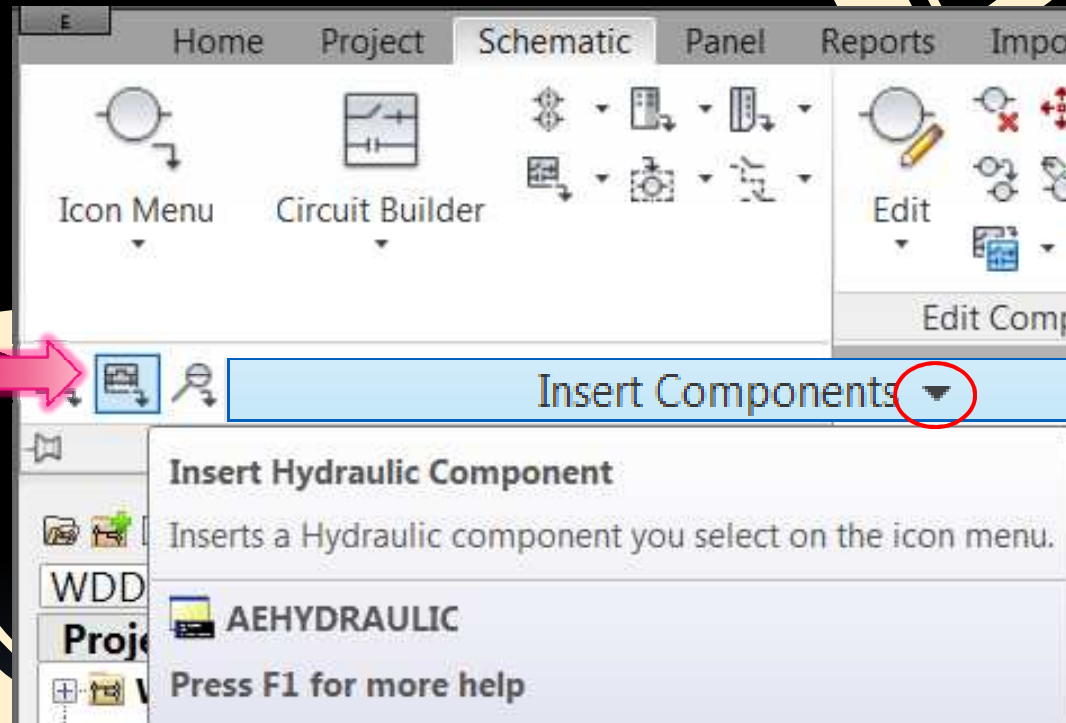


The background features large, stylized yellow letters 'C', 'L', and 'P' arranged diagonally. The 'C' is on the left, 'L' is in the center, and 'P' is on the right. A blue gradient bar is positioned horizontally across the middle of the letters.

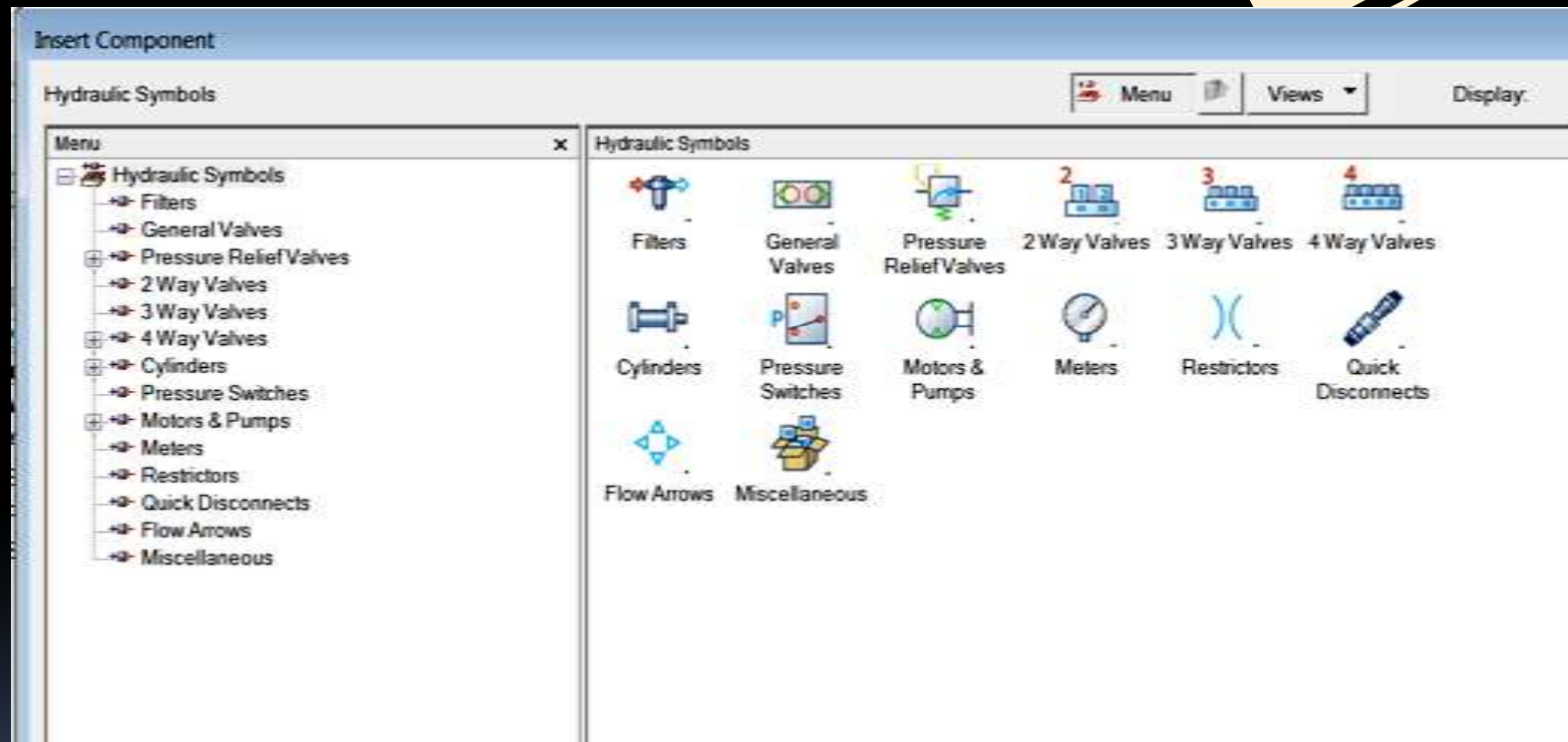
4. Hydraulic components

1. Panel Schematic

click



2. Insert Components





V. Inserarea firelor

The image features three large, stylized yellow letters: 'C', 'L', and 'P'. The 'C' is on the left, the 'L' is in the center, and the 'P' is on the right. They are rendered in a bold, serif font with a slight 3D effect, appearing to float or be layered over the text. The background is a dark blue gradient that transitions from black at the top to a lighter blue at the bottom.

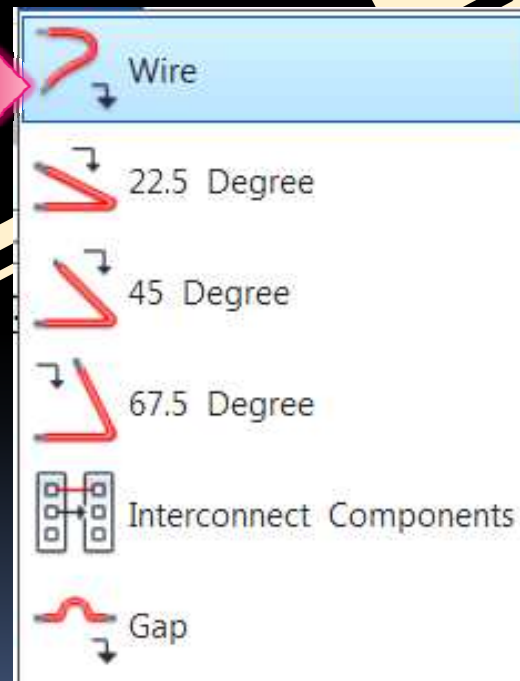
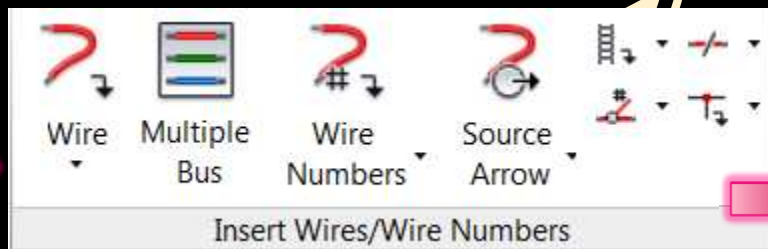
Componentele electrice sunt conectate
între ele cu ajutorul firelor, pentru a
îndeplini diferite sarcini.

C L P

1. Wire

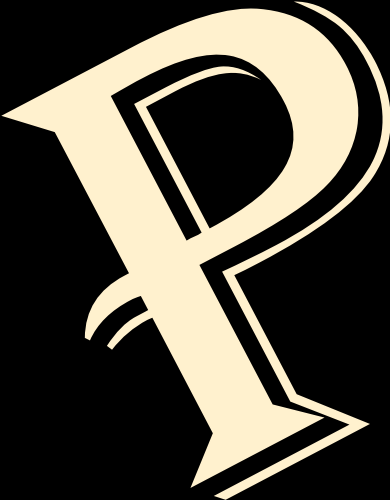
Afişul Wire, panel Insert Wire/Wire Numbers, Ribbon Schematic

click



1. Trasarea firelor

CLICK



```
Insert Wire  
Current wiretype: "BLK_10AWG"  
Specify wire start or [wireType X=show connections]:
```



Caseta de dialog Set Wire Type

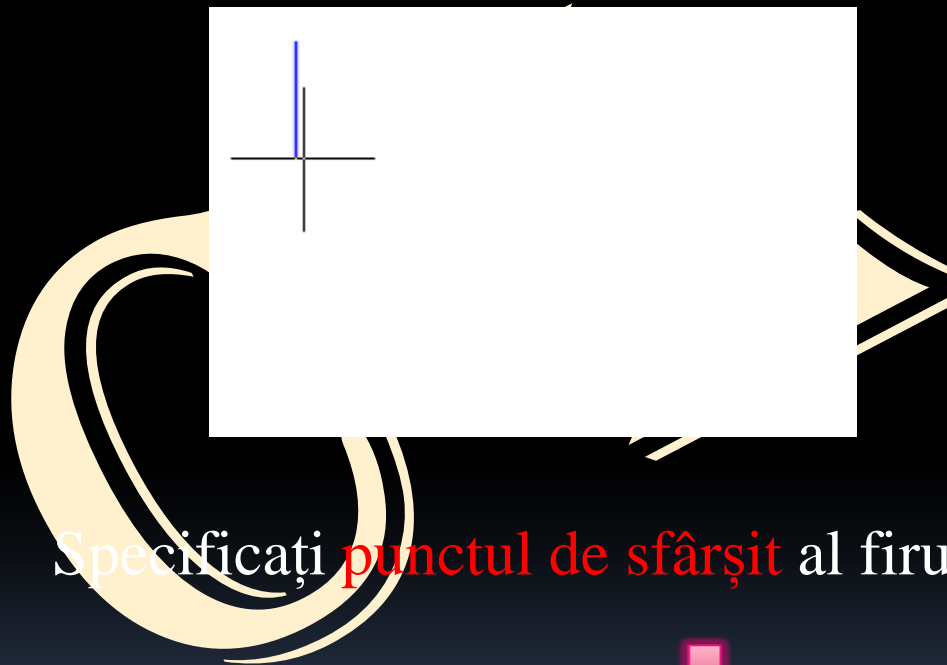


Set Wire Type

	Used	Wire Color	Size	Layer Name	Wire Numbering	USER1	USER2	USER3	USER4
3	X	BLK	14AWG	BLK_14AWG	Yes				
4		BLK	16AWG	BLK_16AWG	Yes				
5		BLK	18AWG	BLK_18AWG	Yes				
6		BLK	20AWG	BLK_20AWG	Yes				
7		BLK	4AWG	BLK_4AWG	Yes				
8		BLK	5AWG	BLK_5AWG	Yes				
9		BLK	6AWG	BLK_6AWG	Yes				
10		BLK	7AWG	BLK_7AWG	Yes				
11		BLK	8AWG	BLK_8AWG	Yes				
12		BLK	9AWG	BLK_9AWG	Yes				
13		BLU	10AWG	BLU_10AWG	Yes				
14		BLU	12AWG	BLU_12AWG	Yes				
15		BLU	14AWG	BLU_14AWG	Yes				
16		BLU	16AWG	BLU_16AWG	Yes				
17		BLU	18AWG	BLU_18AWG	Yes				
18		BLU	20AWG	BLU_20AWG	Yes				

OK Cancel Help

Click în zona de desen pentru a specifica
punctul de pornire al firului



Specificați **punctul de sfârșit** al firului



OPȚIUNI

- ◆ trasarea firelor pe direcție verticală, respectiv orizontală

```
Specify wire end or [V=start Vertical/H=start Horizontal/TAB=Collision off/Continue]:  
Specify wire end or [V=start Vertical/H=start Horizontal/TAB=Collision off/Continue]:  
Specify wire end or [V=start Vertical H=start Horizontal TAB=Collision off Continue]:
```



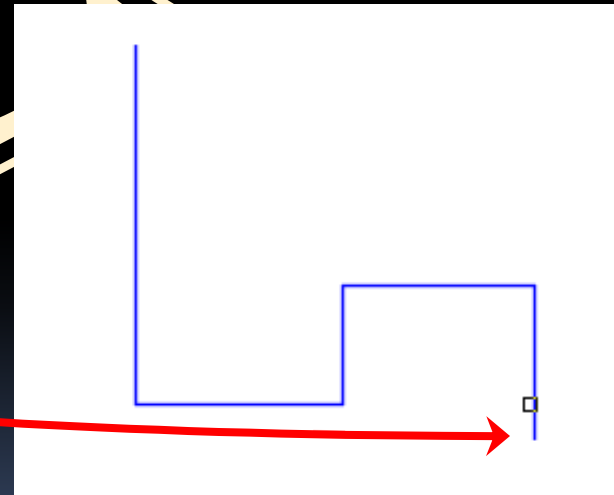
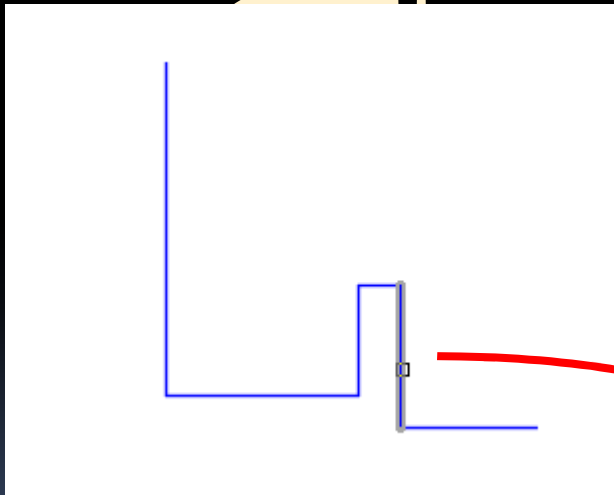
verticală



orizontală

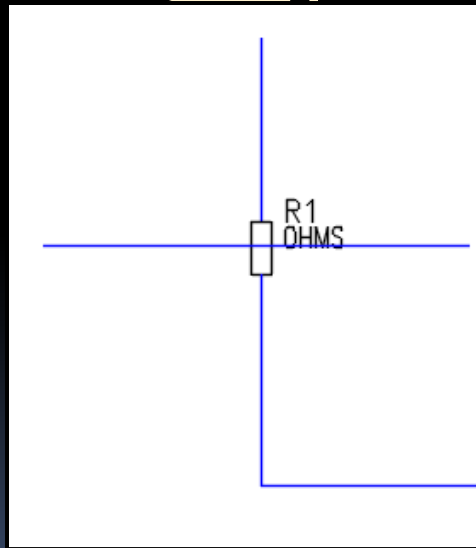
◆ evidențierea deplasării firelor

```
Specify wire end or [V=start Vertical/H=start Horizontal/TAB=  
Specify wire end or [V=start Vertical/H=start Horizontal/TAB=  
Specify wire start or [Scoot wireType X=show connections]:
```

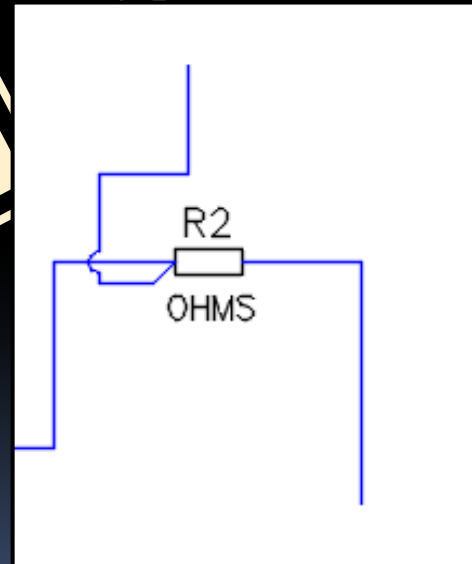


◆ evidențierea conexiunilor

Specify wire start or [wireType/X=show connections]:
Specify wire end or [V=start Vertical/H=start Horizontal/TAB=Collision off/Continue]:v
Specify wire end or [V=start Vertical H=start Horizontal TAB=Collision off Continue]

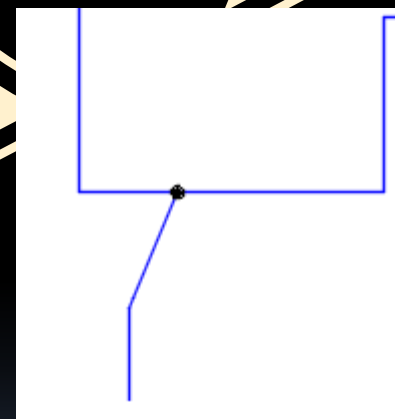
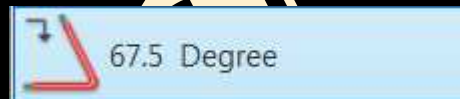
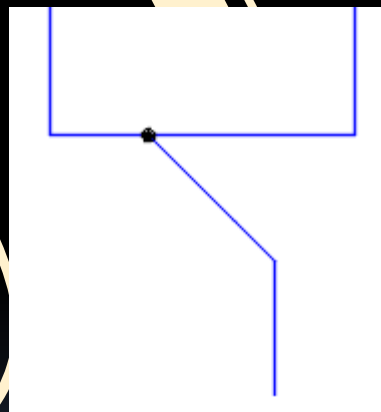
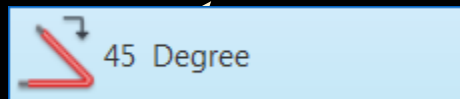
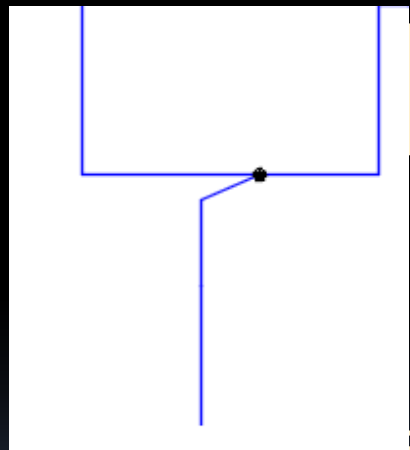
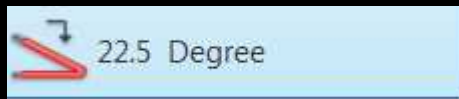


Collision off

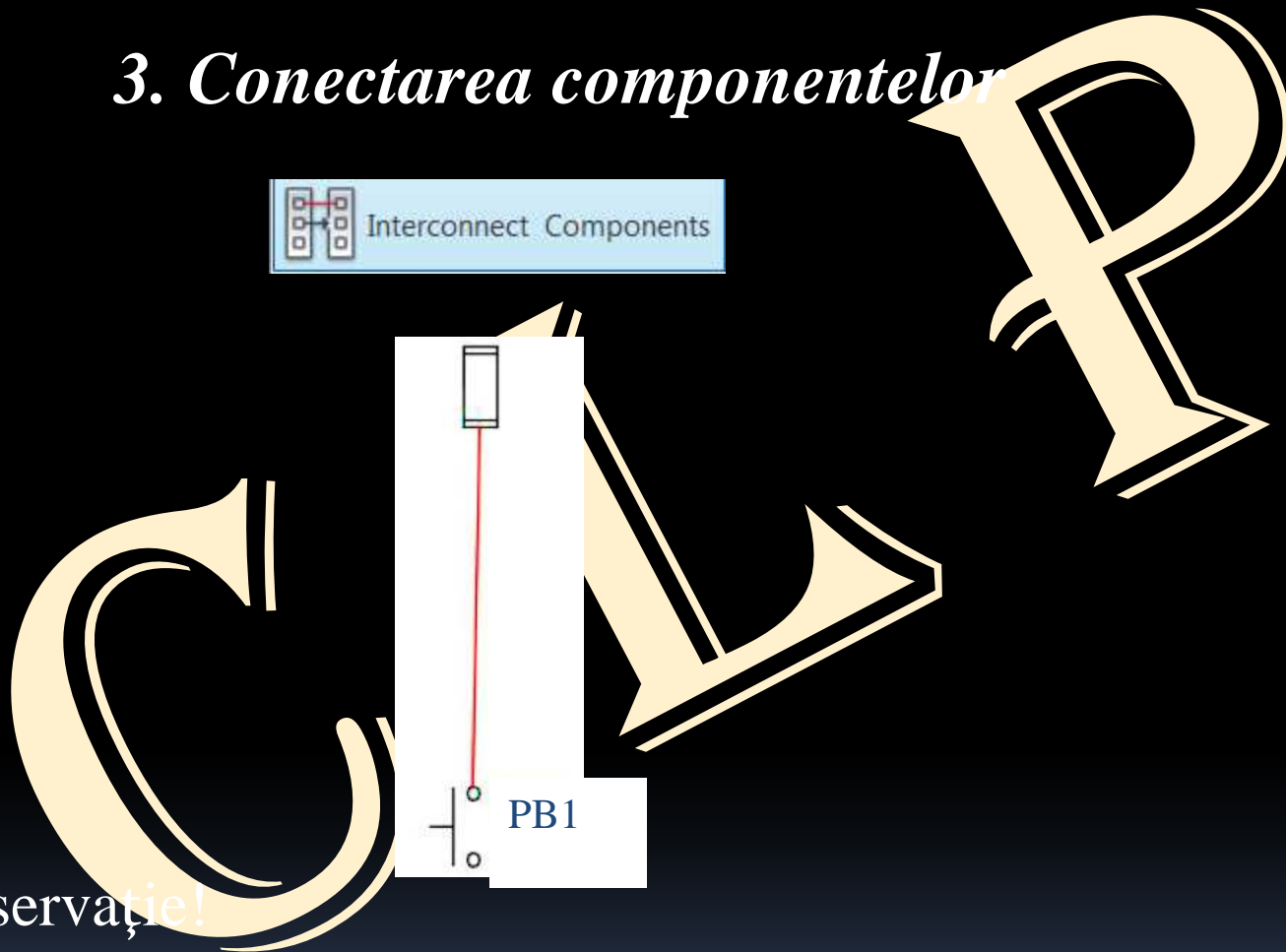


Collision on

2. Evidențierea înclinării firelor



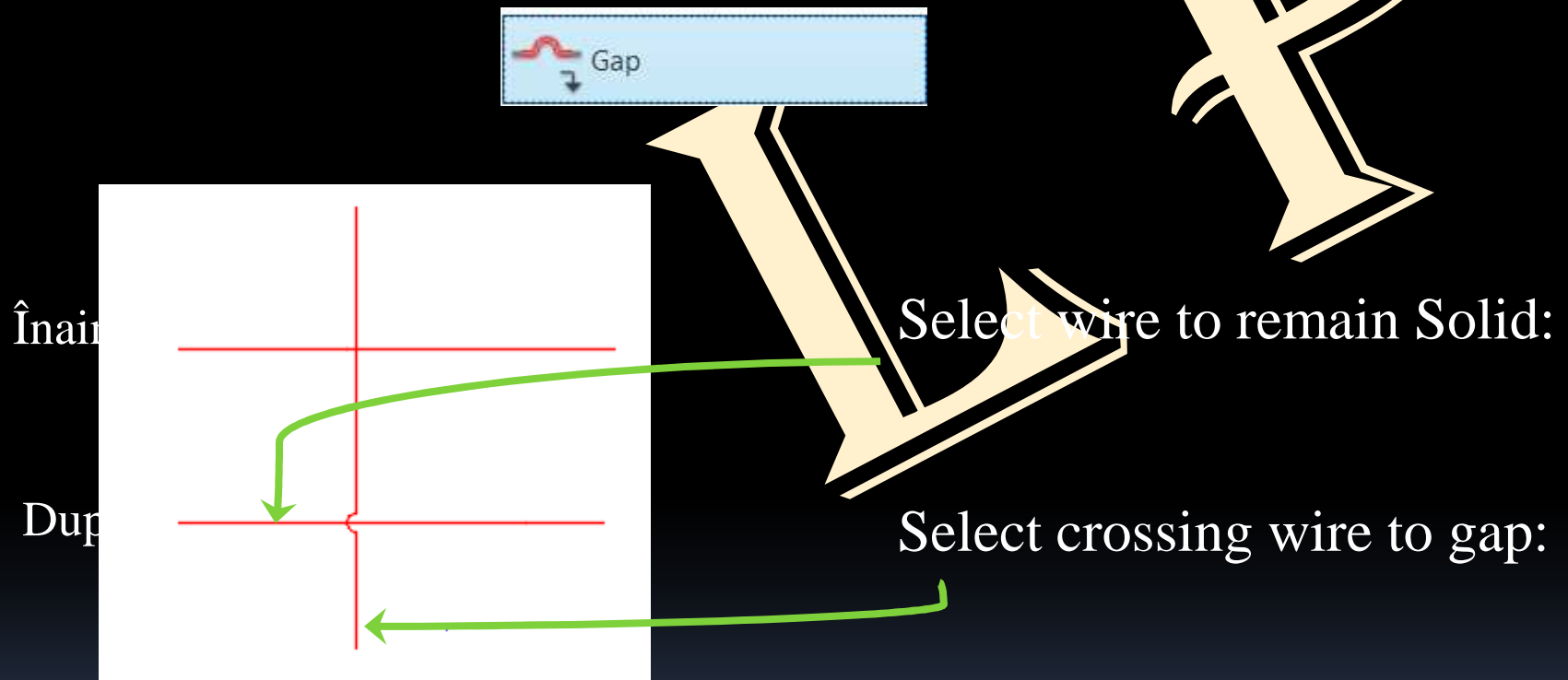
3. Conectarea componentelor

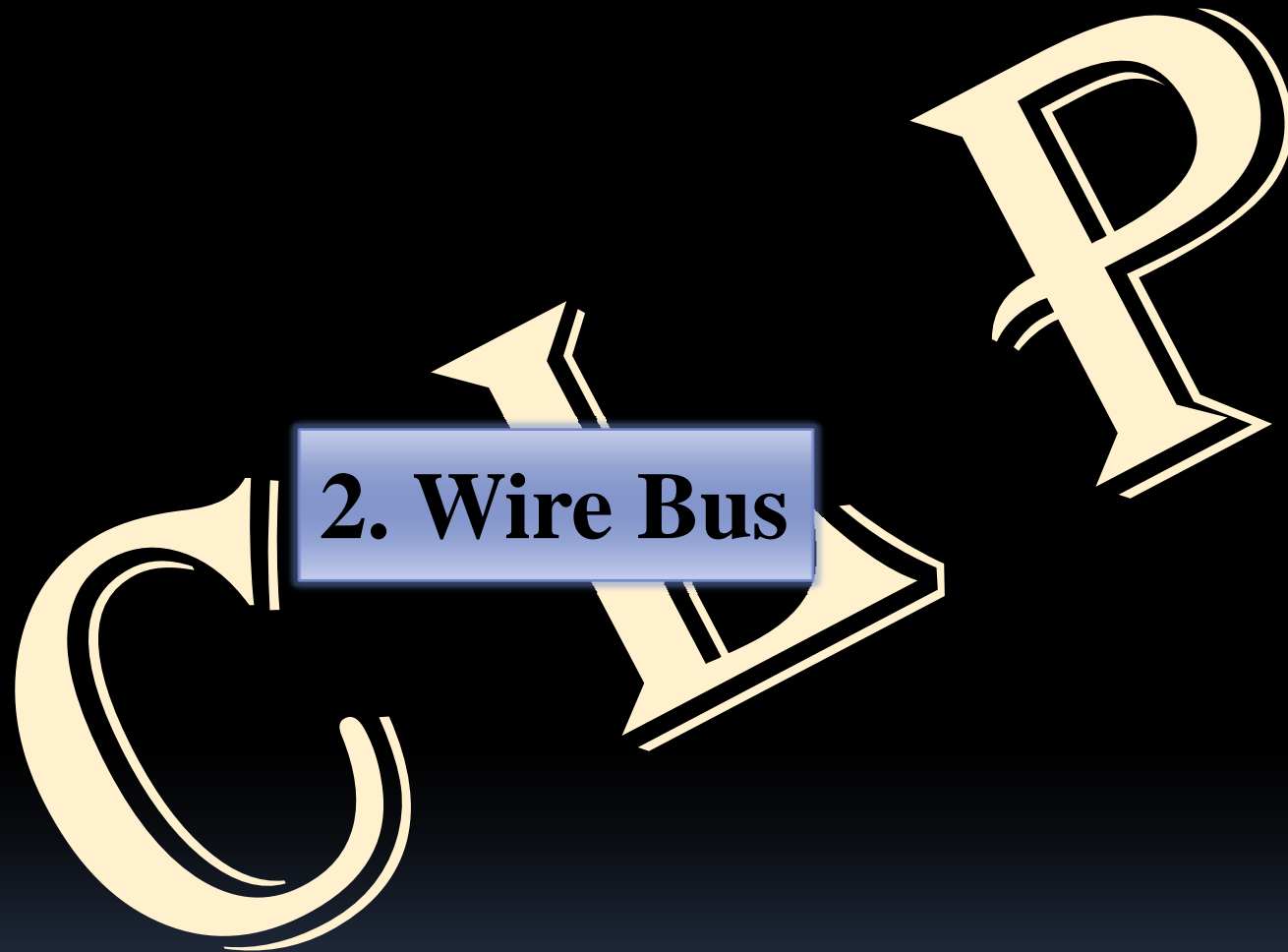


Observație!

Firele se adaugă numai dacă punctele de conexiune ale cablurilor sunt aliniat.

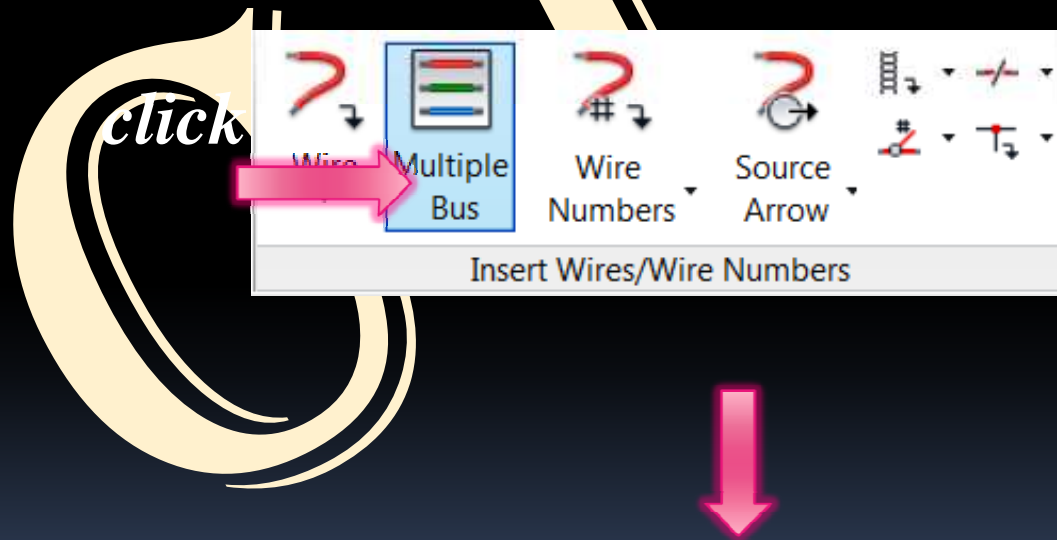
4. Evidențierea intersecției firelor





2. Wire Bus

Afişul Multiple Bus, panel Insert Wire/Wire Numbers, Ribbon Schematic



Caseta de dialog Multiple Wire Bus

Multiple Wire Bus

Horizontal

Spacing: 0.5000

Vertical

Spacing: 0.5000

Starting at:

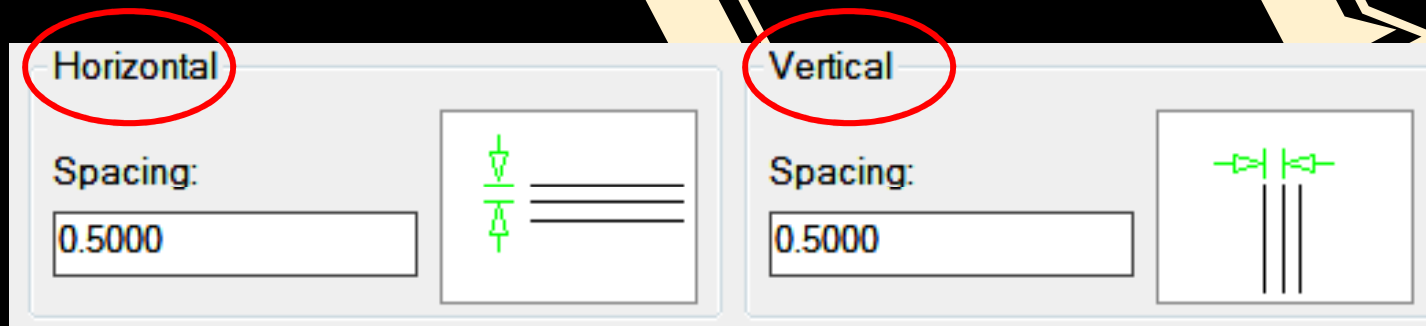
- Component (Multiple Wires)
- Another Bus (Multiple Wires)
- Empty Space, Go Horizontal
- Empty Space, Go Vertical

3 Number of Wires 2 3 4

OK Cancel Help

OPȚIUNI

◆ **Spacing**, specifică distanța orizontală și verticală dintre fire

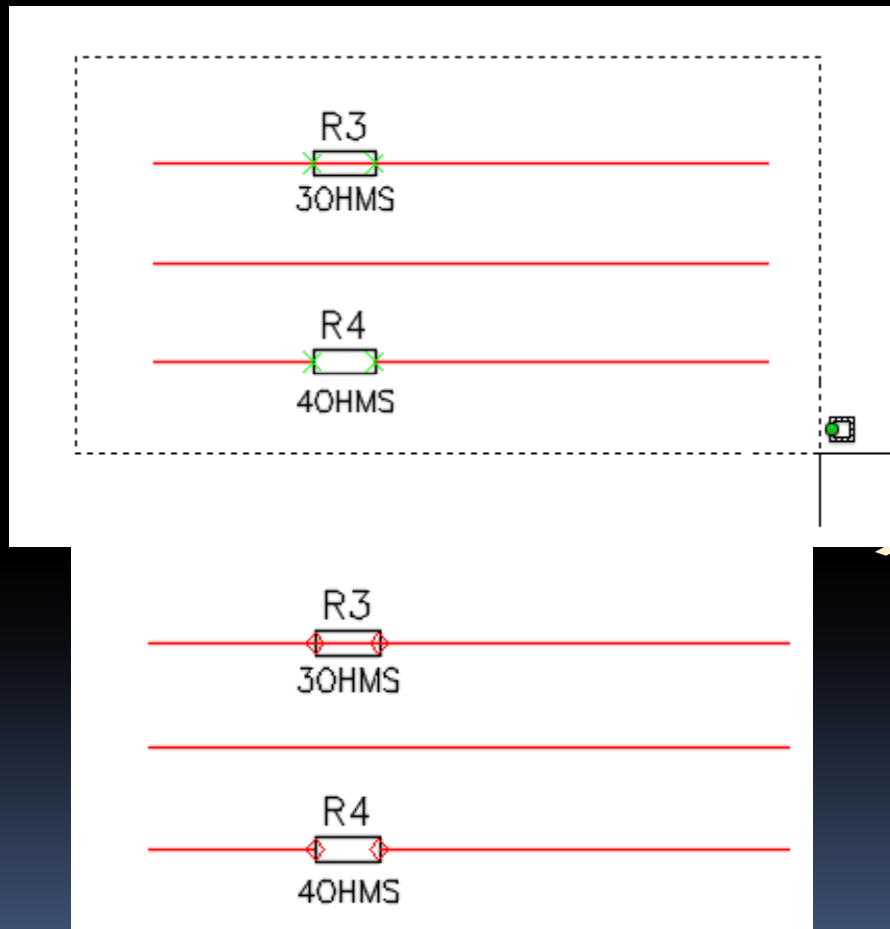


- ◆ **Starting**, specifică tipul magistralei de fire

Starting at

- Component (Multiple Wires)
- Another Bus (Multiple Wires)
- Empty Space, Go Horizontal
- Empty Space, Go Vertical

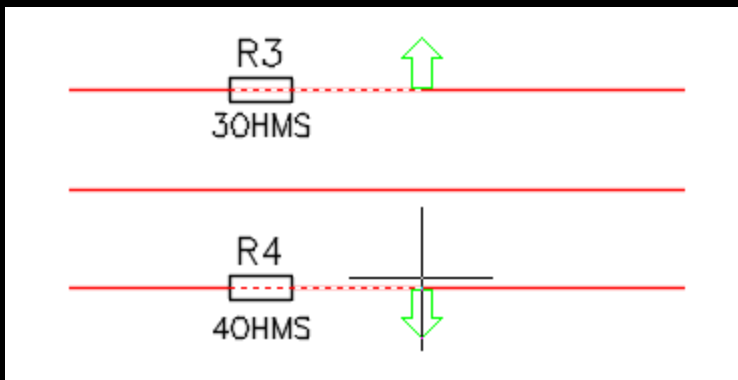
›Component (Multiple Wires)



Fereastră de selecție
pentru porturile de
conectare

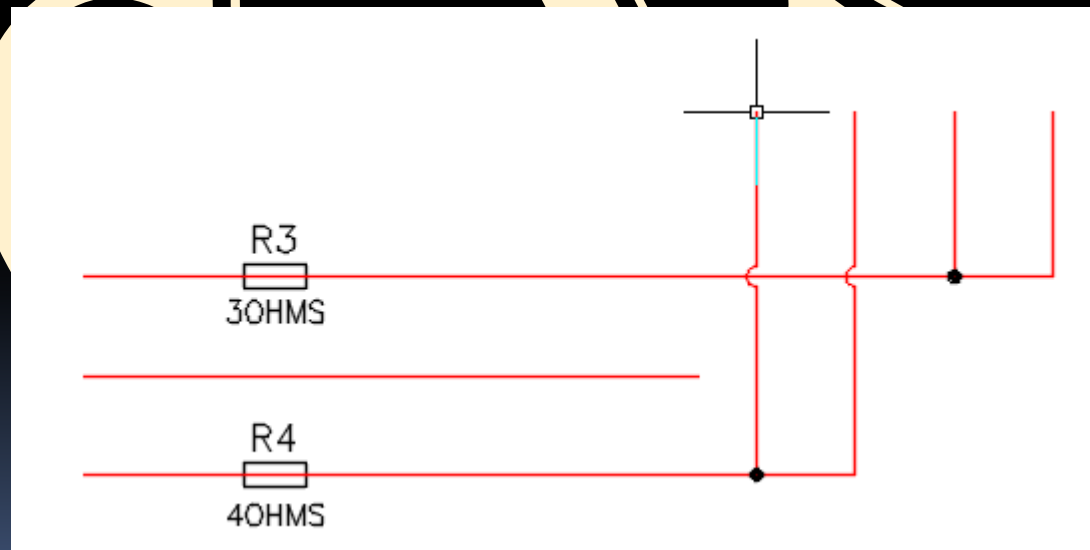
Evidențierea porturilor
de conectare

ENTER



Cablurile vor fi atașate cursorului

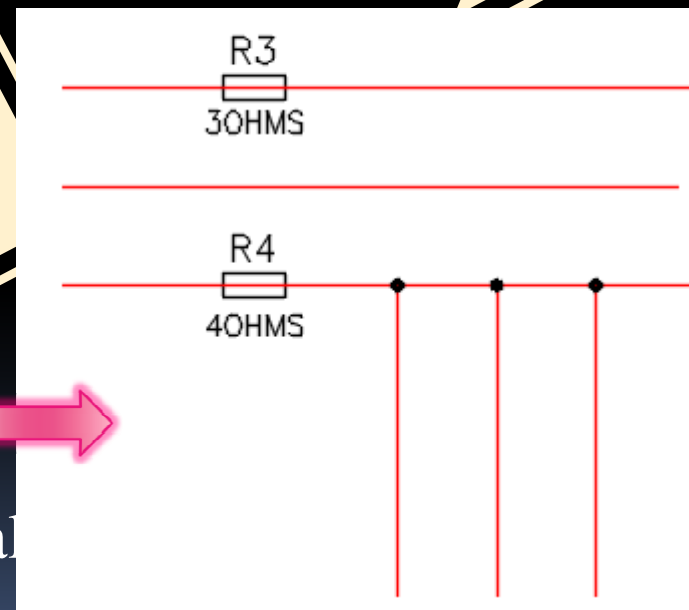
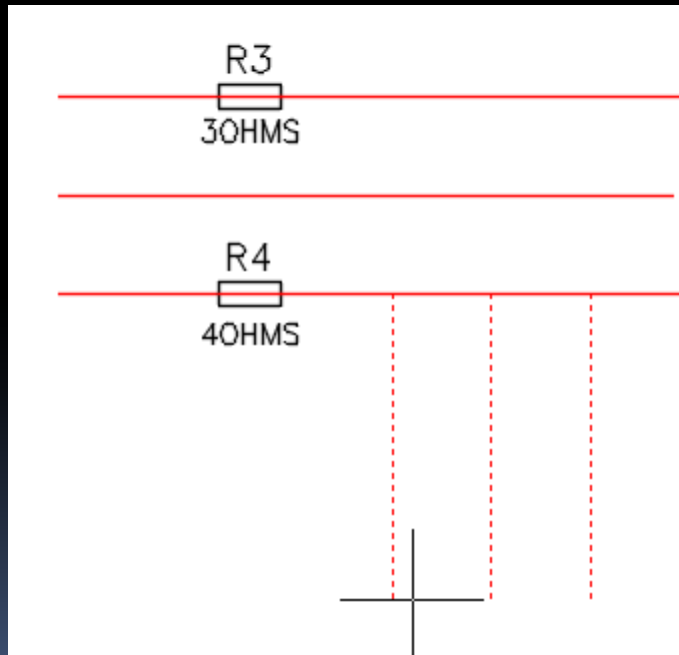
Magistrala este creată



› *Another Bus (Multiple Wires)*

Specificați numărul de fire din
casetă de editare **Number of Wires**

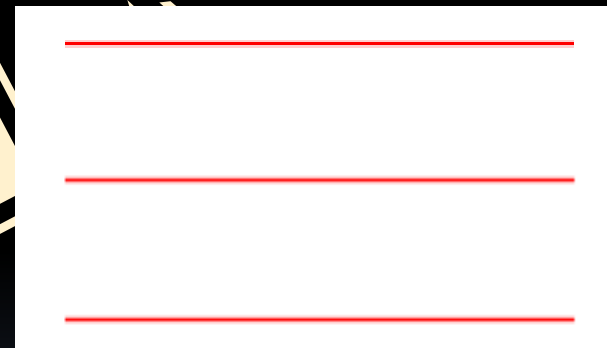
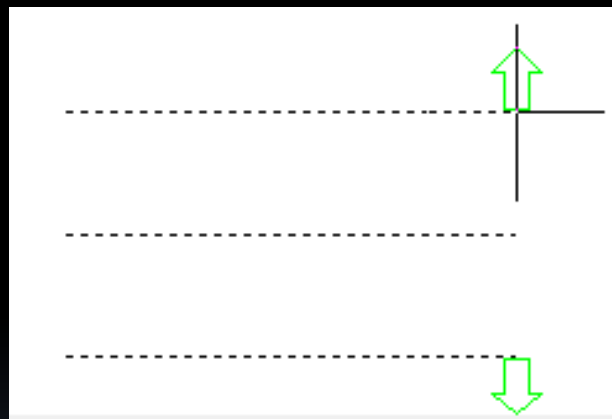
Number of Wires



agistra

› *Empty Space, Go Horizontal*

Crează o magistrală pe direcție orizontală

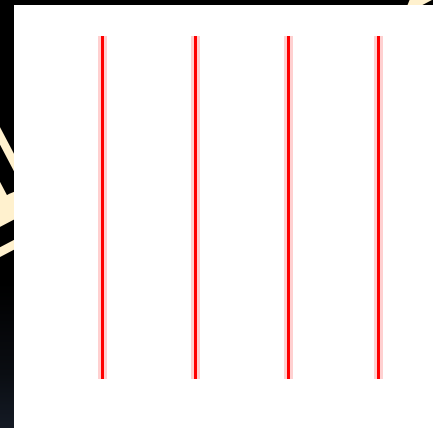
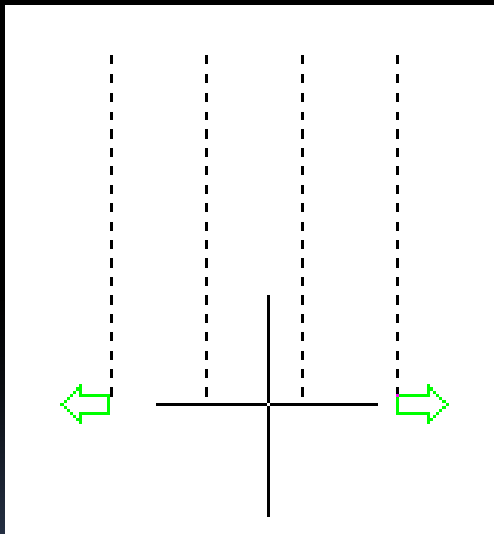


Specificați numărul de fire din
casetă de editare **Number of Wires!!**

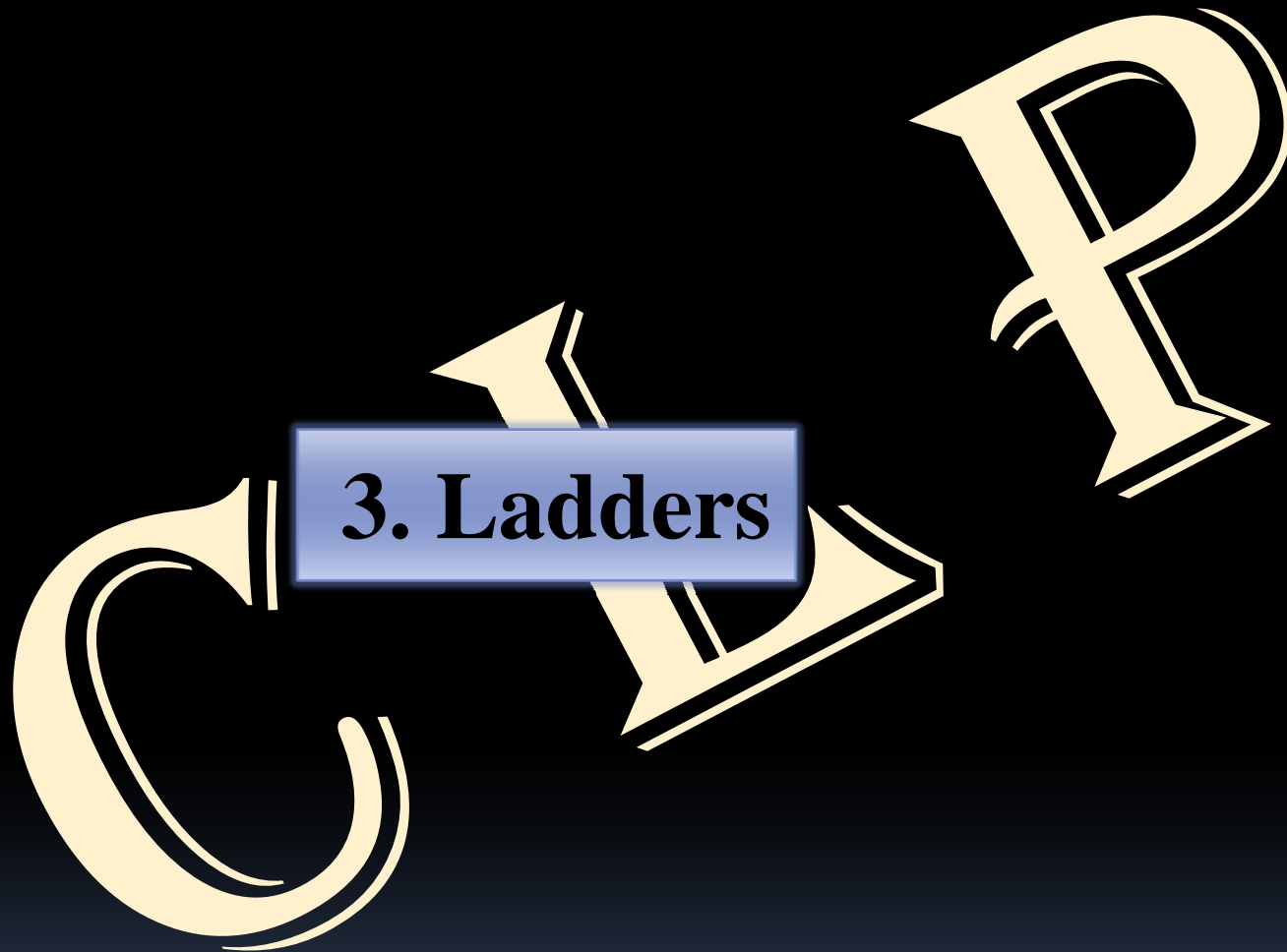
› *Empty Space, Go Vertical*



Crează o magistrală pe direcție verticală

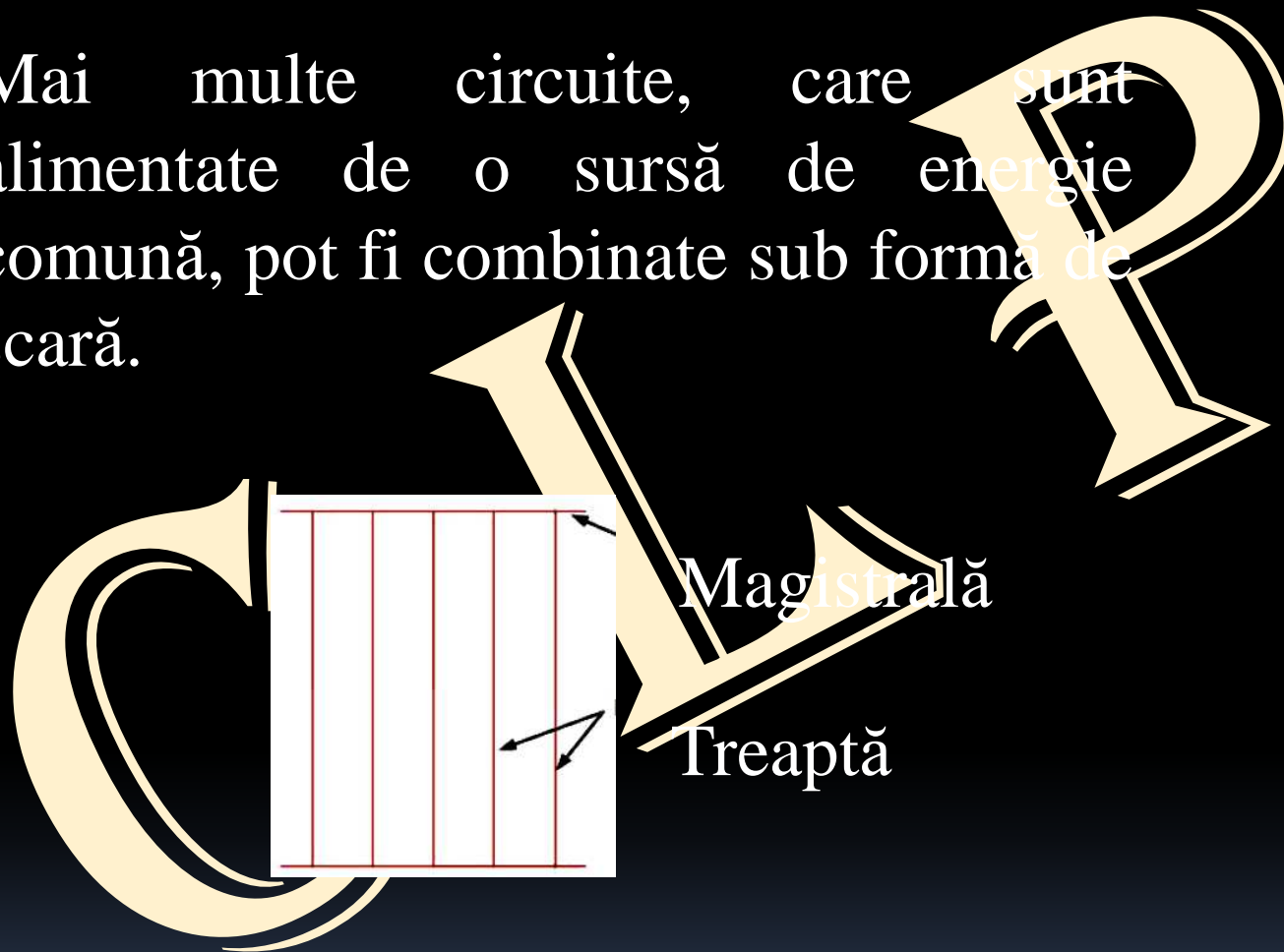


Specificați numărul de fire din
casetă de editare **Number of Wires!!**



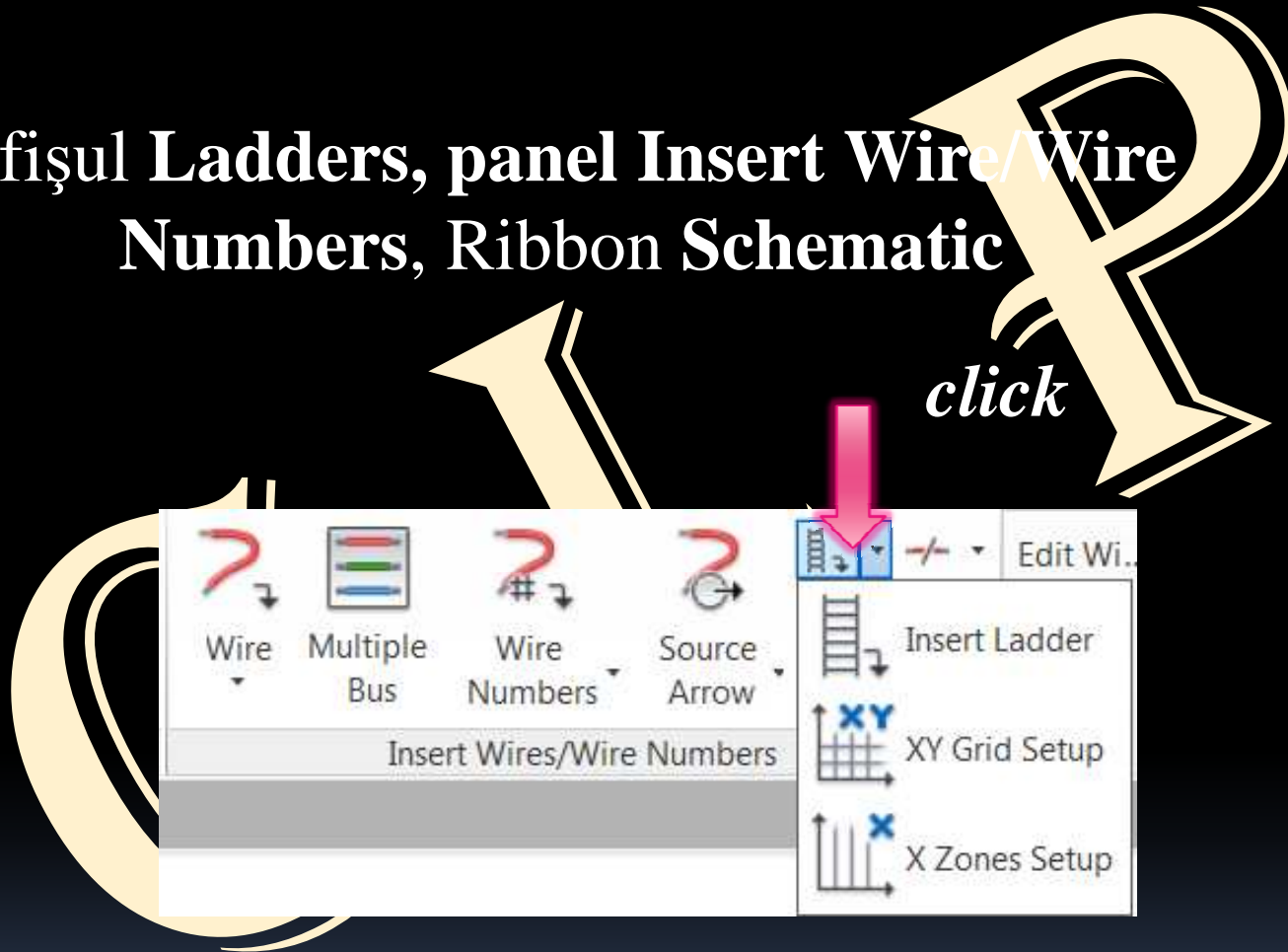
3. Ladders

Mai multe circuite, care sunt alimentate de o sursă de energie comună, pot fi combinate sub formă de scară.



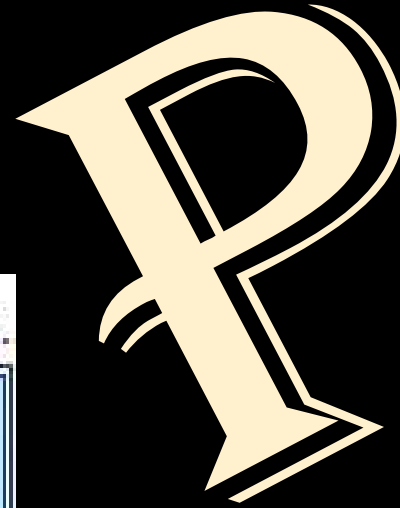
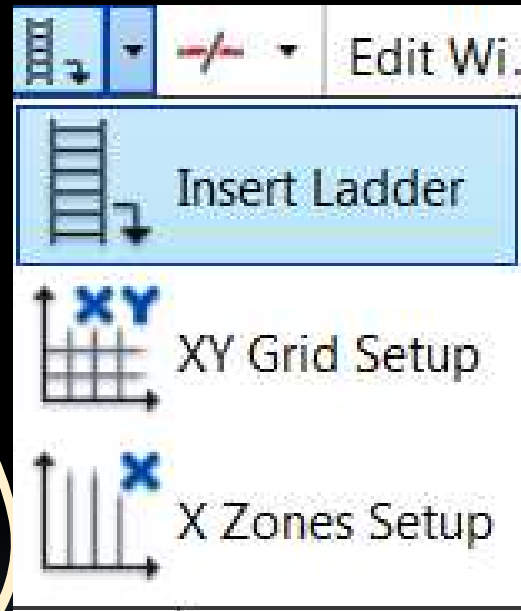
Aranjament tip scară

Afişul Ladders, panel Insert Wire/Wire Numbers, Ribbon Schematic



◆ Insert Ladder

click



Caseta de dialog Sheet 1: Insert Ladder

Sheet: 1 - Insert Ladder

Width: 5.0000

Spacing: 0.5000

Length: []

Rungs

1st Reference: 204

Index

Without reference numbers

Phase:

1 Phase

3 Phase

0.5000 Spacing

Draw Rungs:

No Bus

No Rungs

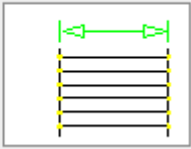
Yes

Skip

OK Cancel Help

OPȚIUNI


Width



5.0000

Setează distanța dintre două magistrale

Spacing



0.5000

Setează distanța dintre două trepte

Length

Rungs

Setează lungimea totală a scării

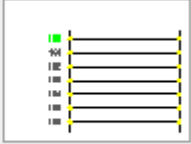
Setează numărul de trepte

1st Reference

204

1 Index

Without reference numbers




Setează referința în numerotarea scării

Phase

1 Phase

3 Phase

0.5000 Spacing



Setează numărul de faze


Draw Rungs

No Bus

No Rungs

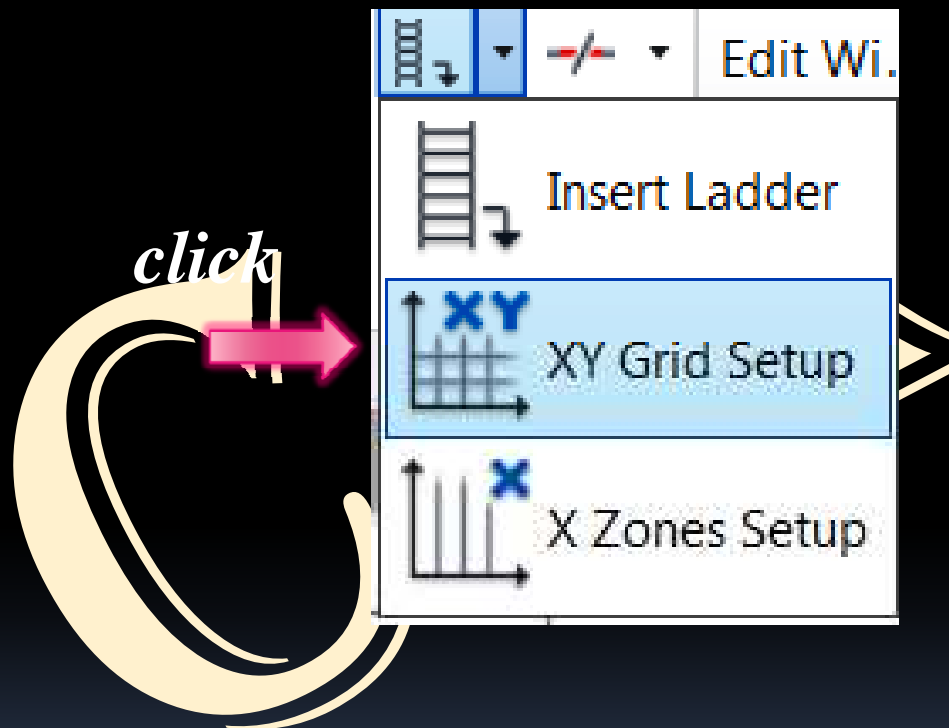
Yes

0 Skip



Setează modul de reprezentare al treptelor

◆ XY Grid Setup

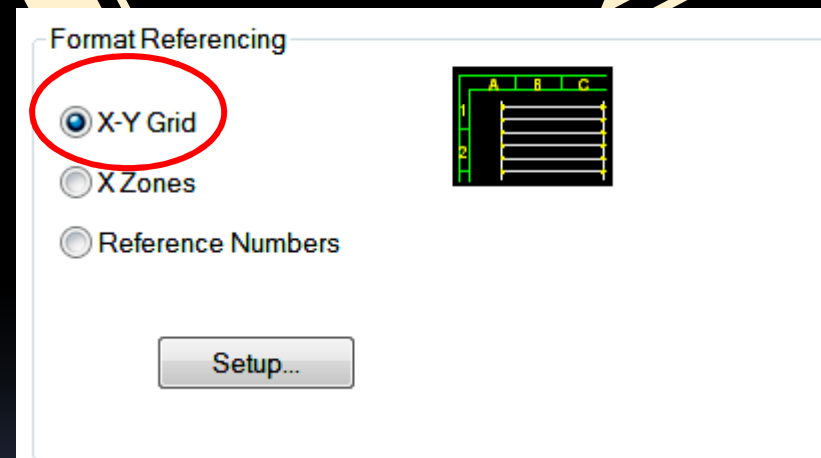
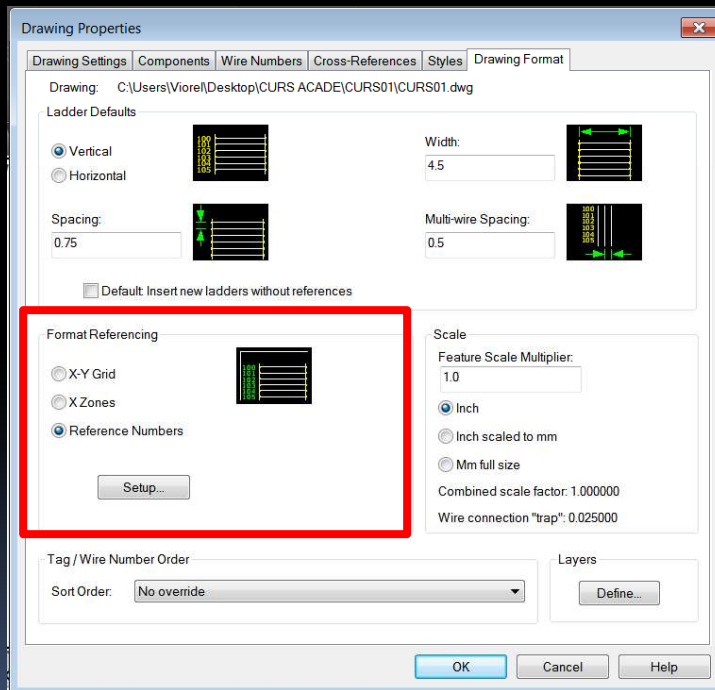


Se configurează o rețea de fire pe baza valorilor specificate.

Observație!

În tabul *Drawing properties* setați X-Y Grid!

P



Caseta de dialog X-Y Grid

X-Y Grid Setup

Origin

Pick>>

X

Y

Spacing

2.0 Horizontal

2.0 Vertical

X-Y format

Horizontal-Vertical

Vertical-Horizontal

- Separator

Grid labels: 1st only or complete list
(list separate with commas A,B,C,D,...)

A Horizontal

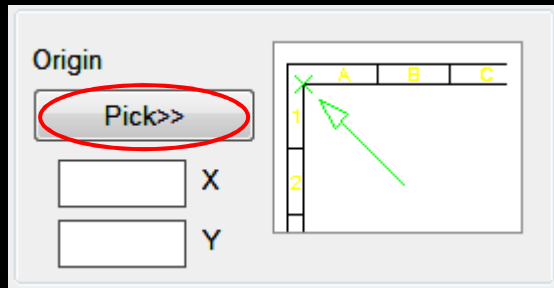
1 Vertical

Insert X-Y grid labels

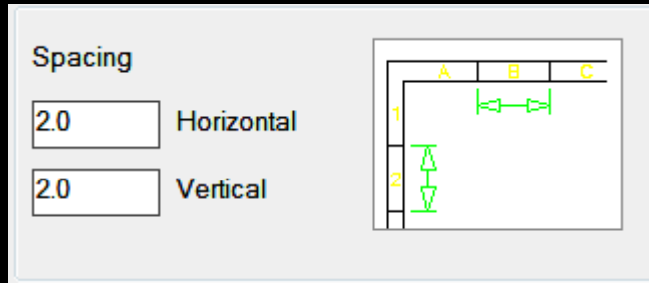
Horizontal count 1 Vertical count 1

OK Cancel Help

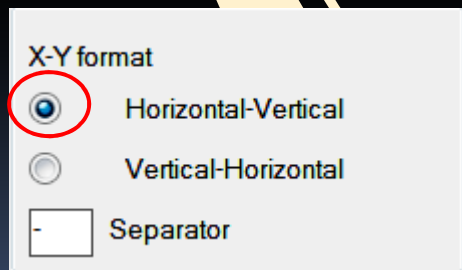
OPȚIUNI



Specifică originea grilei



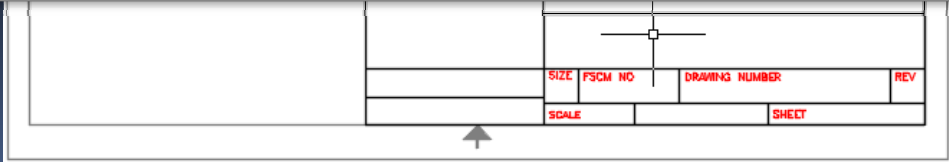
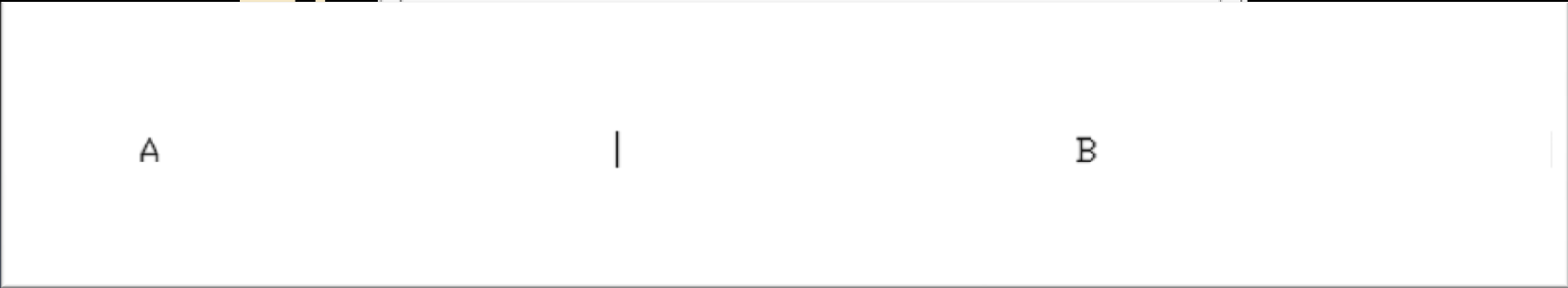
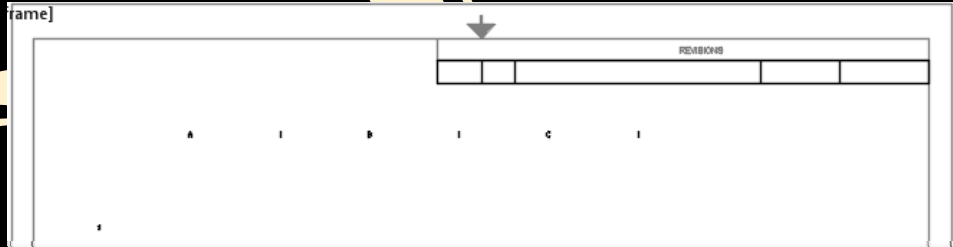
Setează distanța dintre liniile de grilă



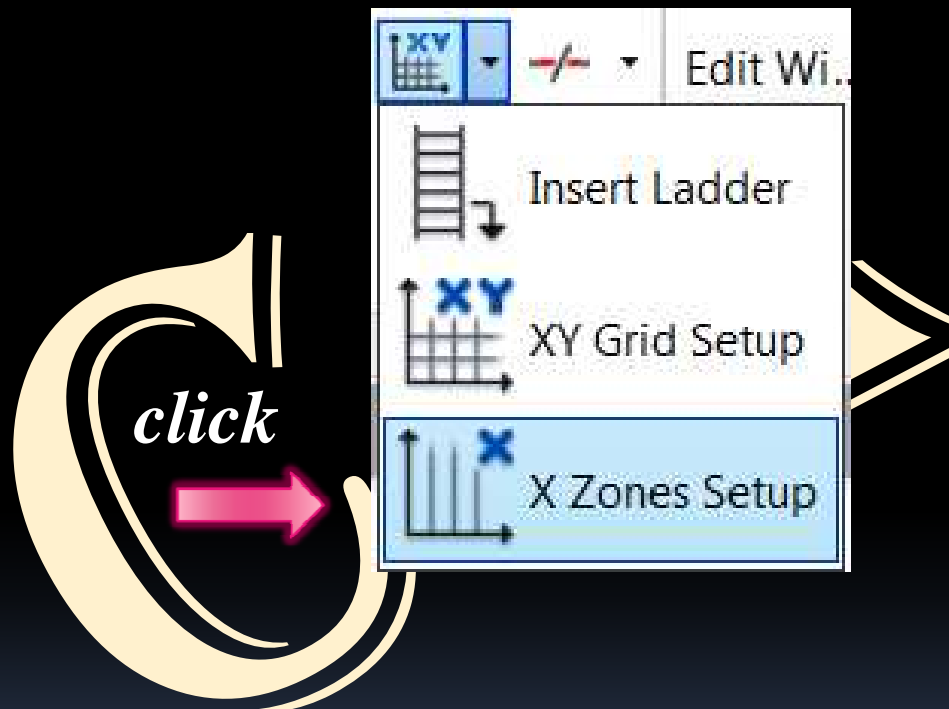
Setează formatul grilei

Insert X-Y grid labels
Horizontal count

Afișează etichetele grilei



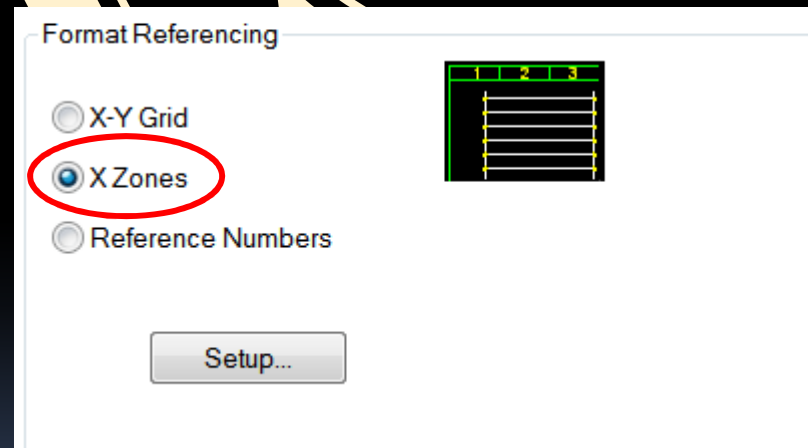
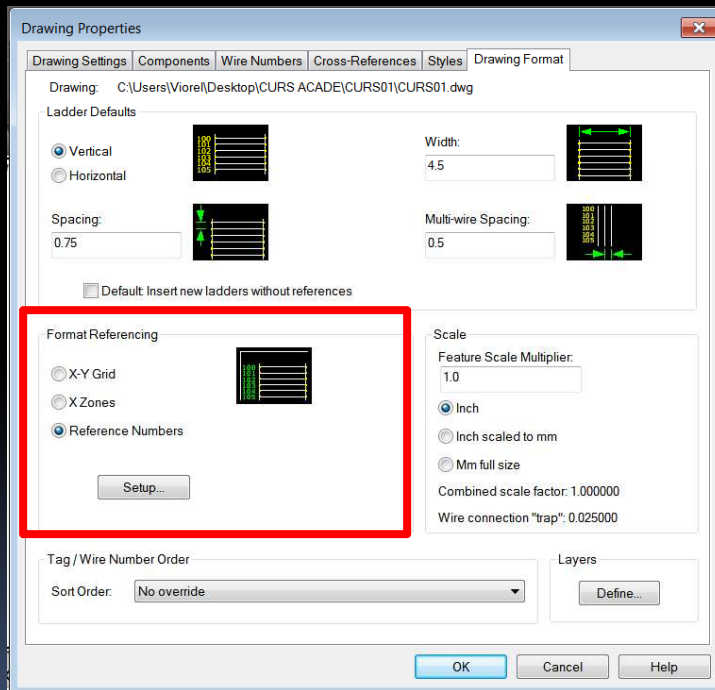
◆ X Zones Setup



Crează referințe verticale în desen
pentru inserarea componentelor.

Observație!
În tabul *Drawing properties* setați XZ Zones!

P



Caseta de dialog X Zones Setup

X Zones Setup

Origin

Pick>>

1.000 X

6.500 Y

Spacing

2.0 Horizontal

Zone labels: Enter 1st only or complete list
(if list separate with commas, ex: 10-1,10-2,10-3,...)

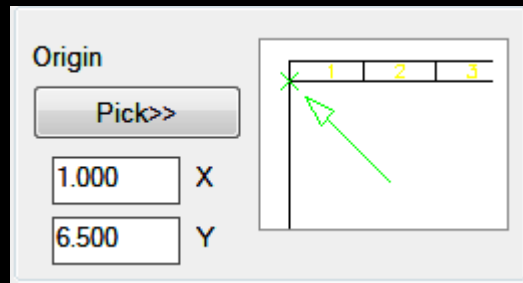
A,B,C Horizontal

Insert zone labels

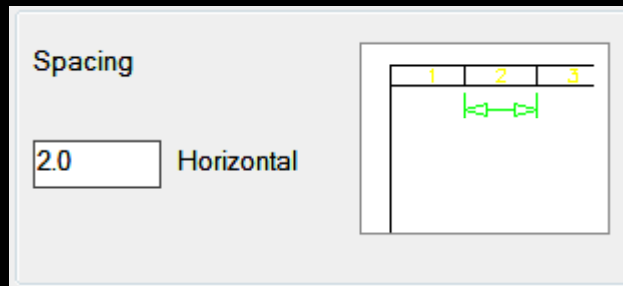
Zone count 3

OK Cancel Help

OPTIUNI



Specifică originea grilei



Setează distanța dintre liniile de grilă



Afișează etichetele grilei

ne]

↓

REVISIONS			

A | B | C |

	Sample Project - Schematics and Panel Layout			
	SIZE	FSCM NO	DRAWING NUMBER	REV
	SCALE			SHEET

↑

A		B		C	
---	--	---	--	---	--



CONCLUZIE

Grilele XY și zonele X sunt folosite pentru a crea sisteme de referințe, necesare pentru a introduce mai ușor componentele electrice.

CLP



VI. Numerotarea firelor



Numerotarea firelor este un instrument important al schemelor electrice.

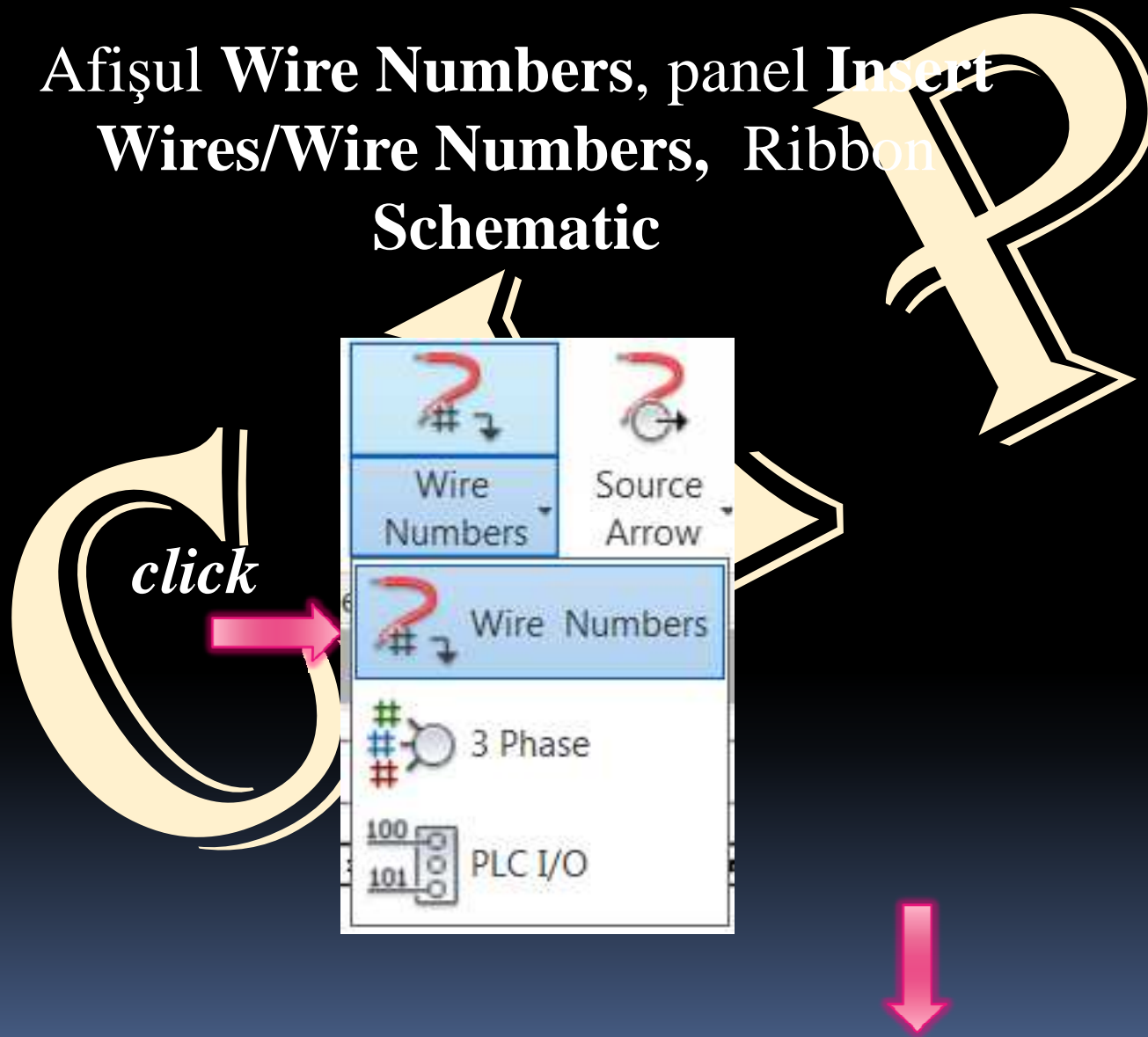
Furnizează detalii despre:

- › identificarea circuitelor legate de acesta,
- › identificarea componentelor conectate,
- › lungimea totală a firului.



1. Wire Numbers

Afişul Wire Numbers, panel Insert Wires/Wire Numbers, Ribbon Schematic



Caseta de dialog Sheet1 - Wire Tagging

Sheet 1 - Wire Tagging

To do

Tag new/un-numbered only

Tag/retag all

Cross-reference Signals

Freshen database (for Signals)

Wire tag mode

Sequential

Start

Increment

Line Reference

Format override

Wire tag format

Use wire layer format overrides

Setup

Insert as Fixed

Project-wide

Pick Individual Wires

Cancel

Help

Drawing-wide

◆ Cum se face numerotarea

To do

Tag new/un-numbered only

Tag/retag all

Specifică numărul de fir pentru firele nenumerotate.

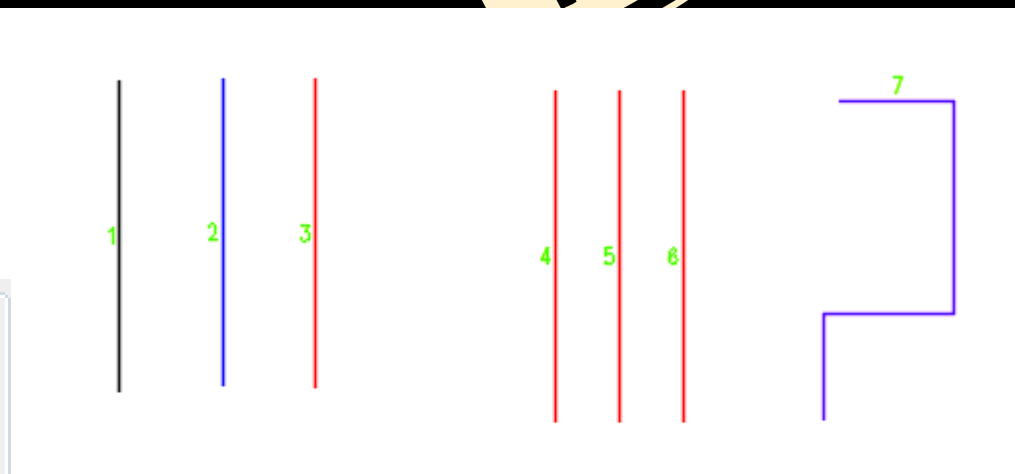
To do

Tag new/un-numbered only

Tag/retag all

Specifică numărul de fir pentru toate firele!

◆ Modul de numerotare



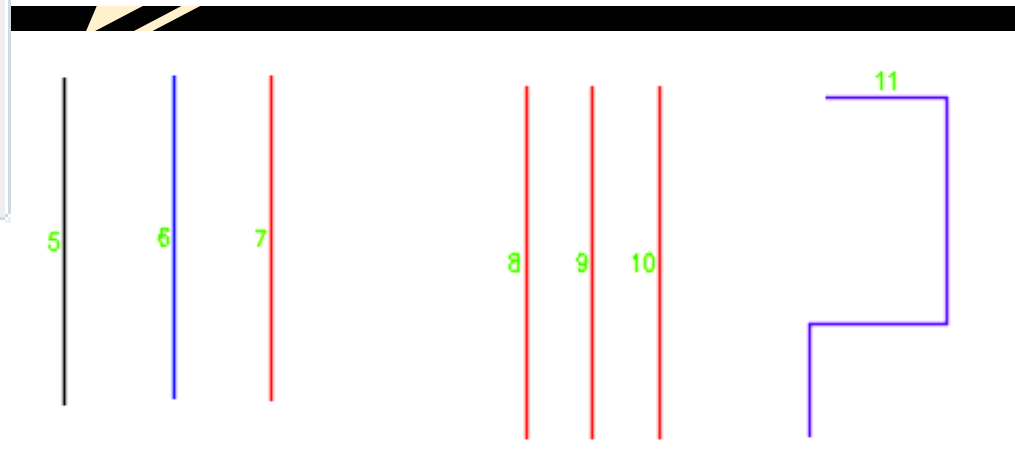
Wire tag mode

Sequential

X-Y Grid

1 Start

1 Increment





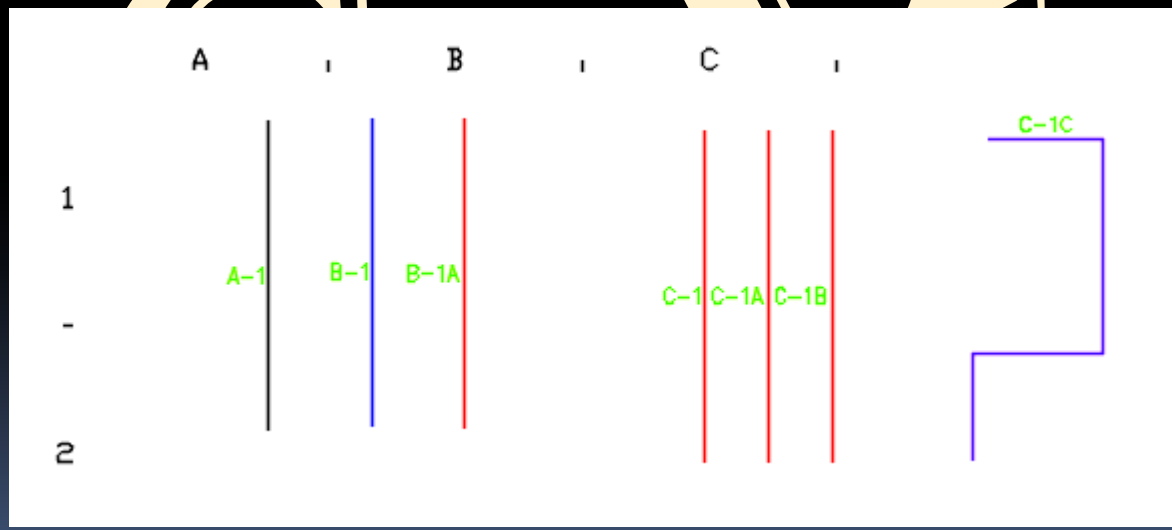
Wire tag mode

Sequential

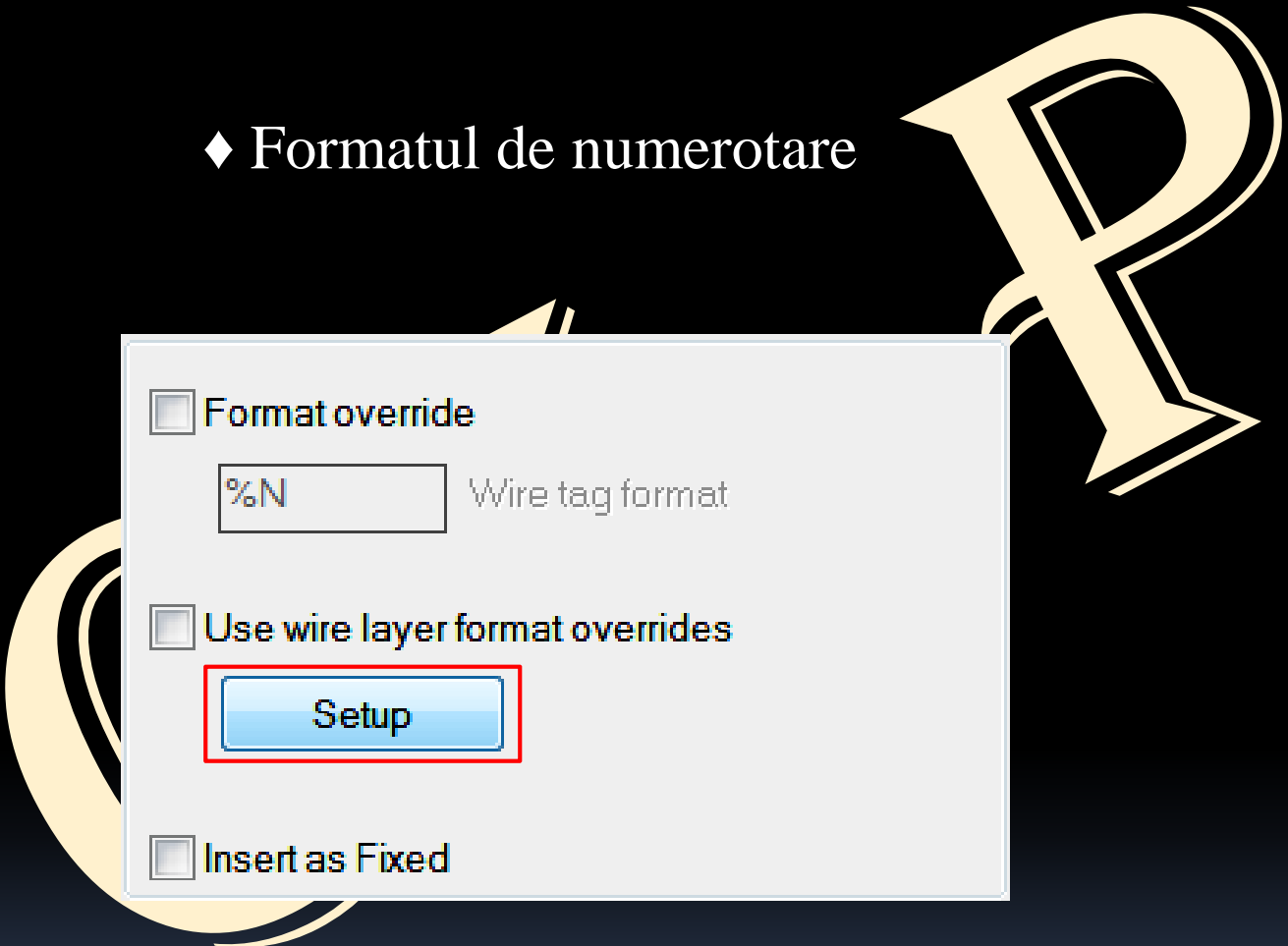
Start

Increment

X-Y Grid



◆ Formatul de numerotare



Format override

Wire tag format

Use wire layer format overrides

Insert as Fixed



Caseta Assign Wire Numbering Formats by Wire Layers



Assign Wire Numbering Formats by Wire Layer ✕

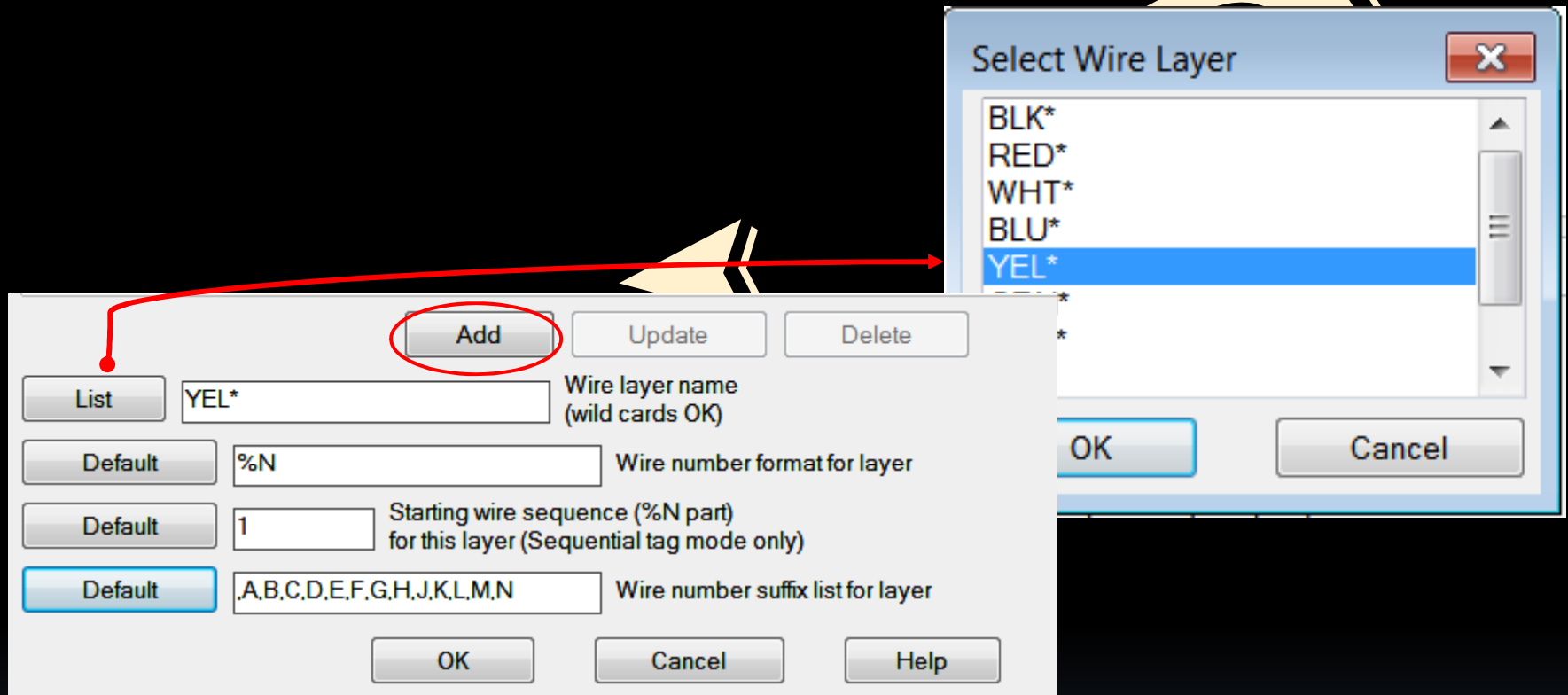
Wire Layer	Wire Number Format	Sequential start %N	Suffix List
-none-			

Wire layer name (wild cards OK)

Wire number format for layer

Starting wire sequence (%N part) for this layer (Sequential tag mode only)

Wire number suffix list for layer



Click pe cele trei butoane **Default**, apoi **Add**

Assign Wire Numbering Formats by Wire Layer

Wire Layer	Wire Number Format	Sequential start %N	Suffix List
YEL*	%N	1	.A,B,C,D,E,F,G,H,J,K,L,M,N
BLU*	%N	1	.A,B,C,D,E,F,G,H,J,K,L,M,N
GRN*	--	--	--

Wire layer name (wild cards OK)

Wire number format for layer

Starting wire sequence (%N part) for this layer (Sequential tag mode only)

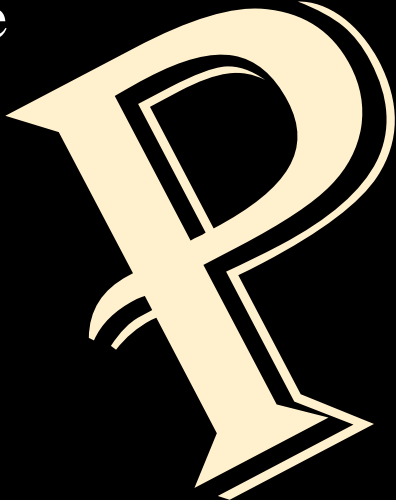
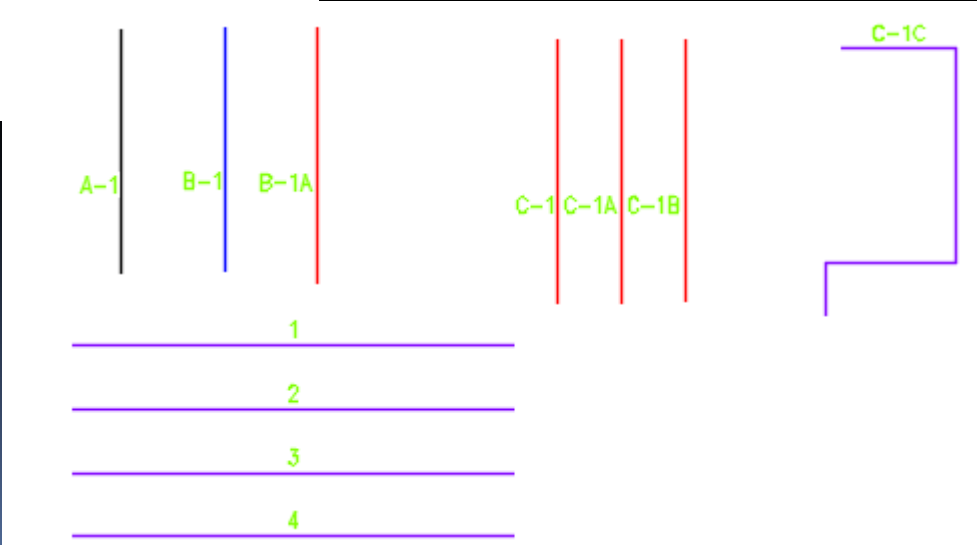
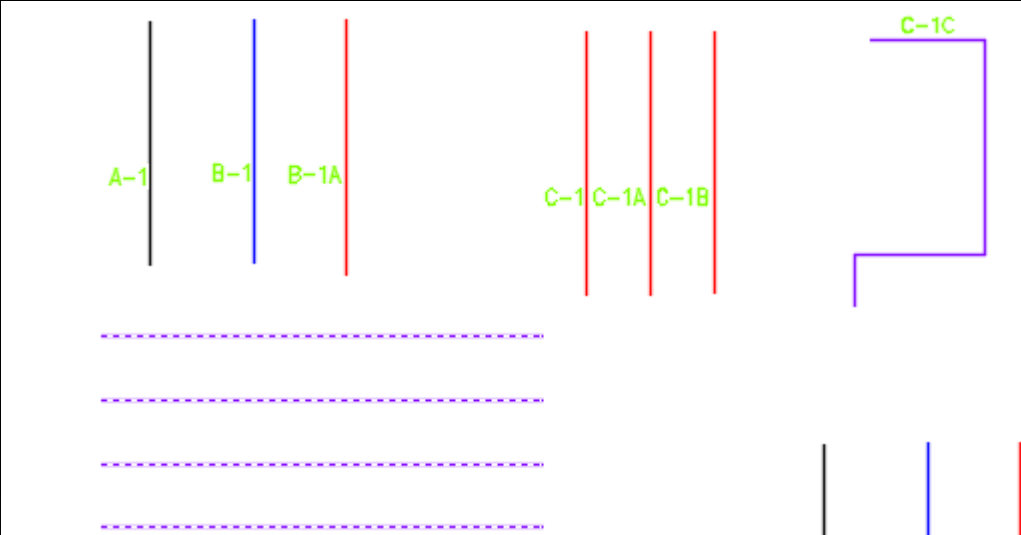
Wire number suffix list for layer

Use wire layer format overrides

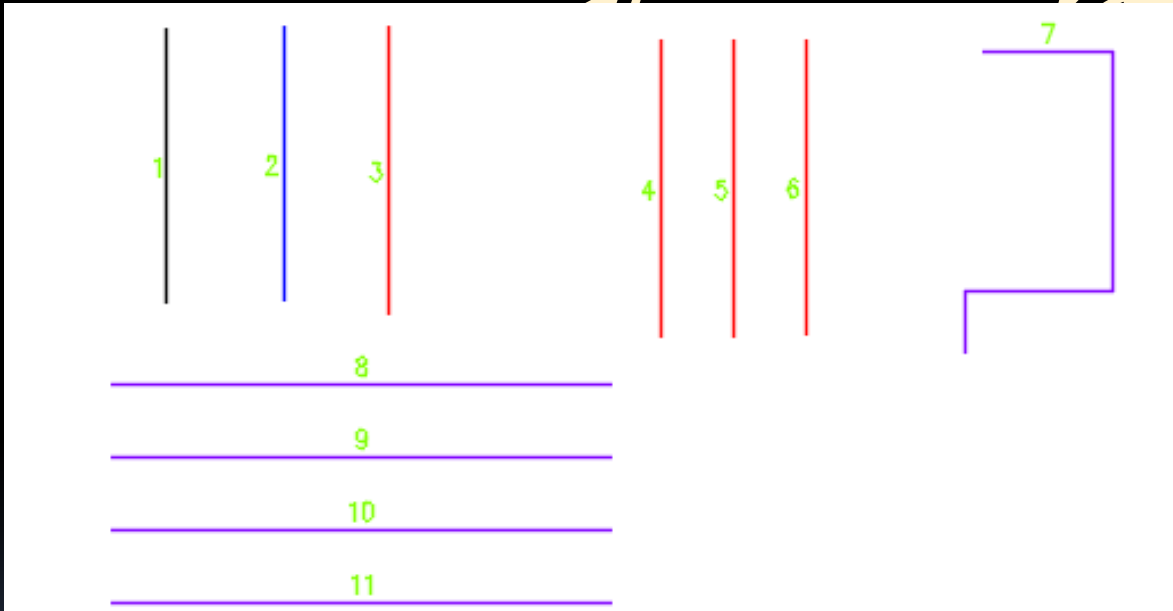
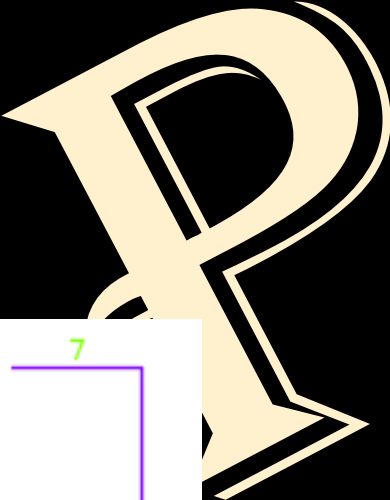
Insert as Fixed

Numerotarea se poate face

Pick Individual Wires



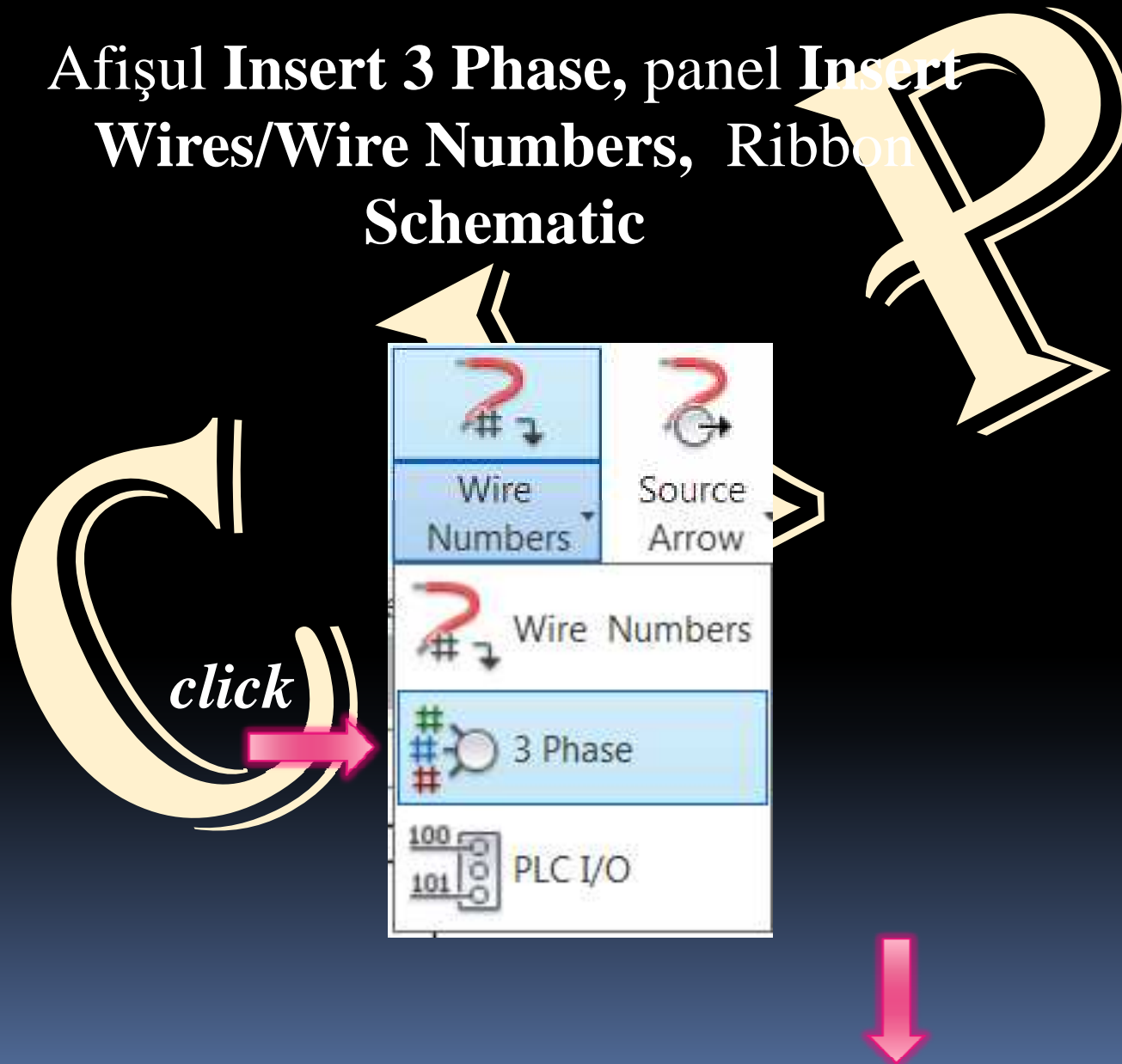
Drawing-wide





3. 3 Phase

Afişul Insert 3 Phase, panel Insert Wires/Wire Numbers, Ribbon Schematic



Caseta de dialog 3 Phase Wire Numbering

3 Phase Wire Numbering

Prefix: L1
 hold
 increment
List

Base: 1
 hold
 increment
< >
Pick

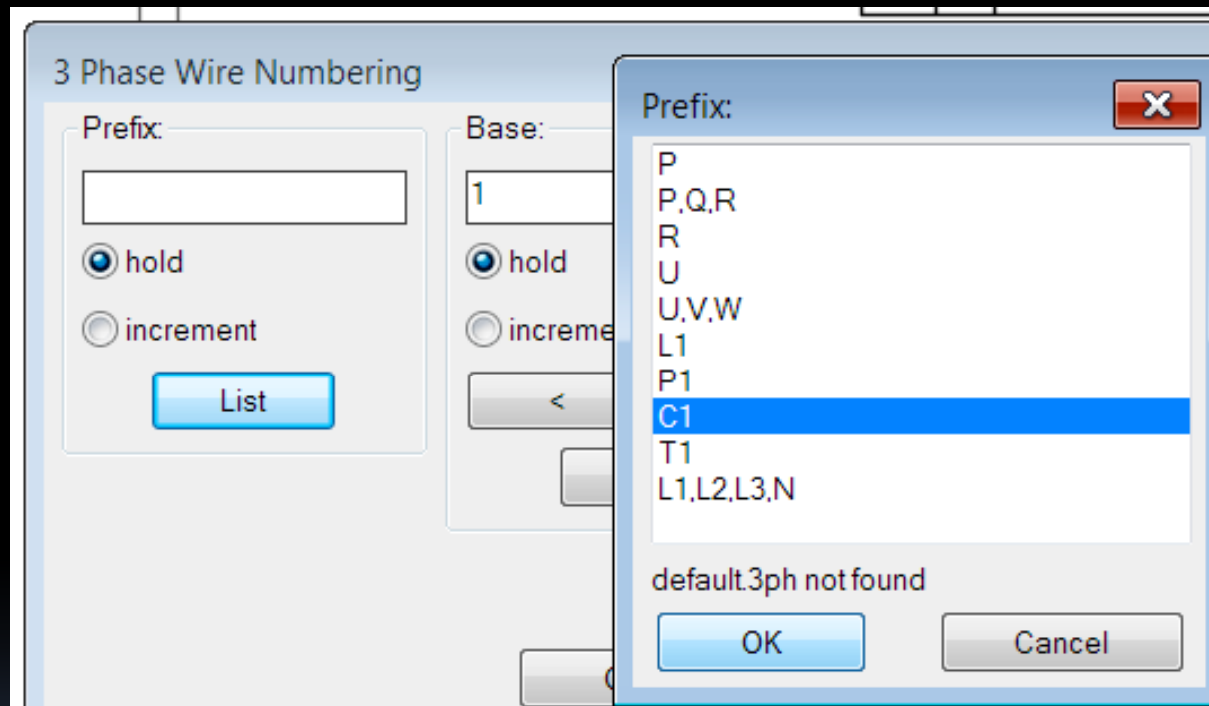
Suffix: C1
 hold
 increment
List

Wire Numbers:
L11C1
L12C1
L13C1

Maximum:
 3 4 None

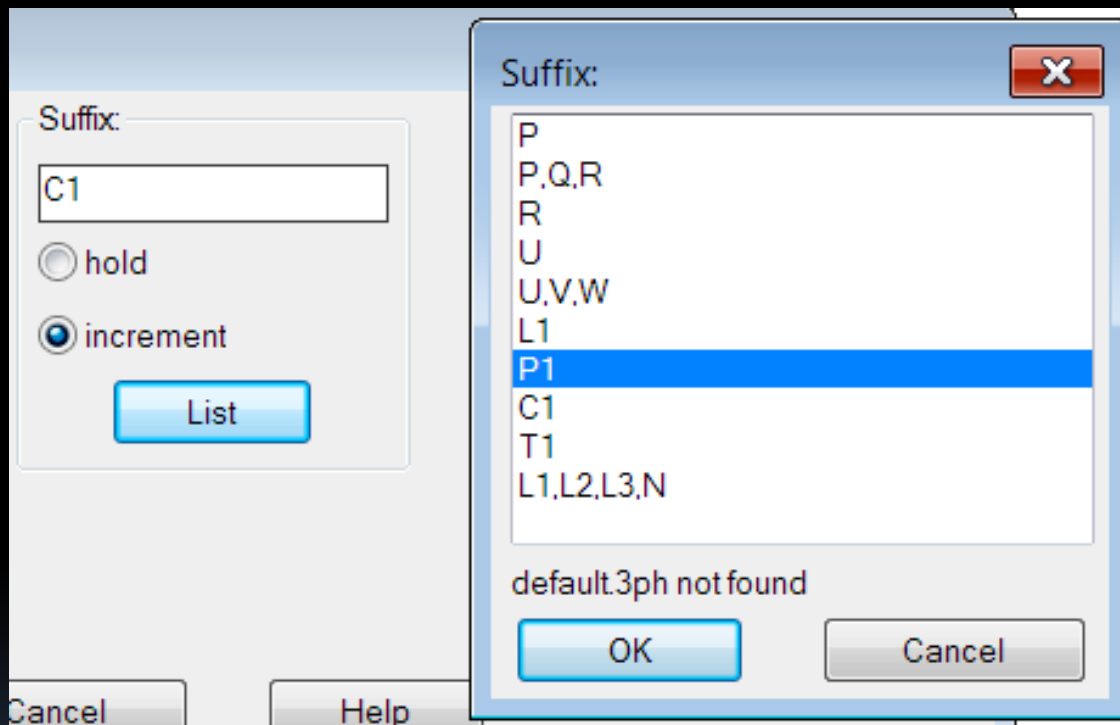
OK Cancel Help

Prefix



Se alege din lista un prefix!

Suffix



Se alege din lista un sufix!

Baza de numerotare

Base:

hold

increment

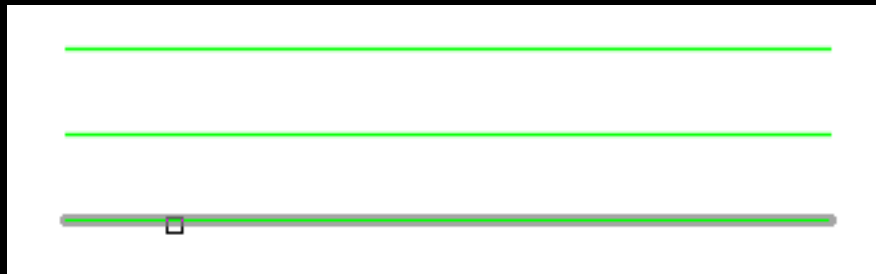
< >

Pick

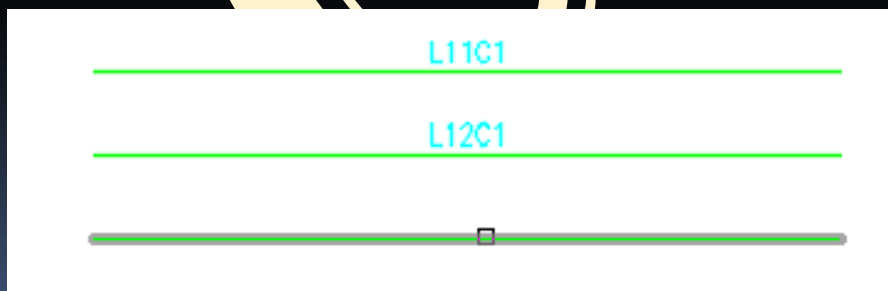
P

L

Selectați pe rând fiecare fir din
linia din linia trifazată.



Numerotați pe rând fiecare fir
din linia din linia trifazată.




OK

CANCEL



The background features a large, stylized yellow letter 'P' with a white outline, positioned in the upper right. A teal banner with a white border and a slight drop shadow is centered horizontally, containing the text 'VII. Adnotarea firelor'. The banner is set against a black background that transitions into a blue gradient at the bottom.

VII. Adnotarea firelor

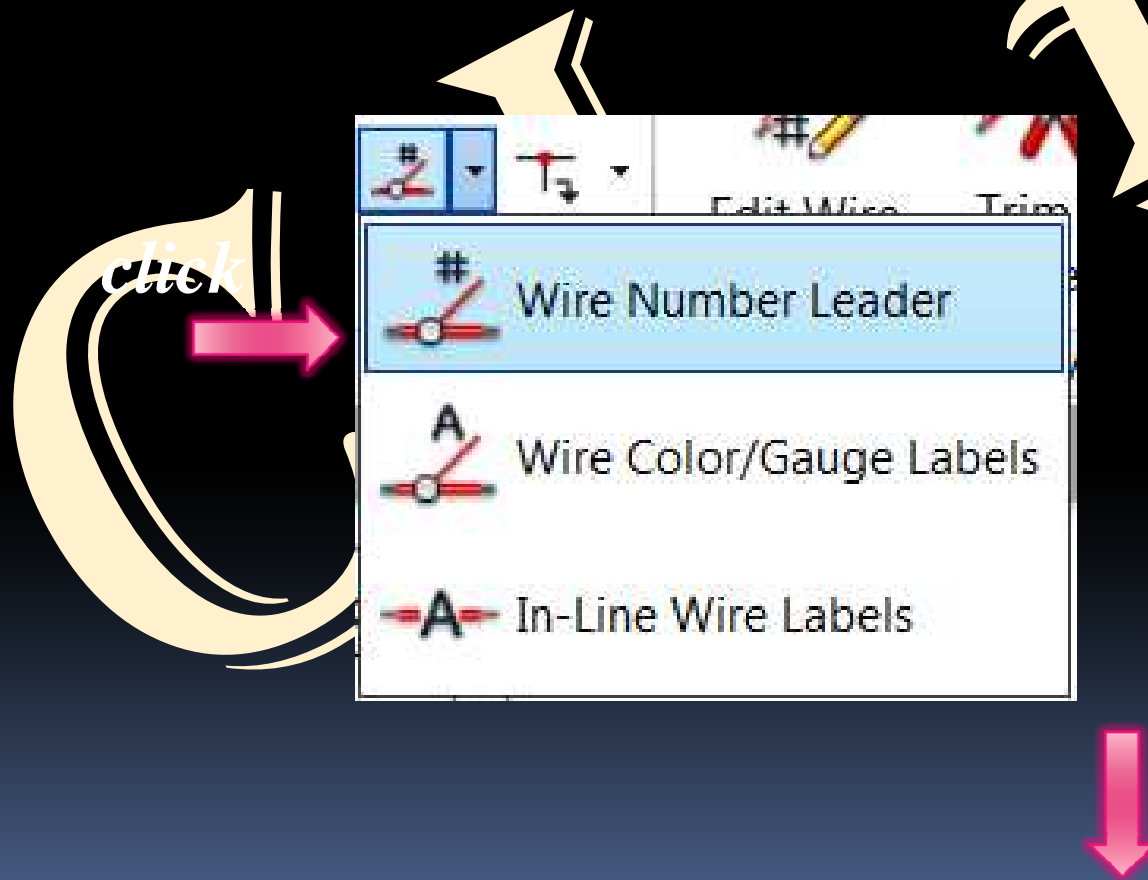


Adnotările și etichetele firelor
sunt, în general, atribuite automat,
dar acestea se pot modifica și
atribui conform nevoilor noastre.




1. Wire Number Leader

Afișul Wires Number Label, panel Insert Wires/Wire Numbers, Ribbon Schematic



Selectați un număr de fir!

Command:
Wire Leader

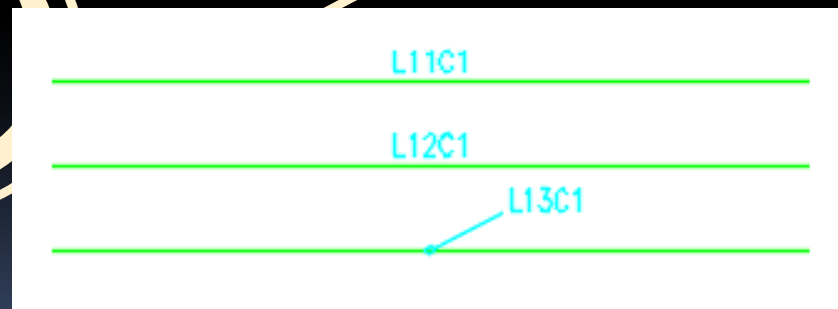
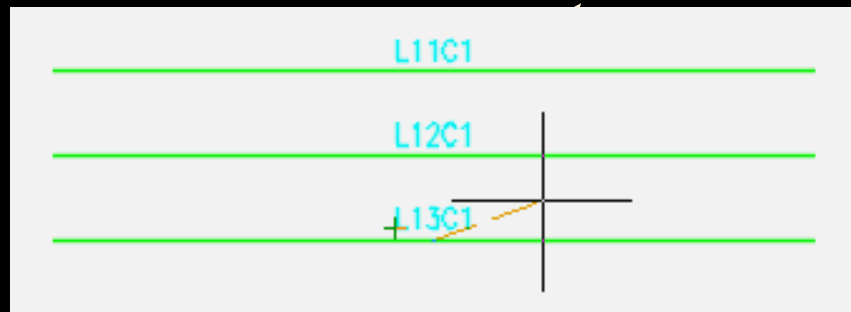
 Collapse/<Select wire number for LEADER>:

L11C1

L12C1

L13C1

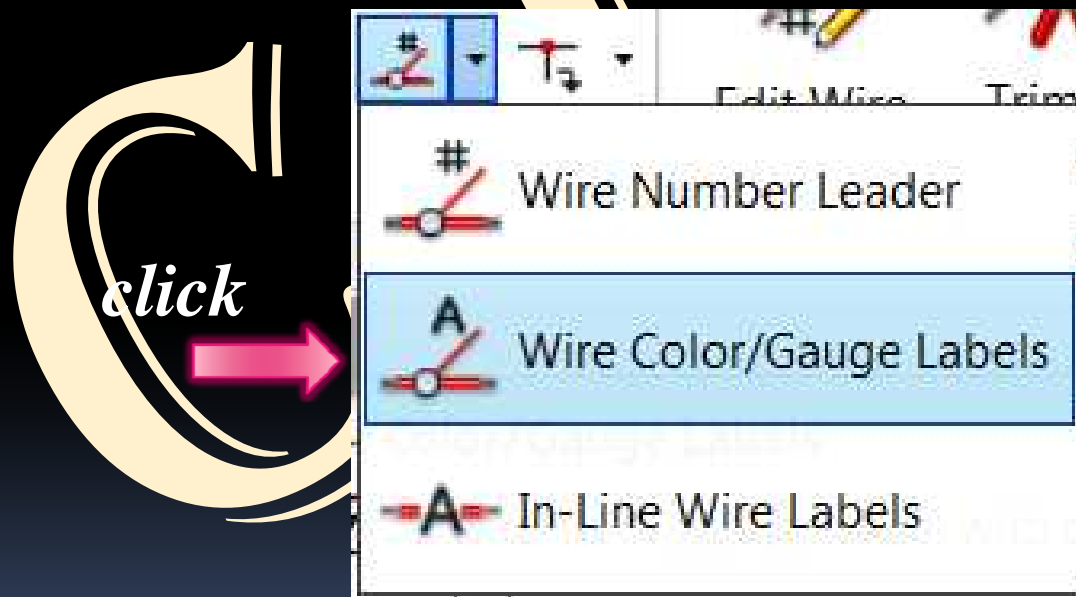
Specificați punctul pentru punctul final al adnotării!



The background features large, stylized yellow letters 'C', 'L', and 'P' with a white outline and a slight shadow effect. The 'C' is on the left, the 'L' is in the center, and the 'P' is on the right. A blue horizontal bar with a white border is positioned across the middle of the letters.

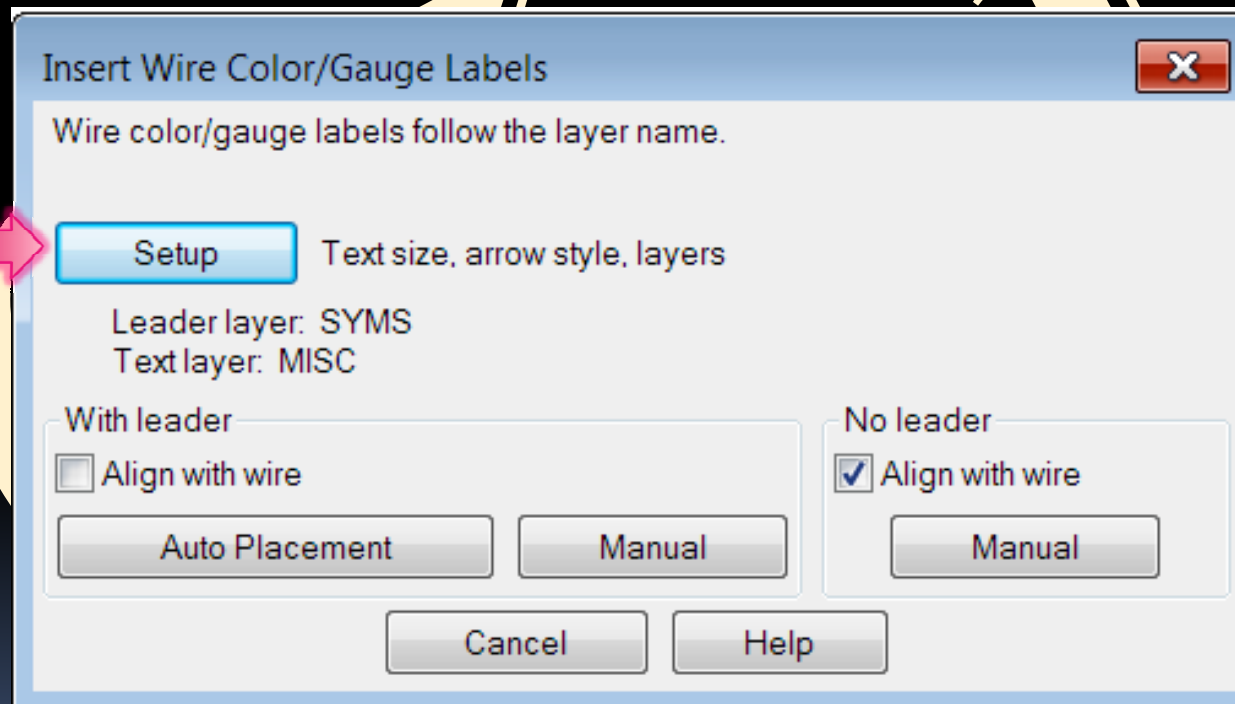
2. Wire Color/Gauge Labels

Afişul Wire Color/Gauge Labels, panel Insert Wires/Wire Numbers, Ribbon Schematic



Caseta de dialog **Insert Wire Color/Gauge Labels**

click



Insert Wire Color/Gauge Labels

Wire color/gauge labels follow the layer name.

Setup Text size, arrow style, layers

Leader layer: SYMS
Text layer: MISC

With leader

Align with wire

Auto Placement Manual

No leader

Align with wire

Manual

Cancel Help



Caseta de dialog Wire label color/gauge setup

Wire label color/gauge setup

Wire Line Layers - Color/Gauge Label Mapping

File: C:\Users\Viorel\A...\Support\user\default.wdw

Wire Layer Name	Color/Gauge label
WHT_12_THHN	#12AWG WHT
RED_14_THHN	#14AWG RED
BLK_12_THHN	#12AWG BLK
YEL_14_THHN	#14AWG YEL
TAN_20AWG	
TAN_18AWG	
TAN_16AWG	

To add/modify the default color/gauge text string, select the layer name from the list above.

To add new wire layer names, select Wires > Create/Edit Wire Type

Leader

0.180 Text Size

0.180 Arrow Size

0.090 Leader/Gap size

Text size follows DIMTXT

Arrow size follows DIMASZ

Arrow Type

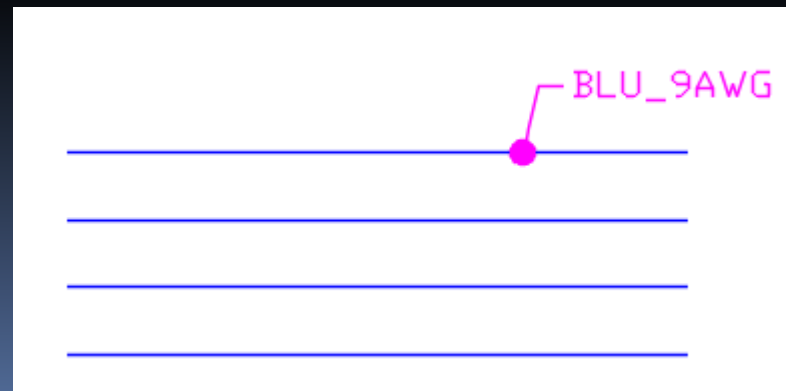
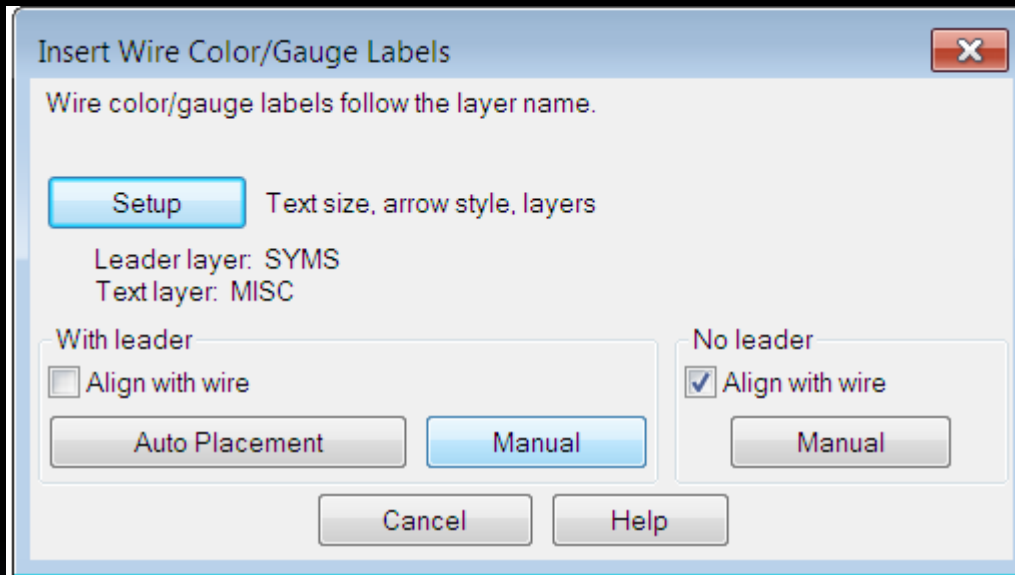
- Closed Filled
- Closed Blank
- Closed
- Dot**
- Architectural Tick
- Oblique

Leader layer SYMS

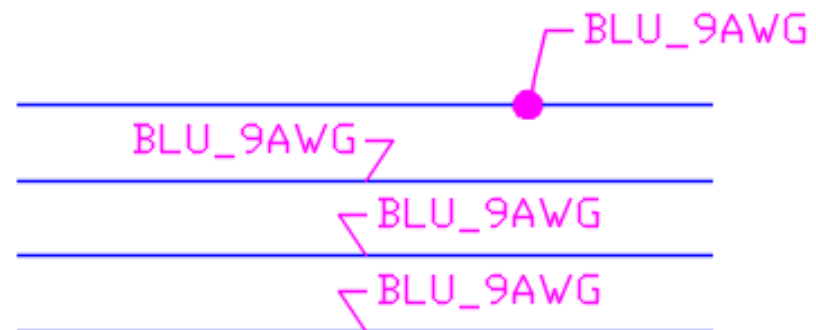
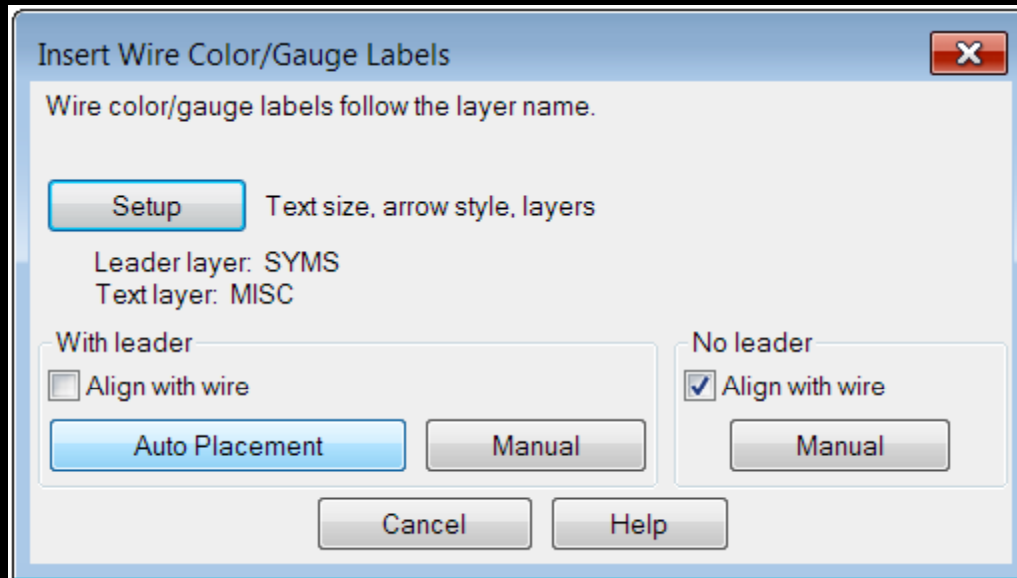
Text layer MISC

OK Cancel

Click **Manual**-urmați instrucțiunile!



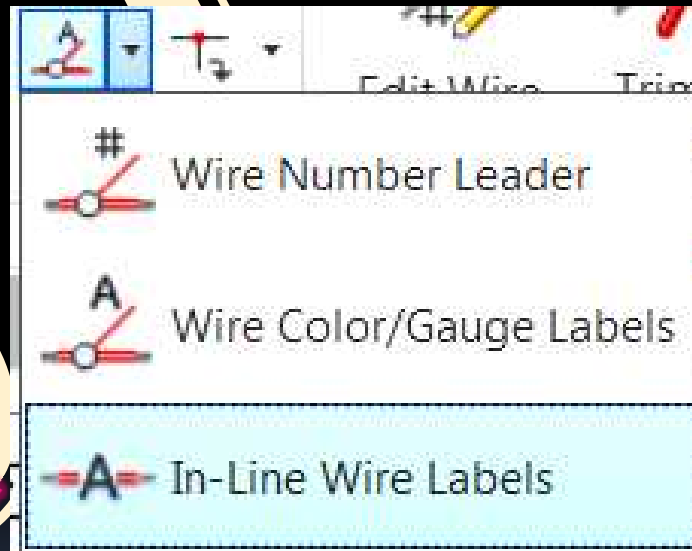
Click **Auto Placement**-urmați instrucțiunile!



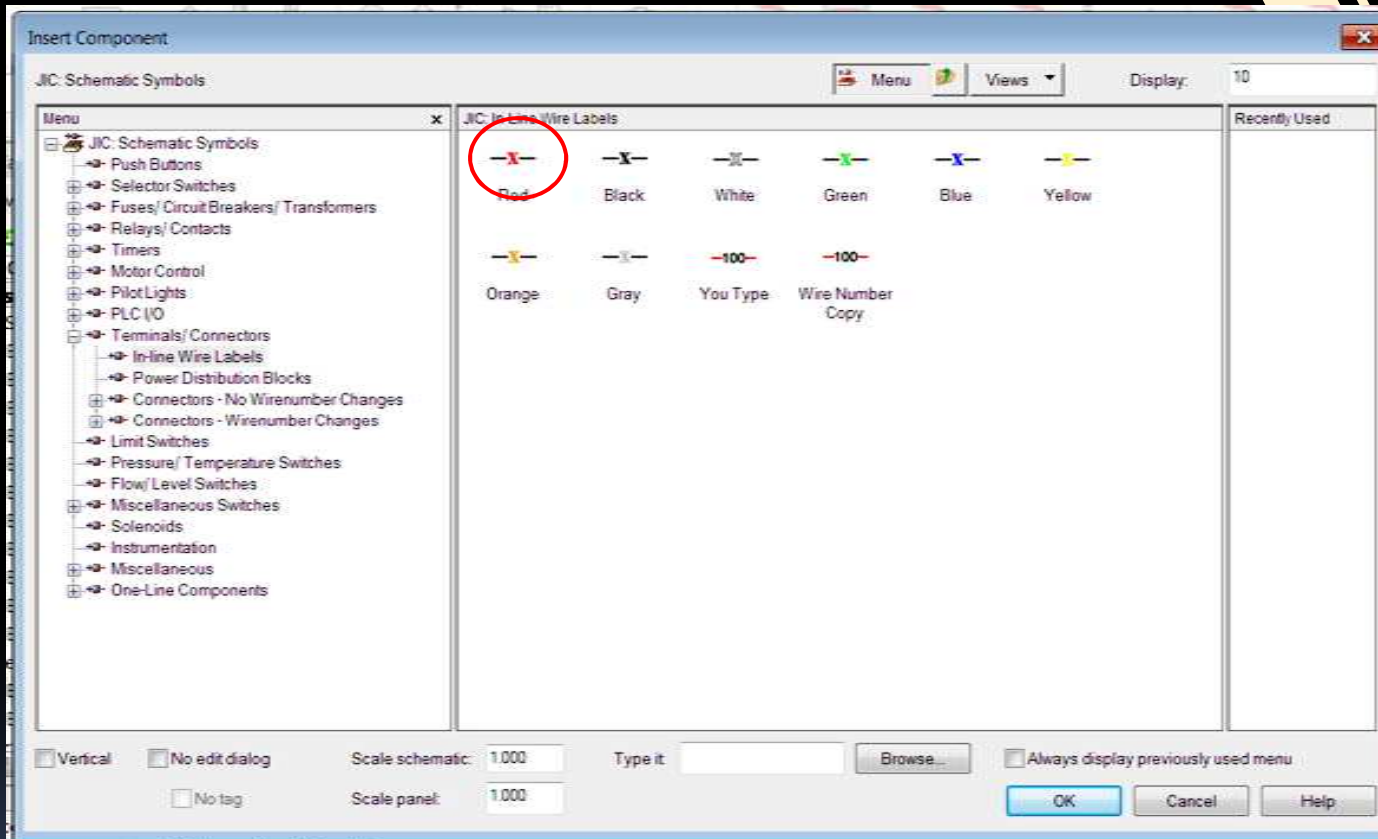
The image features a black background with a blue gradient at the bottom. In the center, the letters 'C', 'V', and 'P' are rendered in a stylized, yellow, outlined font. A semi-transparent blue rectangular box is overlaid on the letters, containing the text '3. In-Line Wire Labels' in a black, italicized serif font.

3. In-Line Wire Labels

Afişul In-Line Wire Labels, panel Insert Wires/Wire Numbers, Ribbon Schematic



Caseta de dialog Insert Component



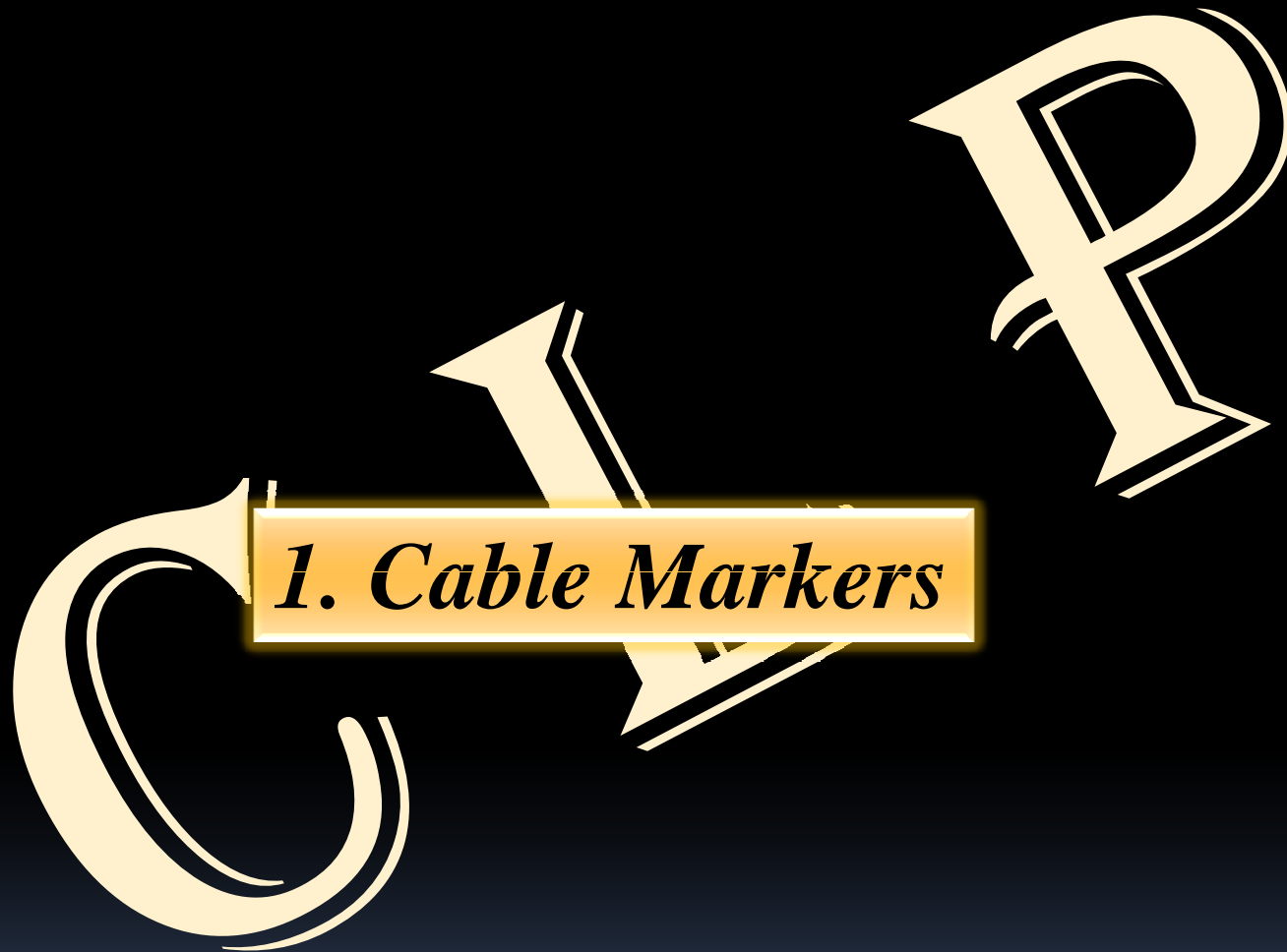
— RED —



VII. Marcarea firelor

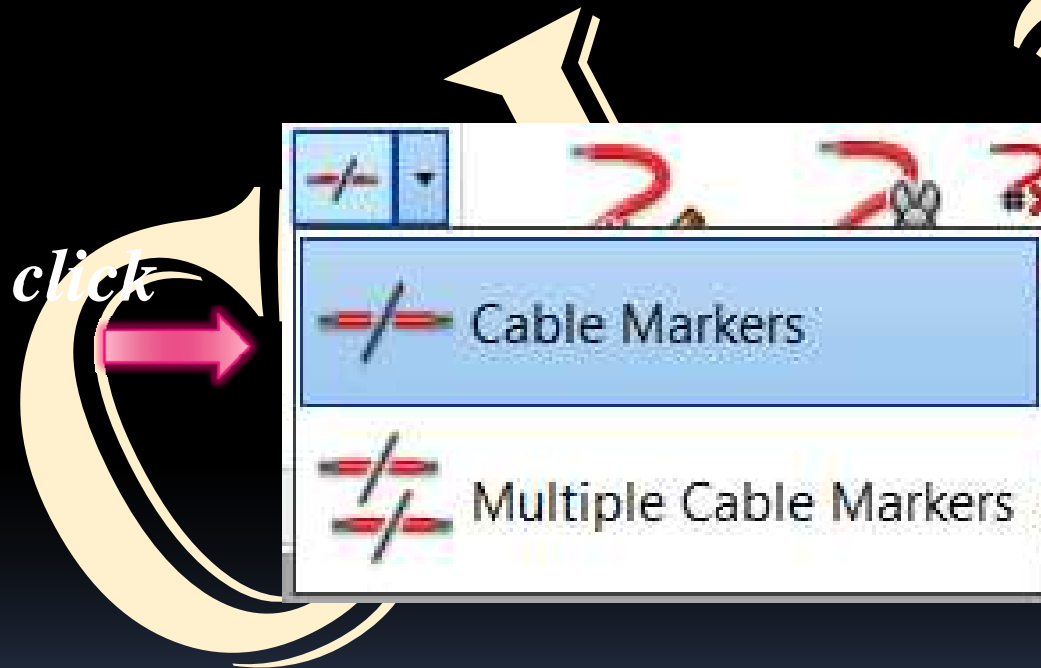
Marcajele sunt utilizate pentru a marca firele în scopul clasificării lor.

C L P

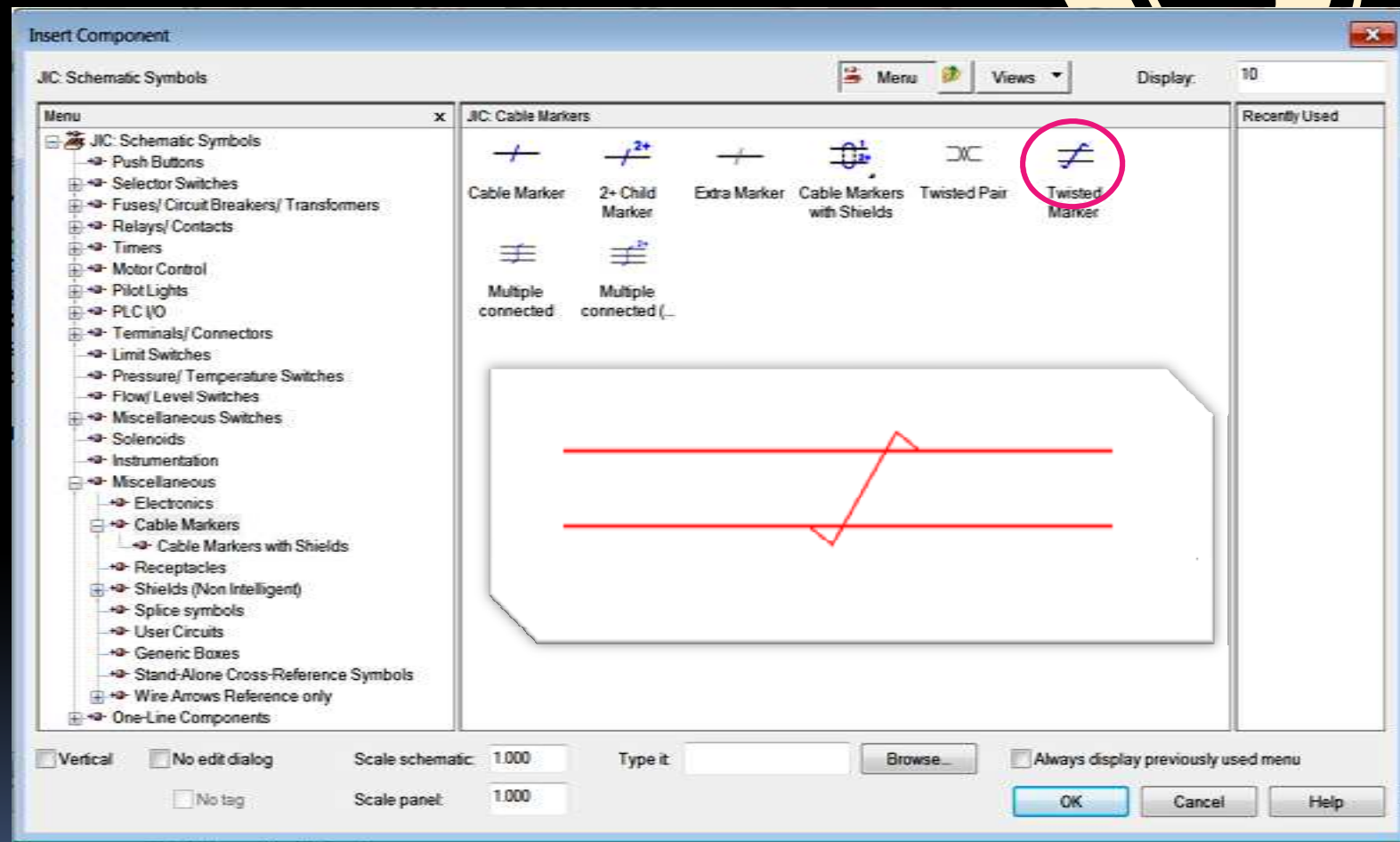
The image features the word "CABLE" in a large, stylized, yellow serif font with a white outline. The letters are arranged in a slightly curved path. A yellow rectangular banner with a white border is positioned horizontally across the middle of the word, overlapping the letters 'A', 'B', and 'L'.

1. Cable Markers

Afişul Cable Markers, panel Insert Wires/Wire Numbers, Ribbon Schematic



Caseta de dialog Insert Component



Caseta de dialog Insert/Edit Marker Cable (Parent wire)

Insert / Edit Cable Marker (Parent wire)

Cable Tag
CBL1 fixed
Use PLC Address
Use End Locations
Tags: Used so far
External List

Wire Color/ID
ID Generic
Drawing Project

Catalog Data
Manufacturer
Catalog
Assembly
Item Count
Next>>
Lookup Previous
Drawing Project
Multiple Catalog
Catalog Check

Description
Line 1
Line 2
Line 3
List Drawing Project Defaults Pick

Child conductor references
 Component override Setup
Cross Reference

Installation code

Drawing Project

Location code

Drawing Project
Show/Edit Miscellaneous

OK Cancel Help

Caseta de dialog **Insert Some Child Components**



Caseta de dialog **Insert/Edit Marker Cable 2nd+wire of cable**

Insert / Edit Cable Marker (2nd+ wire of cable)

Component Tag
Tag:
Drawing Project
Parent/Sibling

Description
Line 1: Pick
Line 2:
Line 3:

Wire Color/ID
ID: Generic
Used: Drawing Project

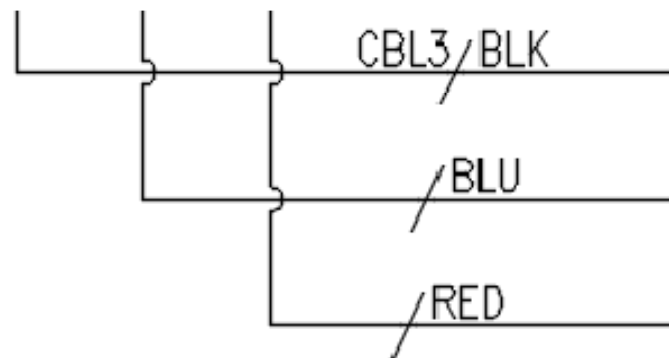
Parent cable marker cross-reference
Cross-reference:

Installation code Drawing Project
Location code Drawing Project

Show/Edit Miscellaneous

OK-Repeat OK Cancel Help

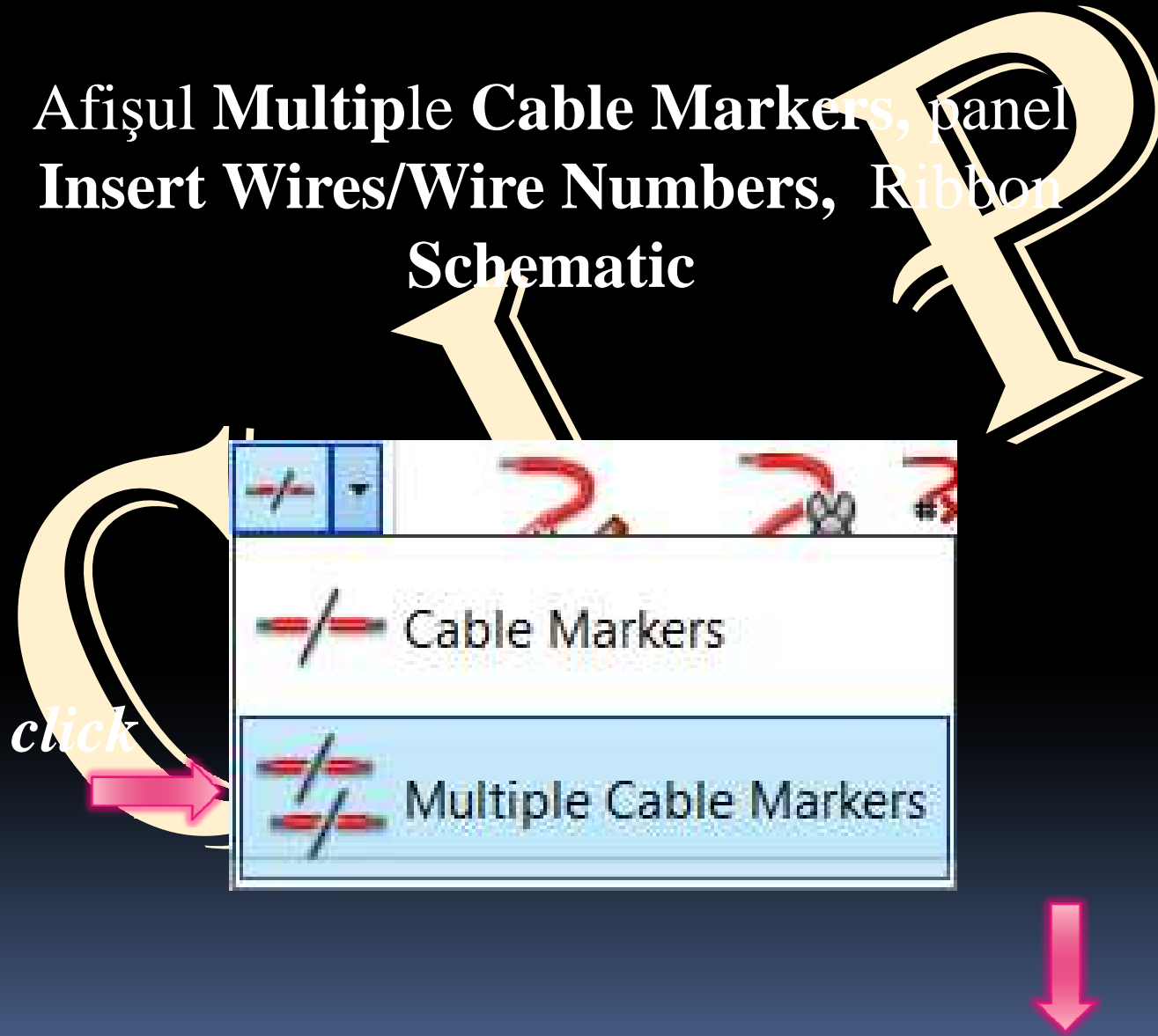
Alegeți firele “CHILD”
și specificați etichetele!



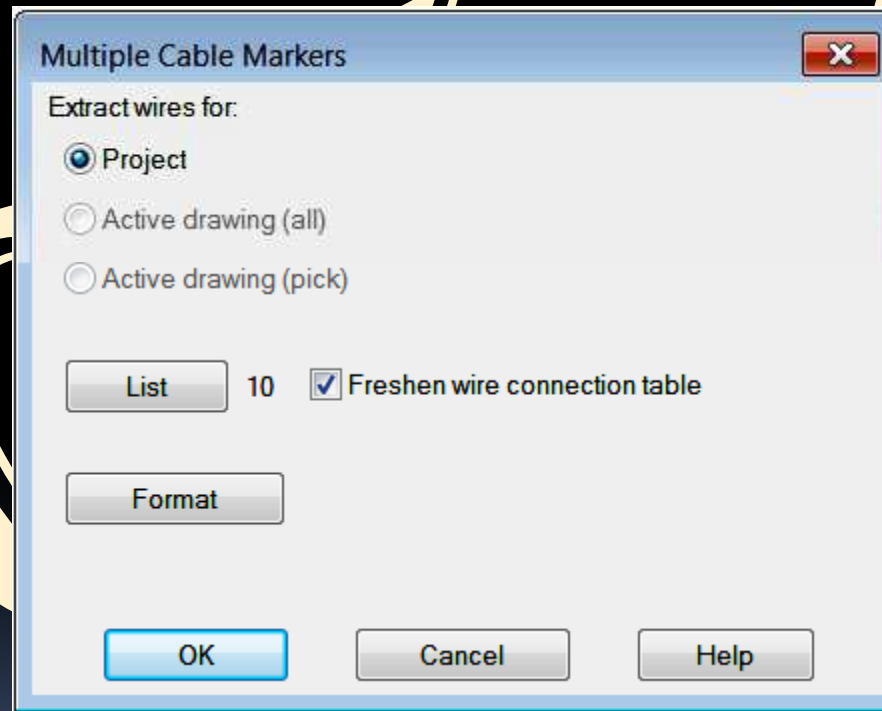
The image features a dark blue gradient background. In the center, the letters 'G', 'A', and 'P' are rendered in a large, stylized, yellow font with a white outline and a slight shadow. The 'G' is on the left, the 'A' is in the middle, and the 'P' is on the right. A horizontal yellow banner with a white border and a slight shadow is positioned across the middle of the letters, containing the text '2. Multiple Cable Markers' in a black, italicized serif font.

2. Multiple Cable Markers

Afişul Multiple Cable Markers, panel Insert Wires/Wire Numbers, Ribbon Schematic



Caseta de dialog **Multiple Cable Markers**



The background features a dark blue gradient. In the center, there are stylized, overlapping letters in a light yellow/gold color. A large 'C' is on the left, a large 'P' is on the right, and a stylized arrow points from the 'C' towards the 'P'.

VIII. Marcarea intersecției firelor

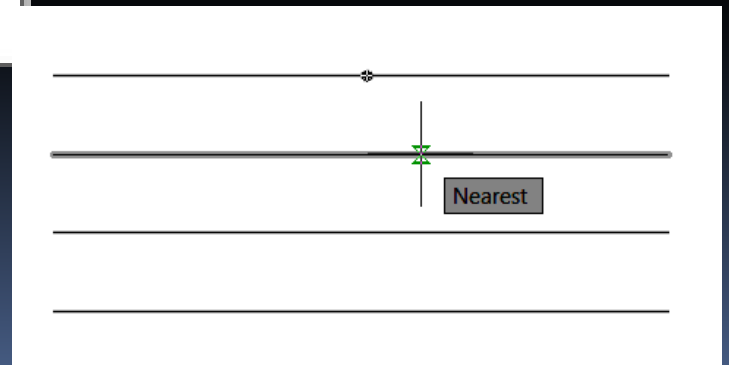
The image features three large, stylized letters: 'C', 'A', and 'P', rendered in a decorative, calligraphic font. The letters are light yellow with a dark outline and are set against a black background. A semi-transparent blue rectangular box is overlaid on the letters, containing the text '1. Insert Dot Tee Markers'.

1. Insert Dot Tee Markers

Afişul Insert Dot Tee Markers, panel Insert Wires/Wire Numbers, Ribbon Schematic



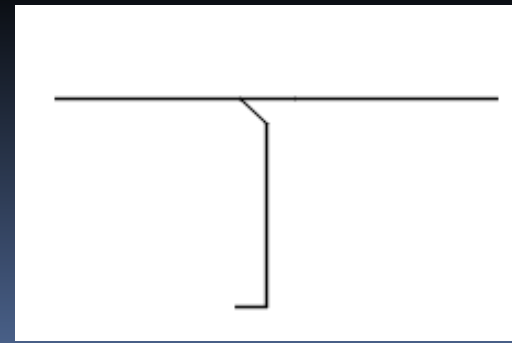
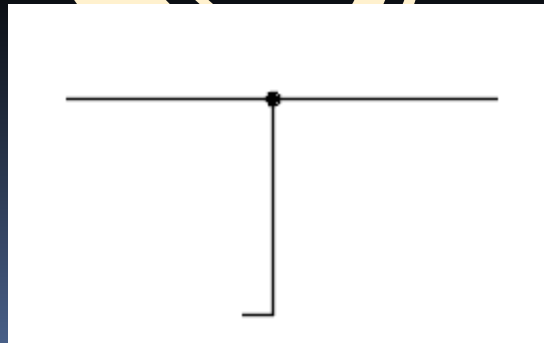
click





2. Insert Angle Tee Markers

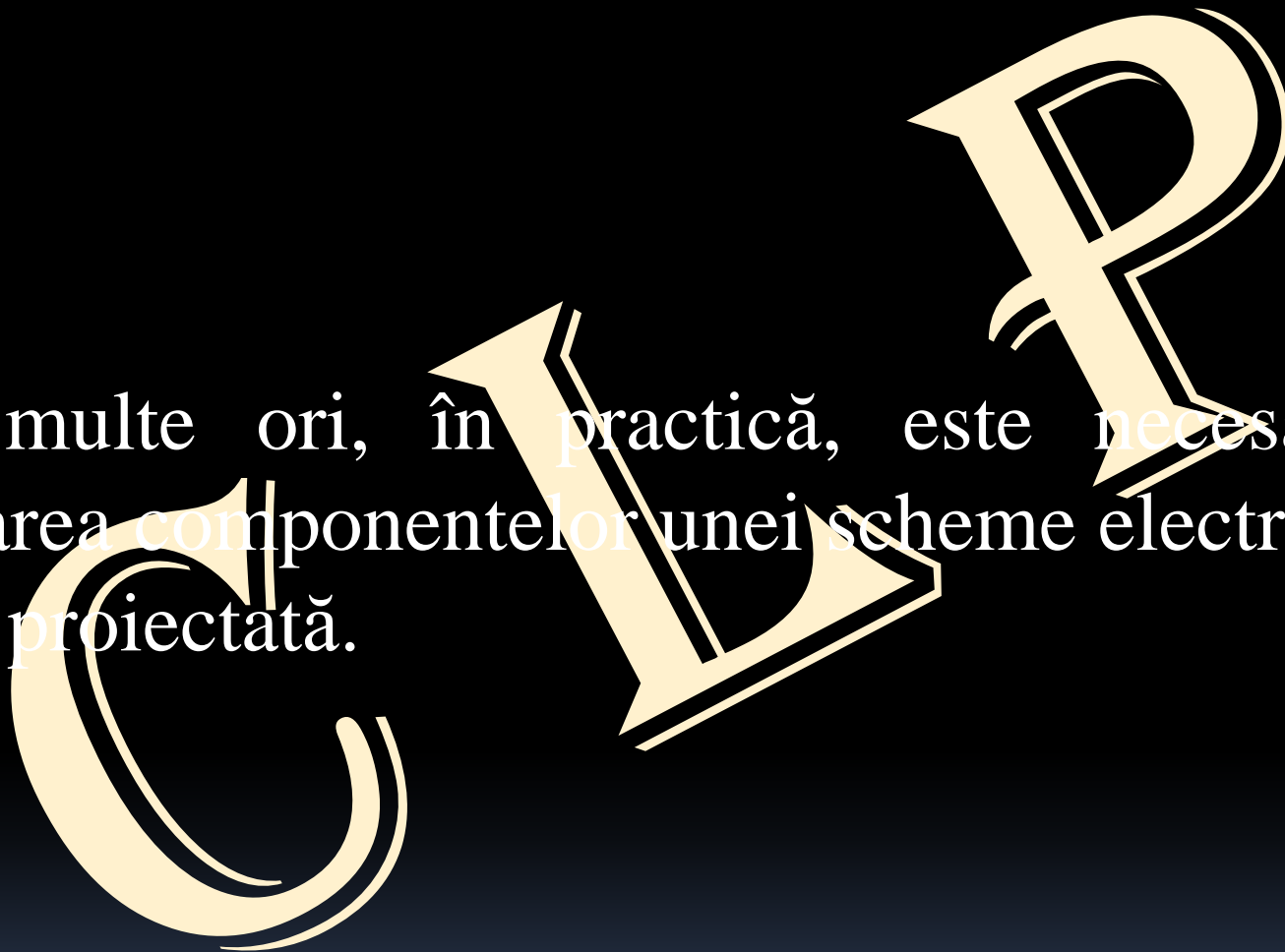
Afişul Insert Angle Tee Markers panel Insert Wires/Wire Numbers, Ribbon Schematic





IX. Editarea componentelor electrice

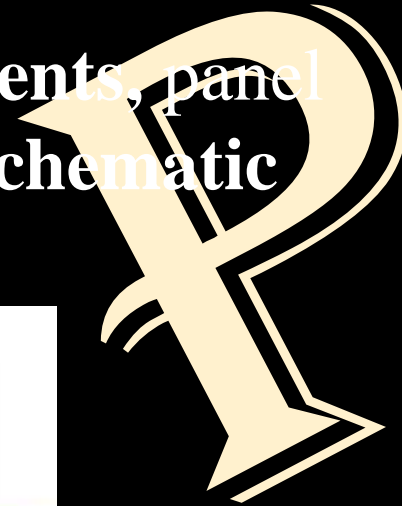
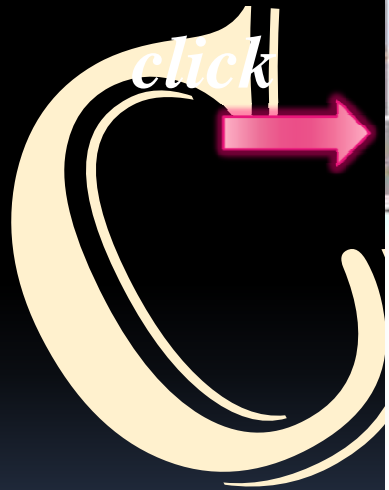
De multe ori, în practică, este necesară editarea componentelor unei scheme electrice deja proiectată.



The background features three large, stylized letters: 'C' on the left, 'V' in the center, and 'P' on the right. They are rendered in a light yellow or gold color with a white outline and a slight shadow, giving them a three-dimensional appearance. The letters are set against a dark blue gradient background that transitions from black at the top to a lighter blue at the bottom.

1. Editarea componentelor


Meniu derulant **Edit Components**, panel **Edit Components**, Ribbon Schematic



Selectați
componentele/cablurile/locațiile!

Command:

Edit Component

 Select component/cable/location box to EDIT:



Caseta de dialog **Insert/Edit Component**

The screenshot shows the 'Insert / Edit Component' dialog box with the following fields and controls:

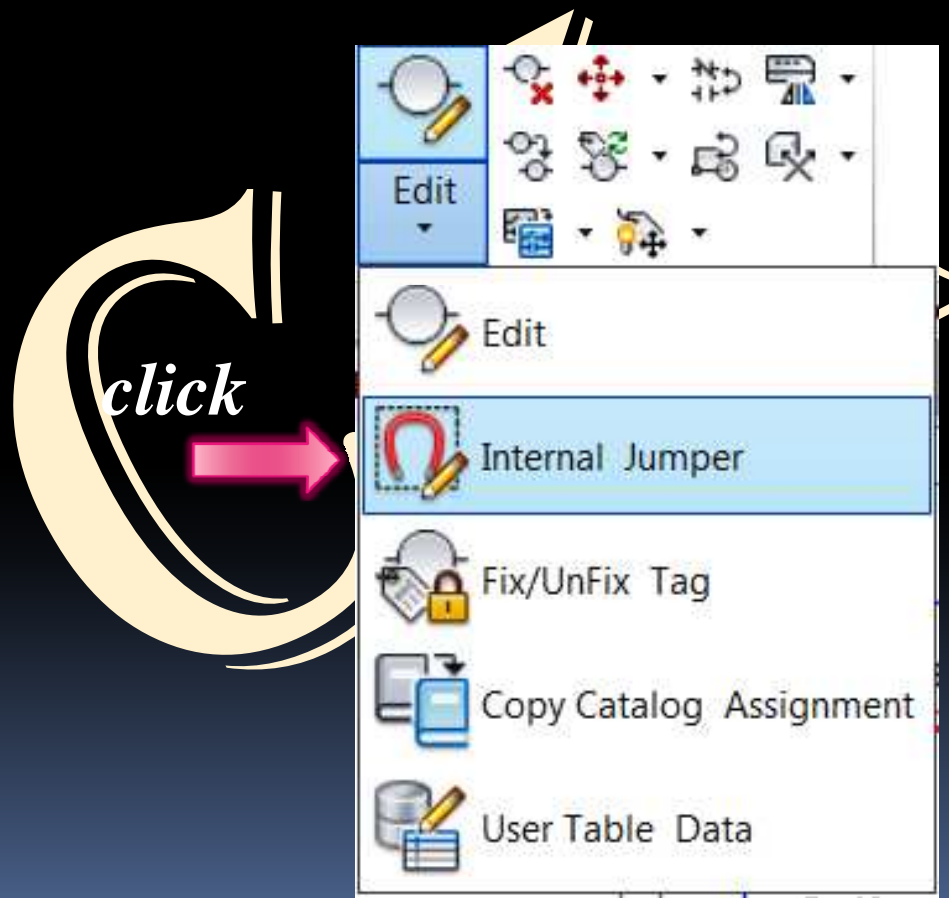
- Component Tag:** FU105, fixed, Use PLC Address, Schematic, Panel, External List, Options buttons.
- Description:** Line 1: 2A, Line 2, Line 3, List buttons: Drawing, Project, Defaults, Pick.
- Cross-Reference:** Component override, Setup, Reference NO, Reference NC, NO/NC Setup buttons.
- Catalog Data:** Manufacturer, Catalog, Assembly, Item, Count, Next>>, Lookup, Previous, Drawing, Project, Multiple Catalog, Catalog Check buttons.
- Installation code:** Installation code field, Drawing, Project buttons.
- Location code:** Location code field, Drawing, Project buttons.
- Ratings:** Rating field, Show All Ratings button.
- Pins:** Pins 1, 2, X with input fields and navigation buttons.
- Buttons:** OK-Repeat, OK, Cancel, Help.

Specificați modificările dorite în caseta de dialog și faceți click pe butonul **OK**!



2. Internal jumper

Afişul Internal Jumper, panel Edit Components, Ribbon Schematic

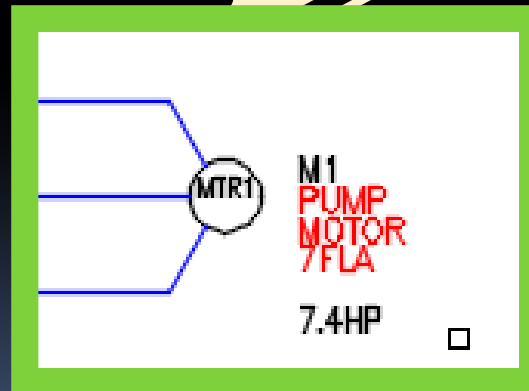


Selectați o componentă pentru editarea jumperilor!

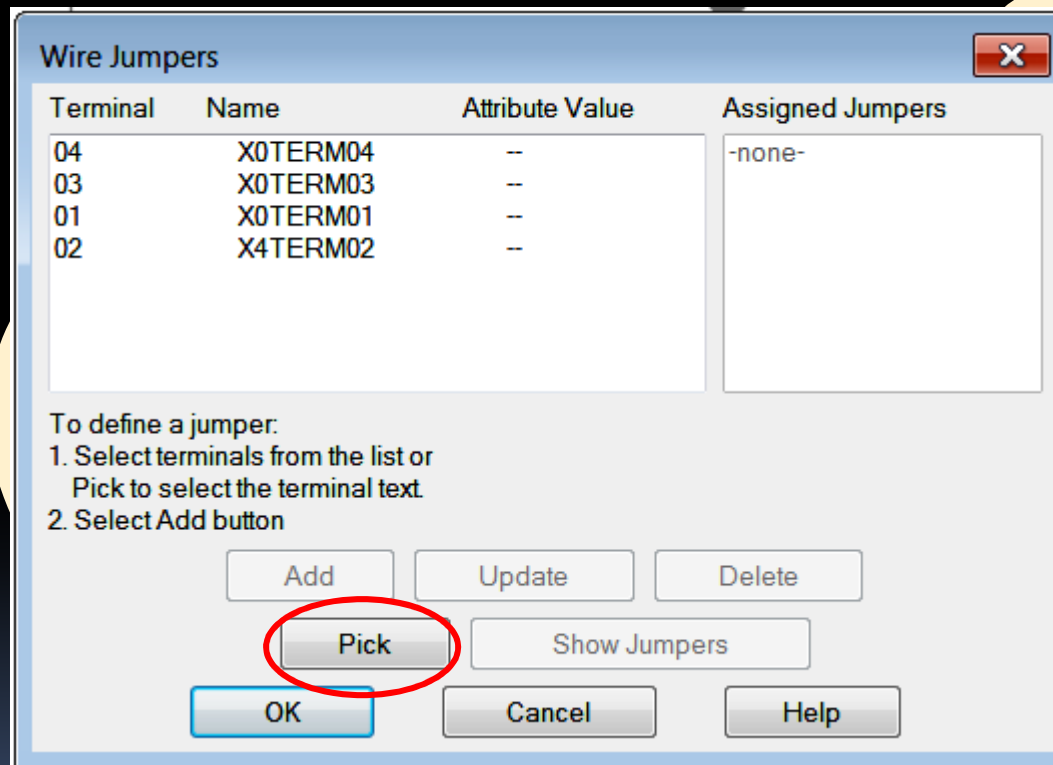
Command:
Internal Wire Jumpers

> -Select component:

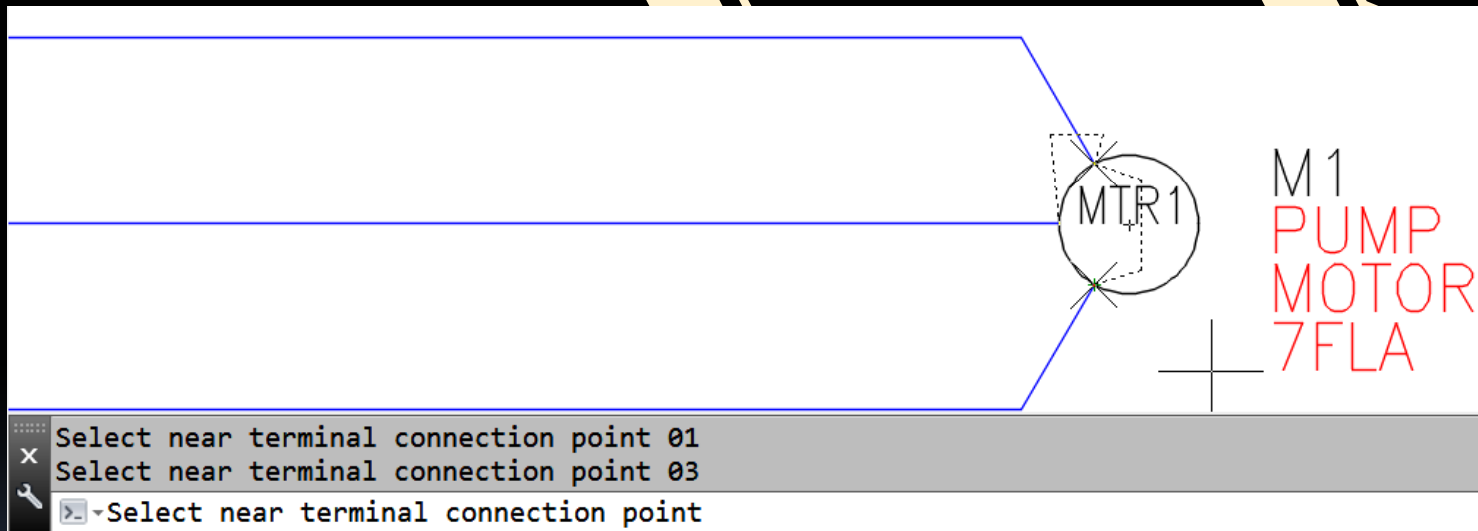
Componenta
selectată



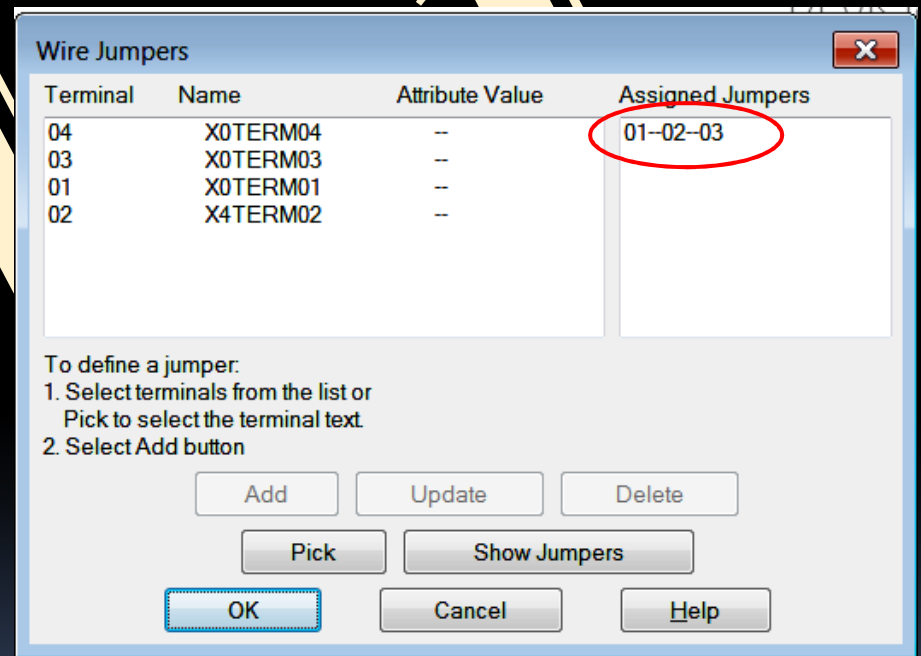
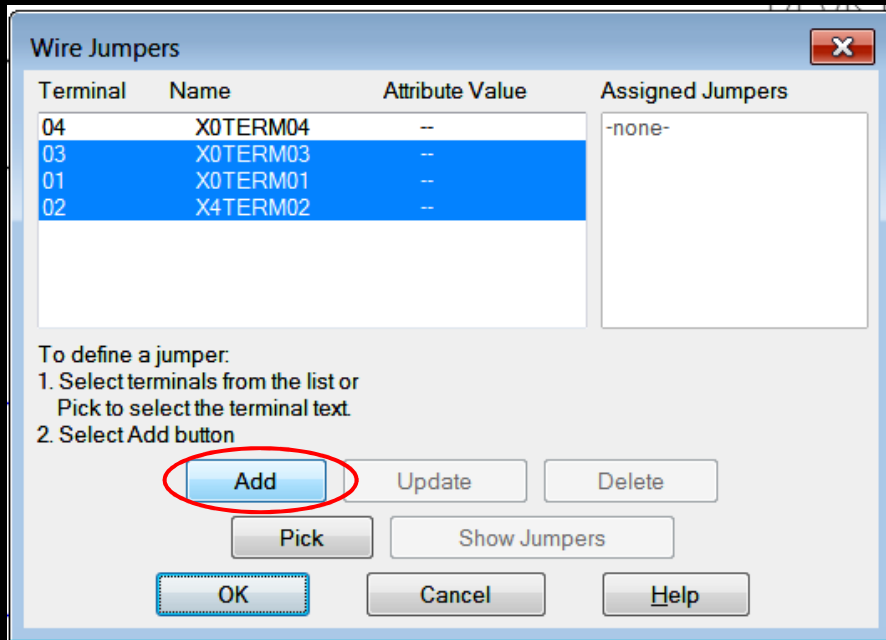
Caseta de dialog Jumper Wire



Selectați punctele jumperului
de pe componentă!



ENTER

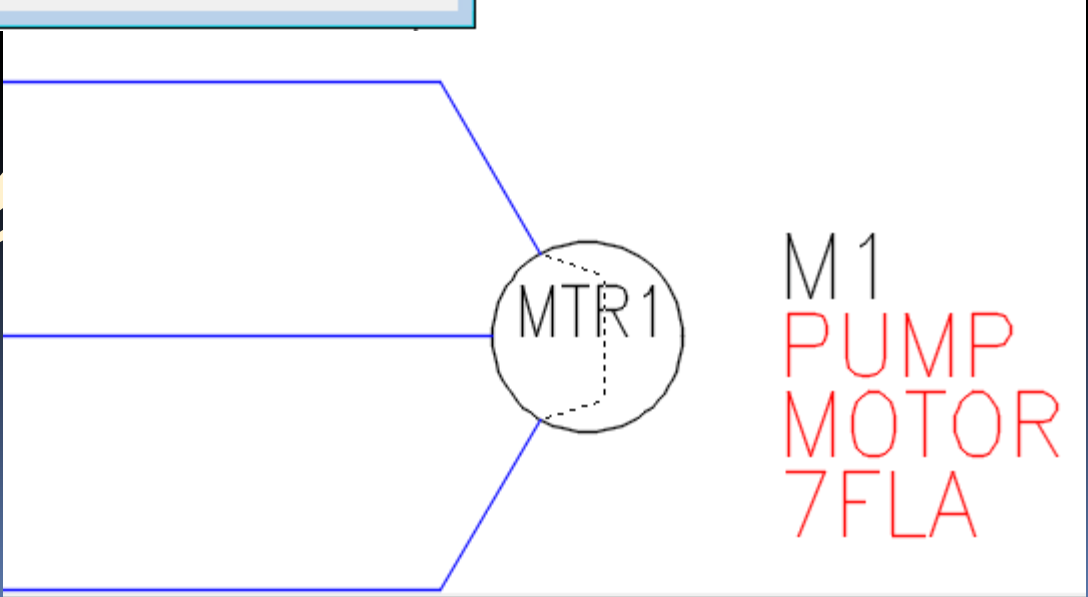


Wire Jumpers

Terminal	Name	Attribute Value	Assigned Jumpers
04	X0TERM04	--	01-02-03
03	X0TERM03	--	
01	X0TERM01	--	
02	X4TERM02	--	

To define a jumper:
1. Select terminals from the list or
Pick to select the terminal text.
2. Select Add button

Buttons: Add, Update, Delete, Pick, Show Jumpers, OK, Cancel, Help



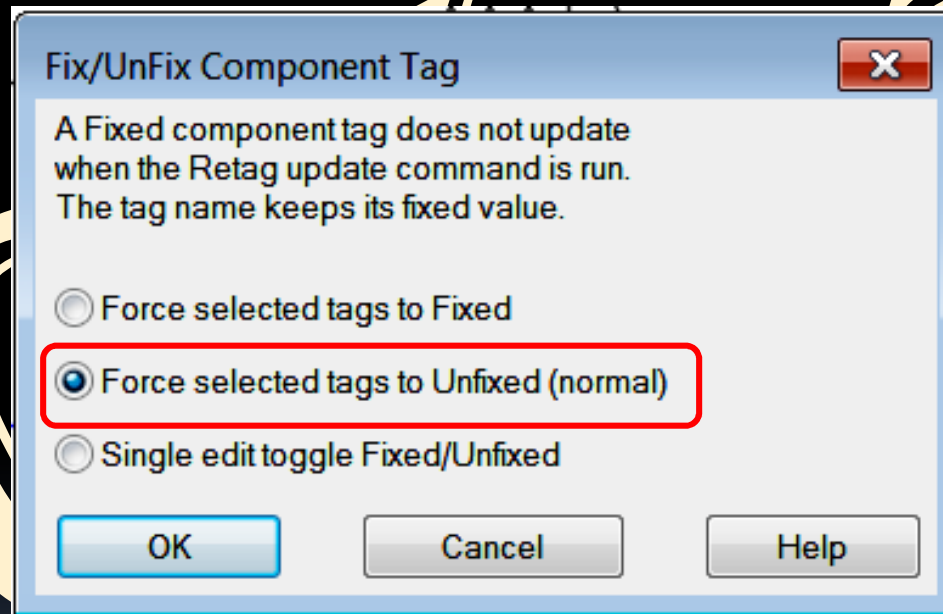


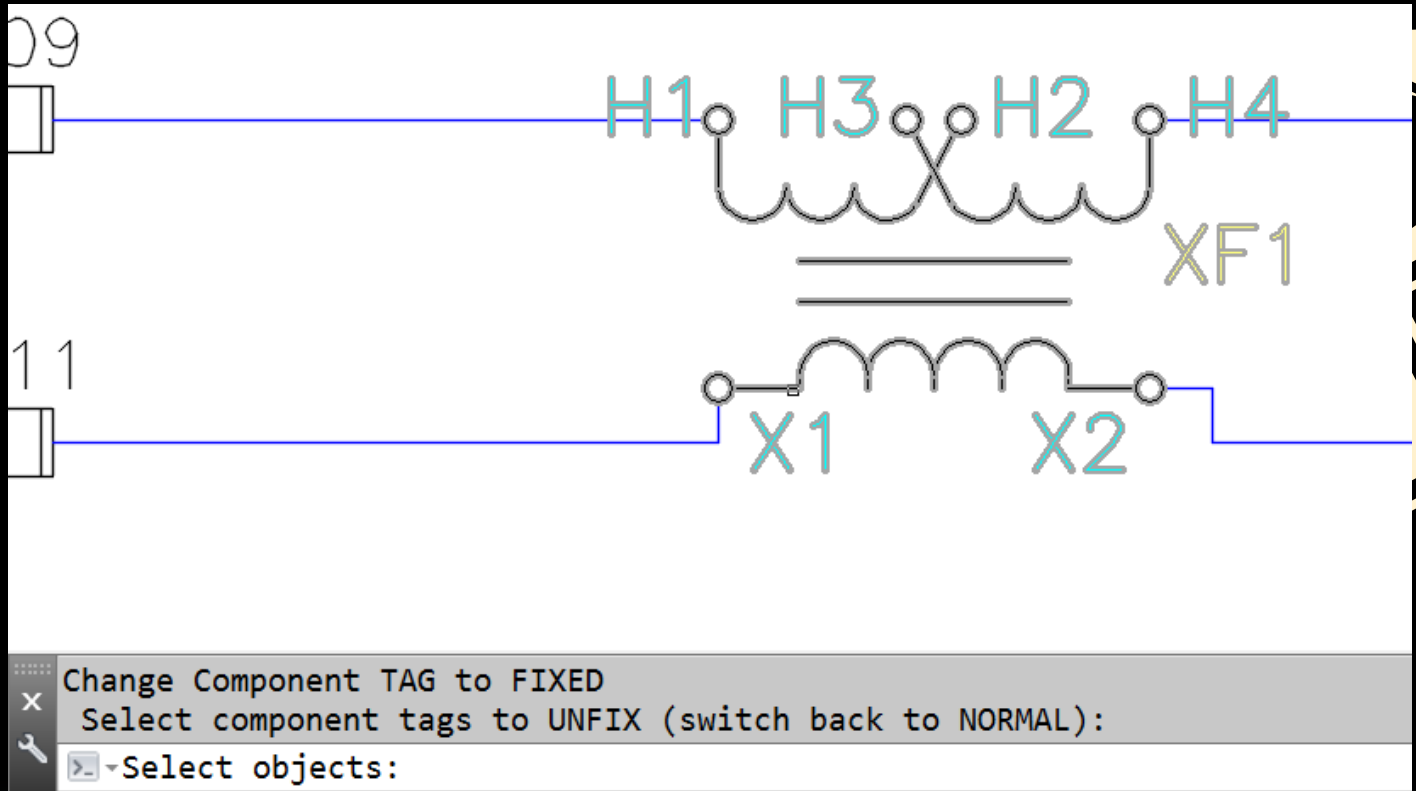
3. Fix/UnFix Tag

Afişul **Fix/UnFix Tag** , panel **Edit Components**, Ribbon **Schematic**



Caseta de dialog Fix/UnFix Tag





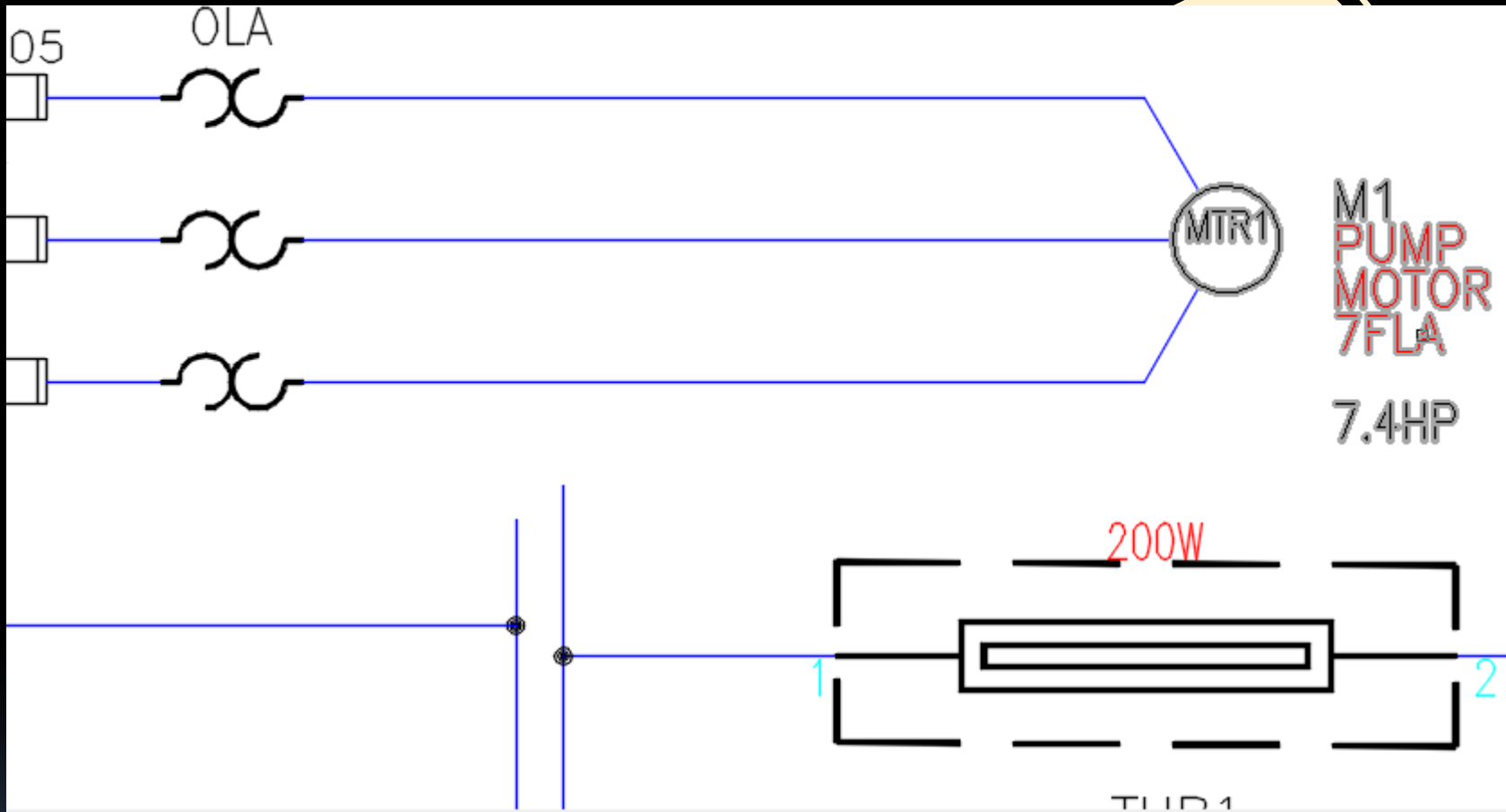
Change Component TAG to FIXED
Select component tags to UNFIX (switch
back to NORMAL):
Select objects: 1 found
Select objects:
Unfixed TAG1F (XF1)

The image features a dark blue gradient background. In the center, there are large, stylized yellow letters 'C', 'L', and 'P' arranged in a curve. A horizontal blue banner with a white border is overlaid across the middle of the letters. The banner contains the text '4. Copy catalog assignment' in a black, italicized serif font.

4. Copy catalog assignment

Afişul Copy Catalog Assignment, panel Edit Components, Ribbon Schematic





Command:
Copy catalog part number assignment
Select "Master" component:

Caseta de dialog Copy Catalog Assignment

click



Copy Catalog Assignment

Manufacturer

Catalog

Assembly

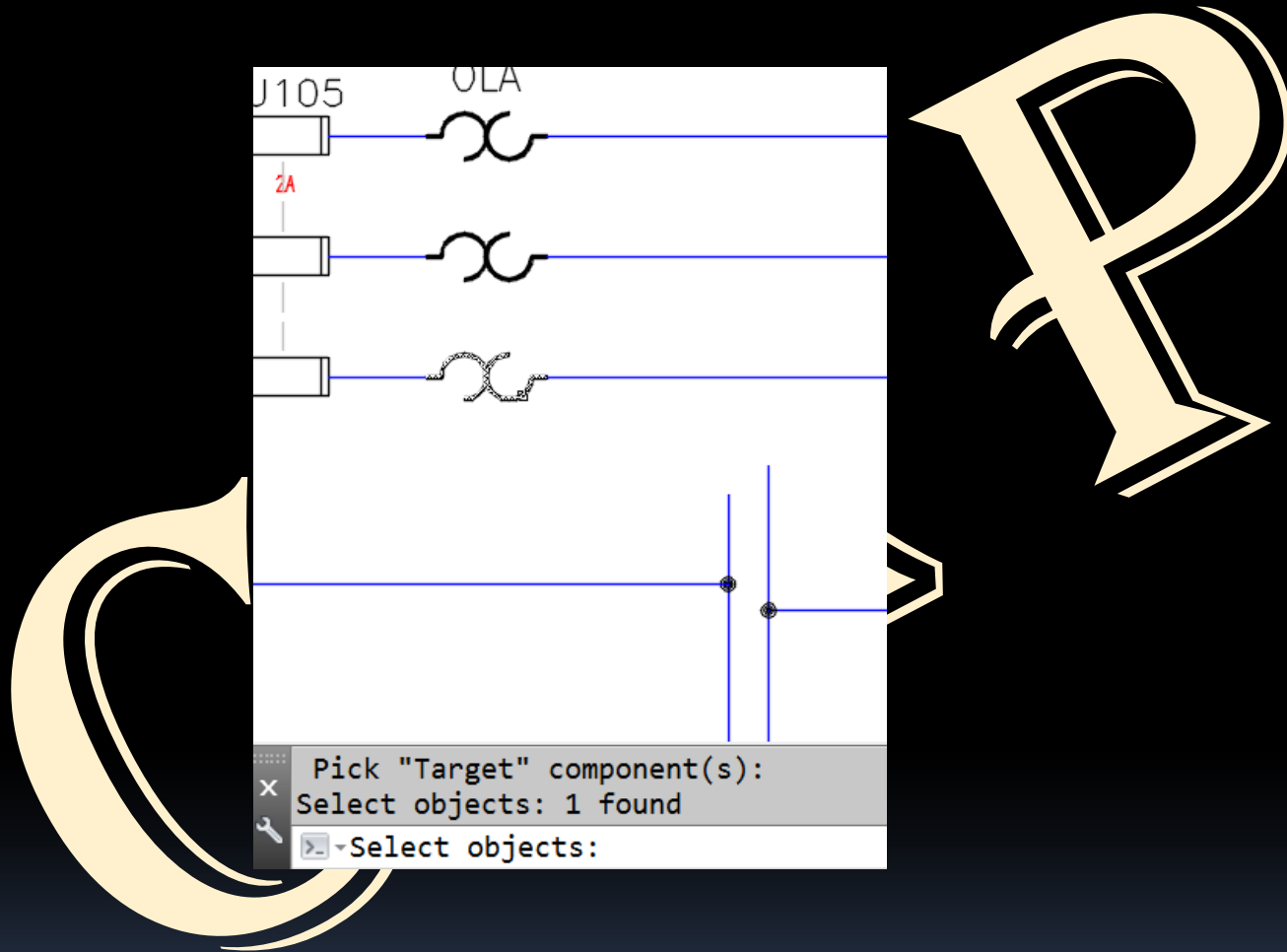
Step 1: Select part number information.
Select OK

Step 2: Select devices to copy to.

Note: child or related devices
are not automatically updated.
They must be included in the selection.

Faceți modificările dorite,
apoi click pe butonul **OK!**



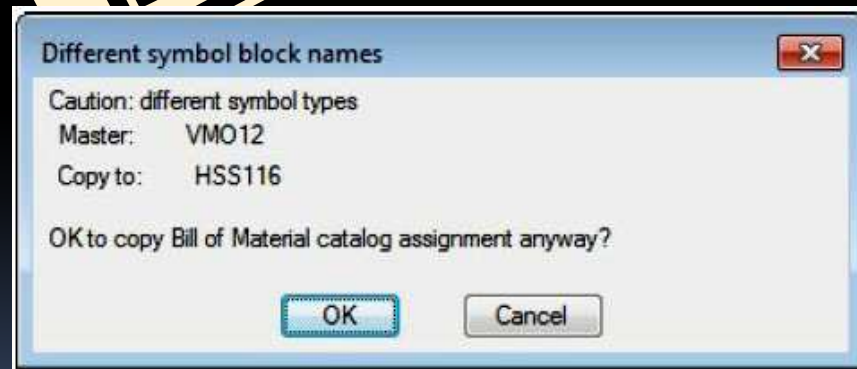


Selectați componenta țintă!

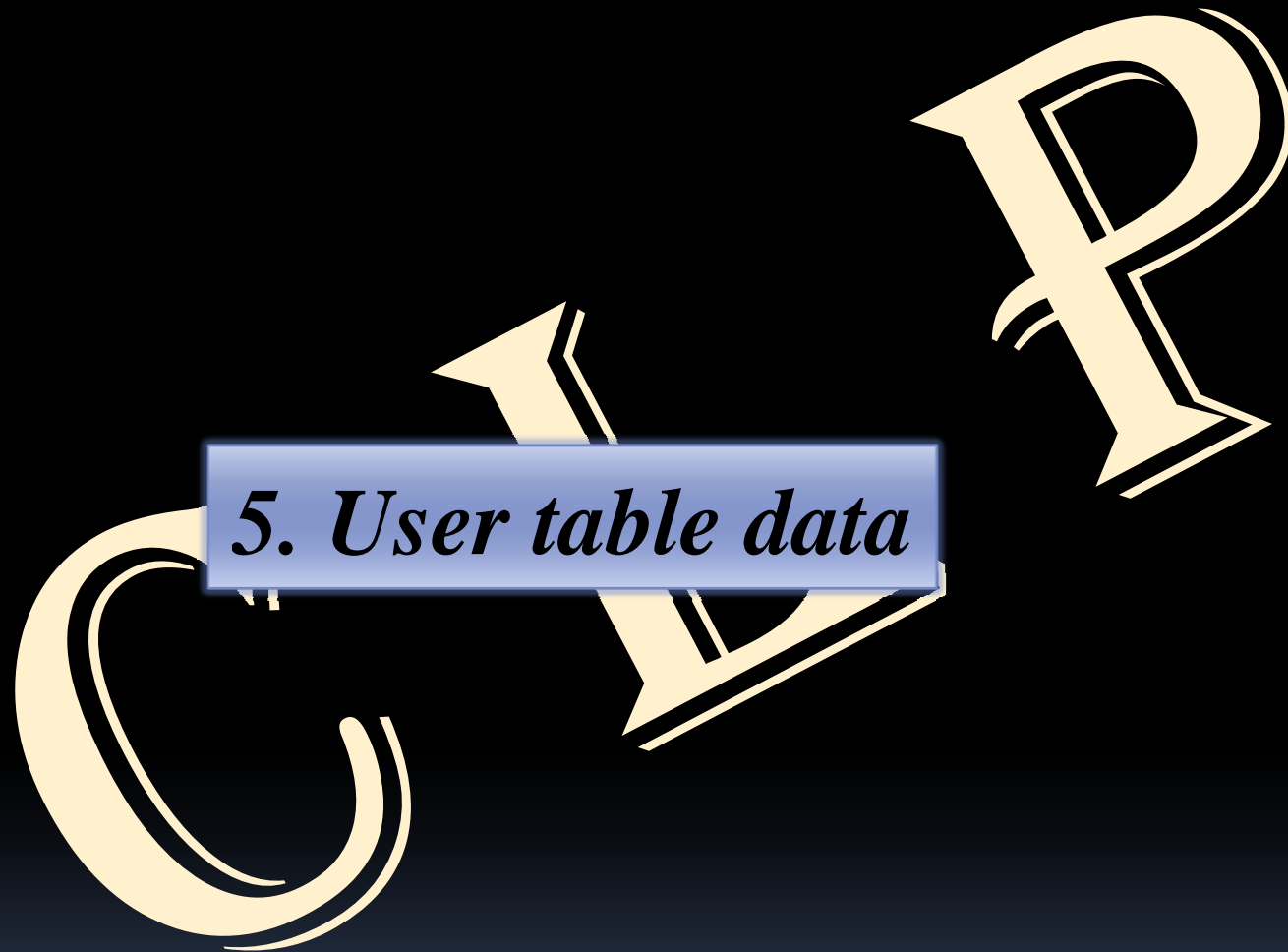
Caseta de dialog
Update Related Components



Caseta de dialog
Different symbol block names



sau



5. User table data

Afişul User Table Data , panel Edit Components, Ribbon Schematic



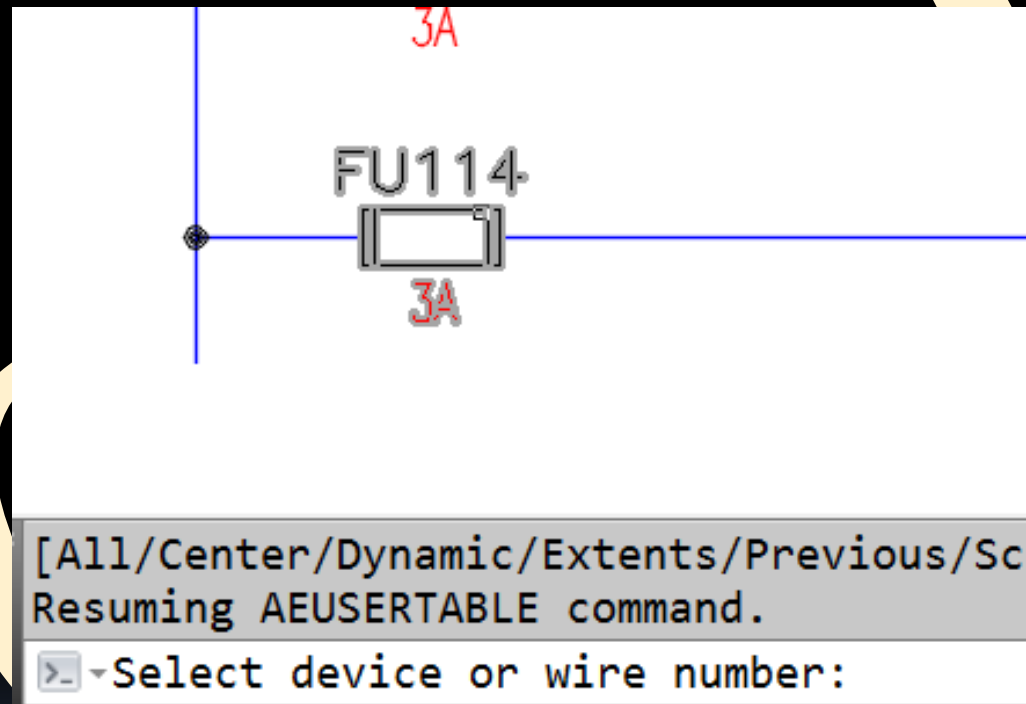
click



The screenshot shows a software ribbon menu. The top section is a ribbon with a 'User Table Data' tab selected. Below the ribbon is a dropdown menu with the following items:

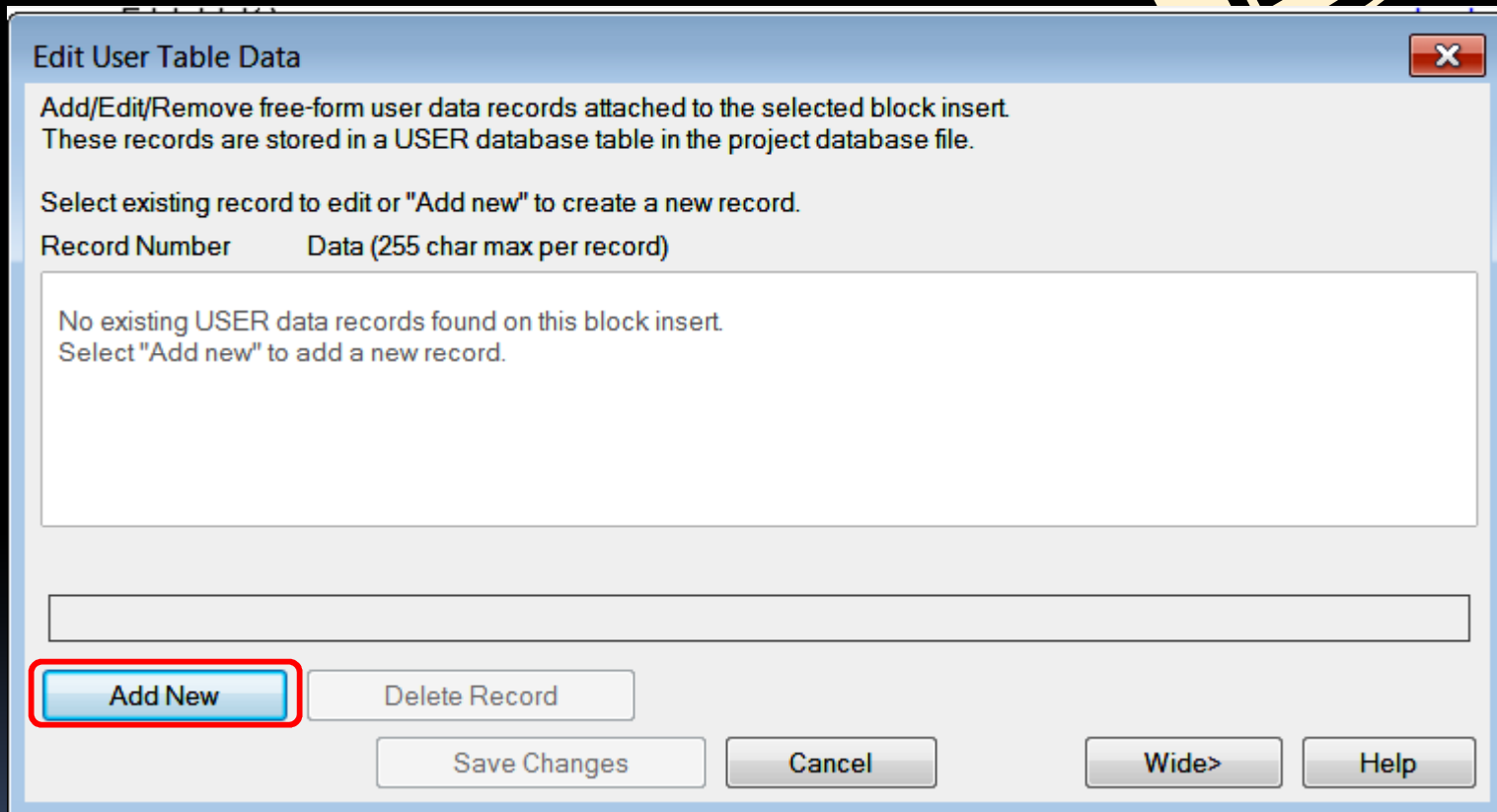
- Edit
- Internal Jumper
- Fix/UnFix Tag
- Copy Catalog Assignment
- User Table Data





Selectați componenta!

Caseta de dialog Edit User Table Data



Edit User Table Data

Add/Edit/Remove free-form user data records attached to the selected block insert.
These records are stored in a USER database table in the project database file.

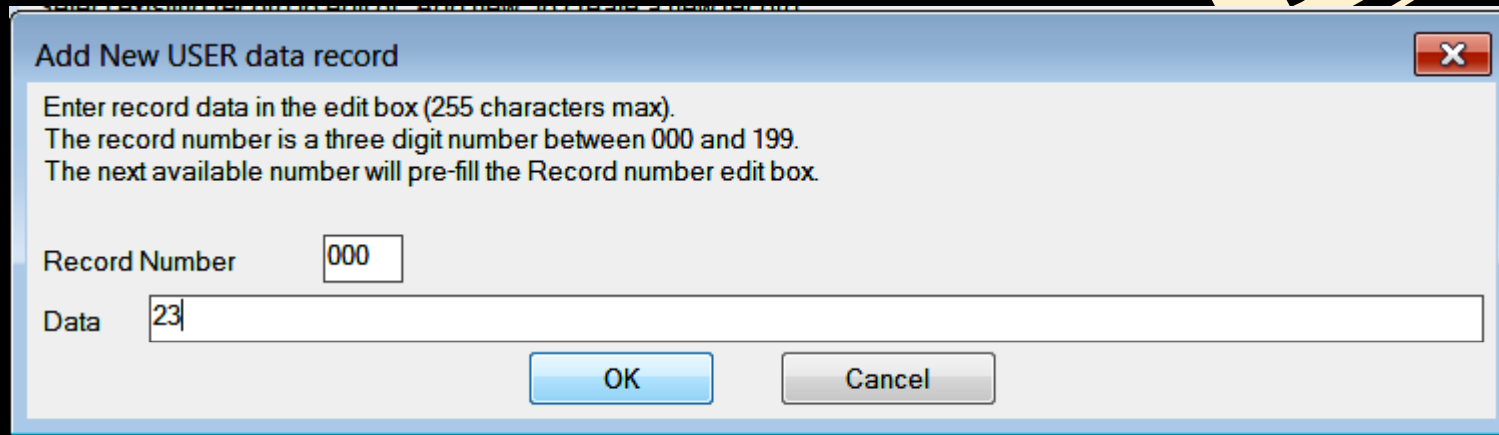
Select existing record to edit or "Add new" to create a new record.

Record Number	Data (255 char max per record)
No existing USER data records found on this block insert. Select "Add new" to add a new record.	

Buttons: Add New, Delete Record, Save Changes, Cancel, Wide>, Help



Introduceți datele dorite în câmp!



The image shows a Windows-style dialog box titled "Add New USER data record". The dialog box has a blue title bar with a close button (X) in the top right corner. The main area contains the following text: "Enter record data in the edit box (255 characters max). The record number is a three digit number between 000 and 199. The next available number will pre-fill the Record number edit box." Below this text are two input fields. The first is labeled "Record Number" and contains the value "000". The second is labeled "Data" and contains the value "23". At the bottom of the dialog box are two buttons: "OK" and "Cancel".

Click **Save Changes** din caseta de dialog **Edit User Table Data!**



X. Modificarea poziției componentelor

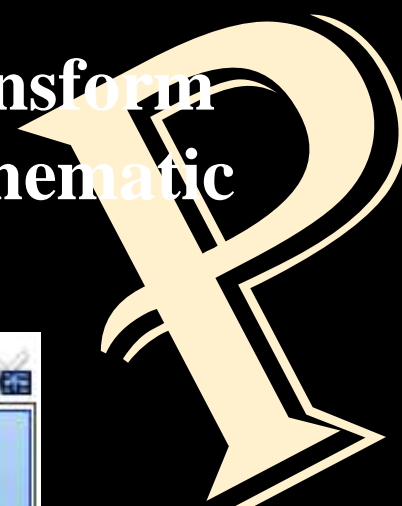
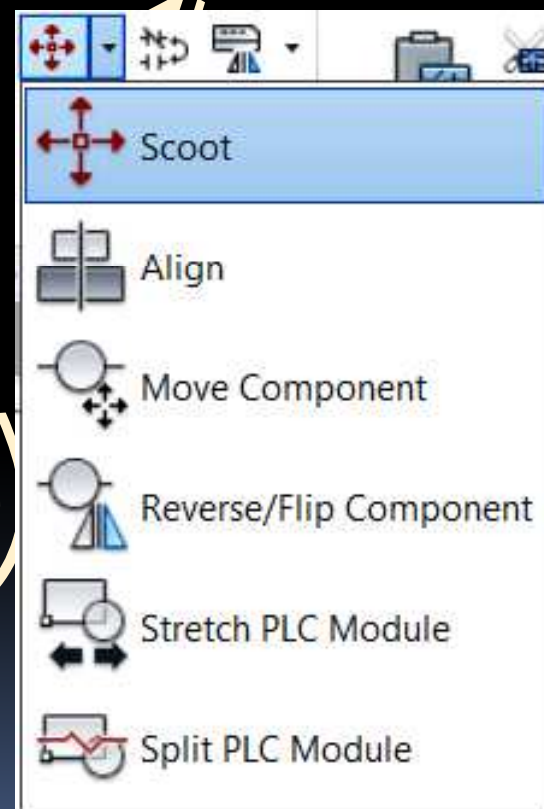


C *H* *P*

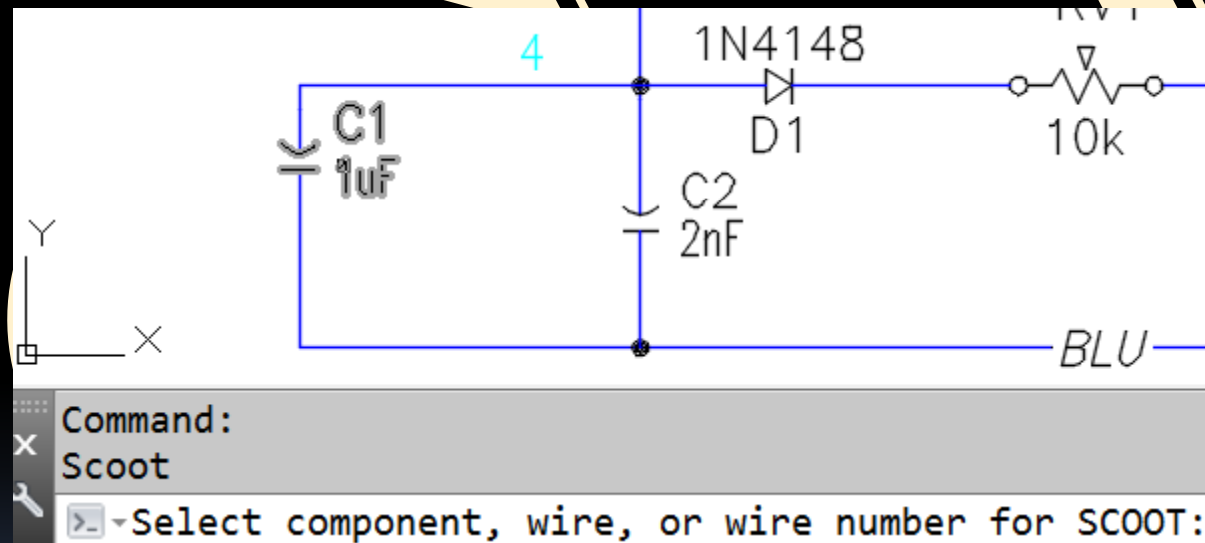
1.Scoot

The image features three large, stylized, yellow letters with black outlines: 'C', 'H', and 'P'. The 'H' is positioned in the center and is partially obscured by a pink rectangular box with a white border. Inside this box, the text '1.Scoot' is written in a black, italicized serif font. The background is a dark blue gradient that transitions from black at the top to a lighter blue at the bottom.

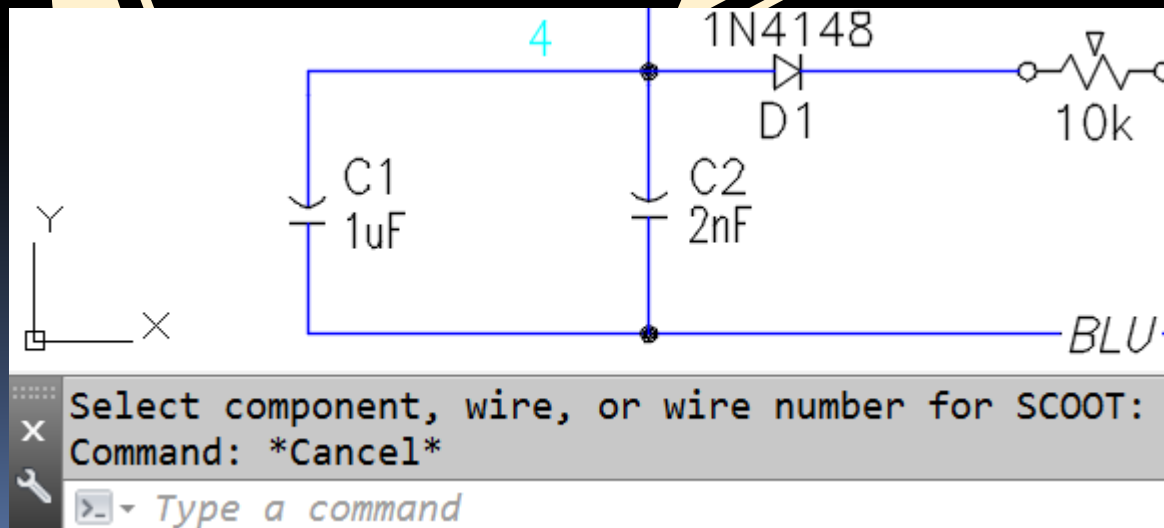
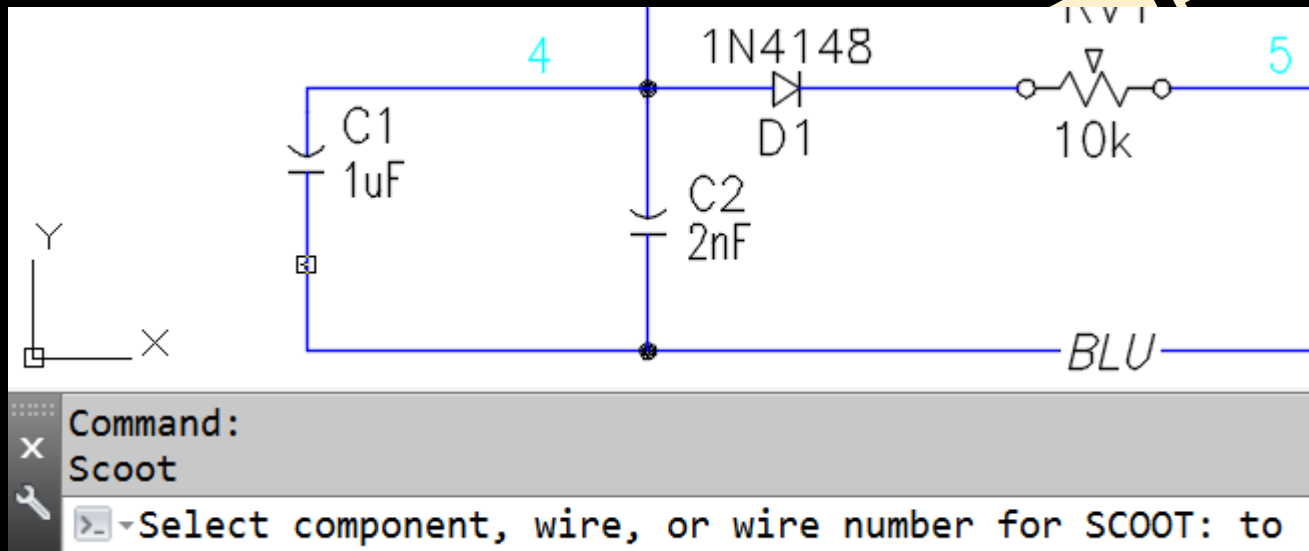
Afișul Scoot , panel Transform Component, Ribbon Schematic

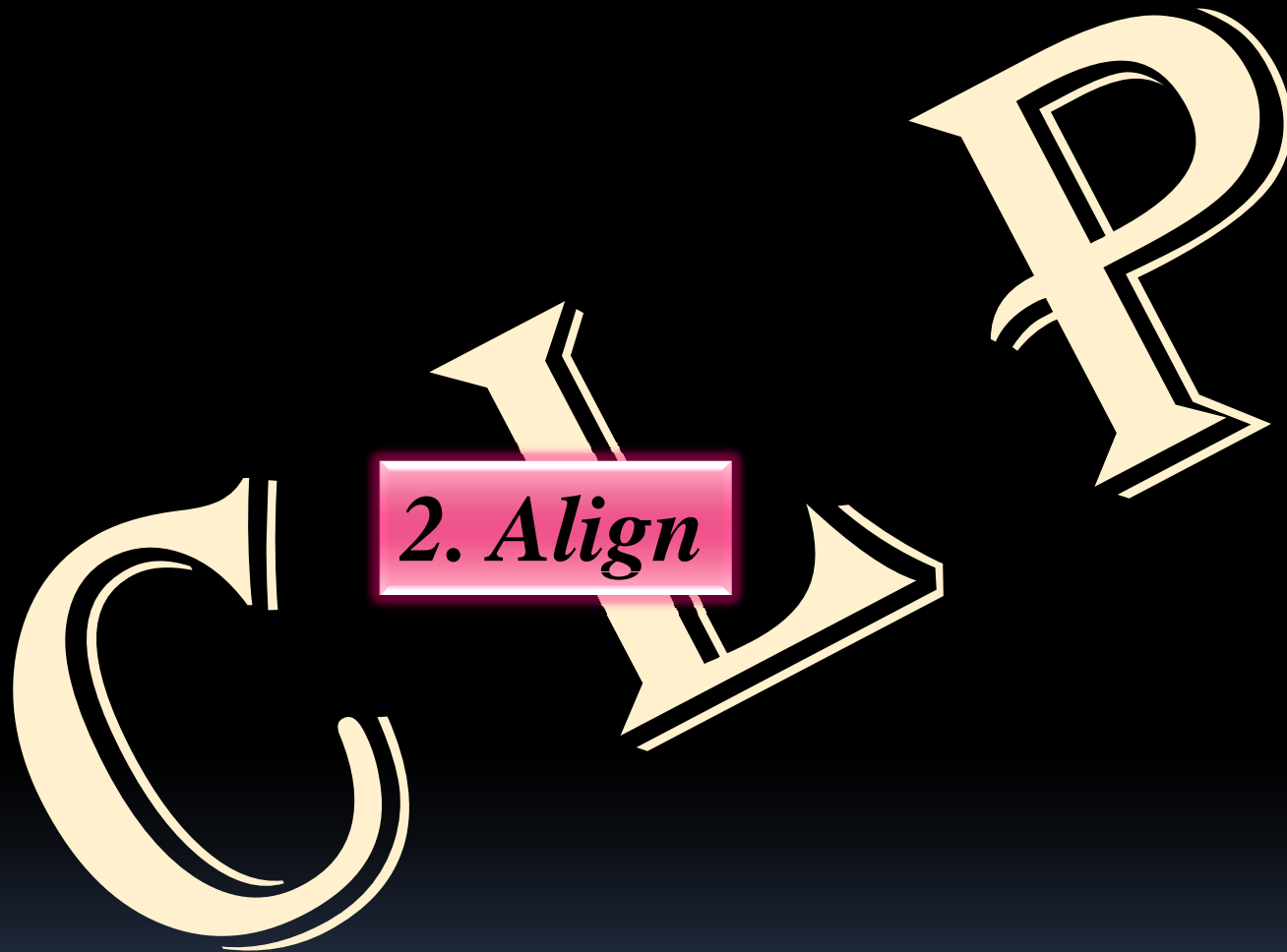


Selectați componenta care trebuie mișcată!



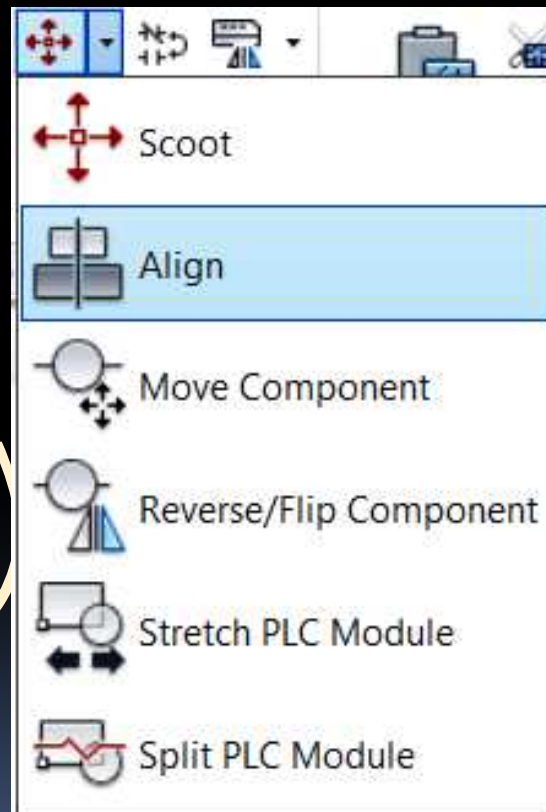
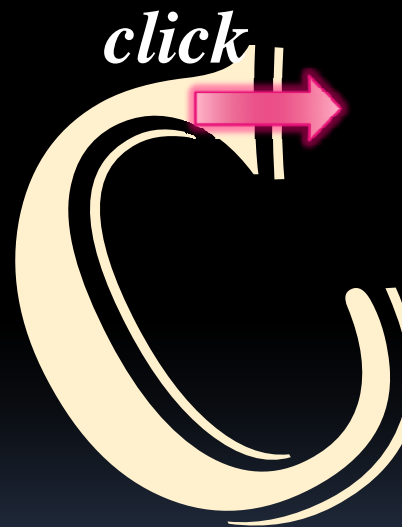
Selectați locul unde trebuie mișcată!



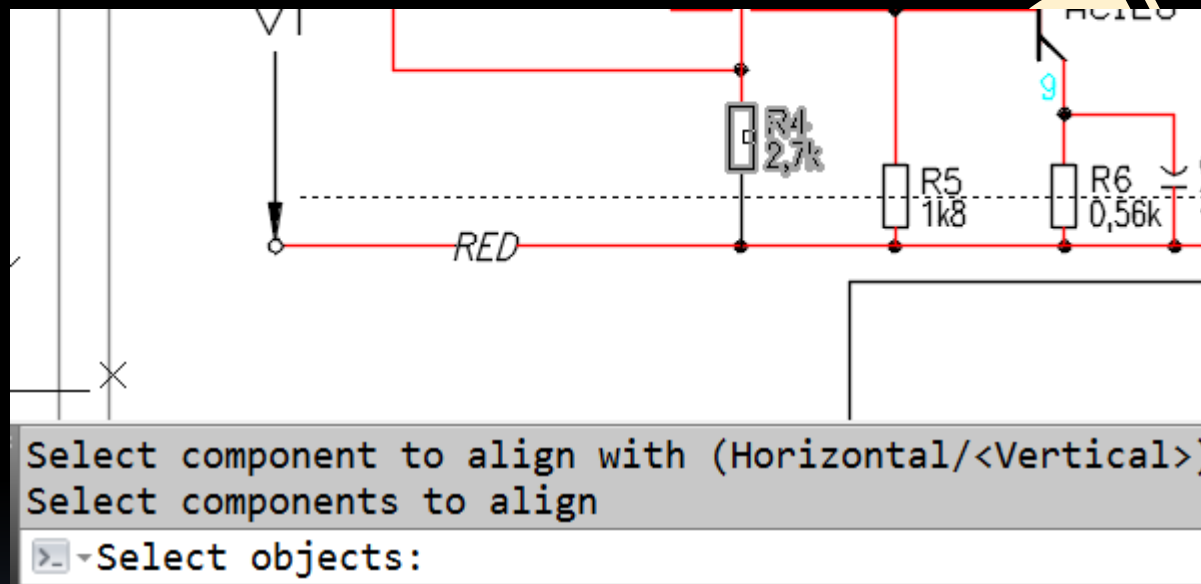


2. Align

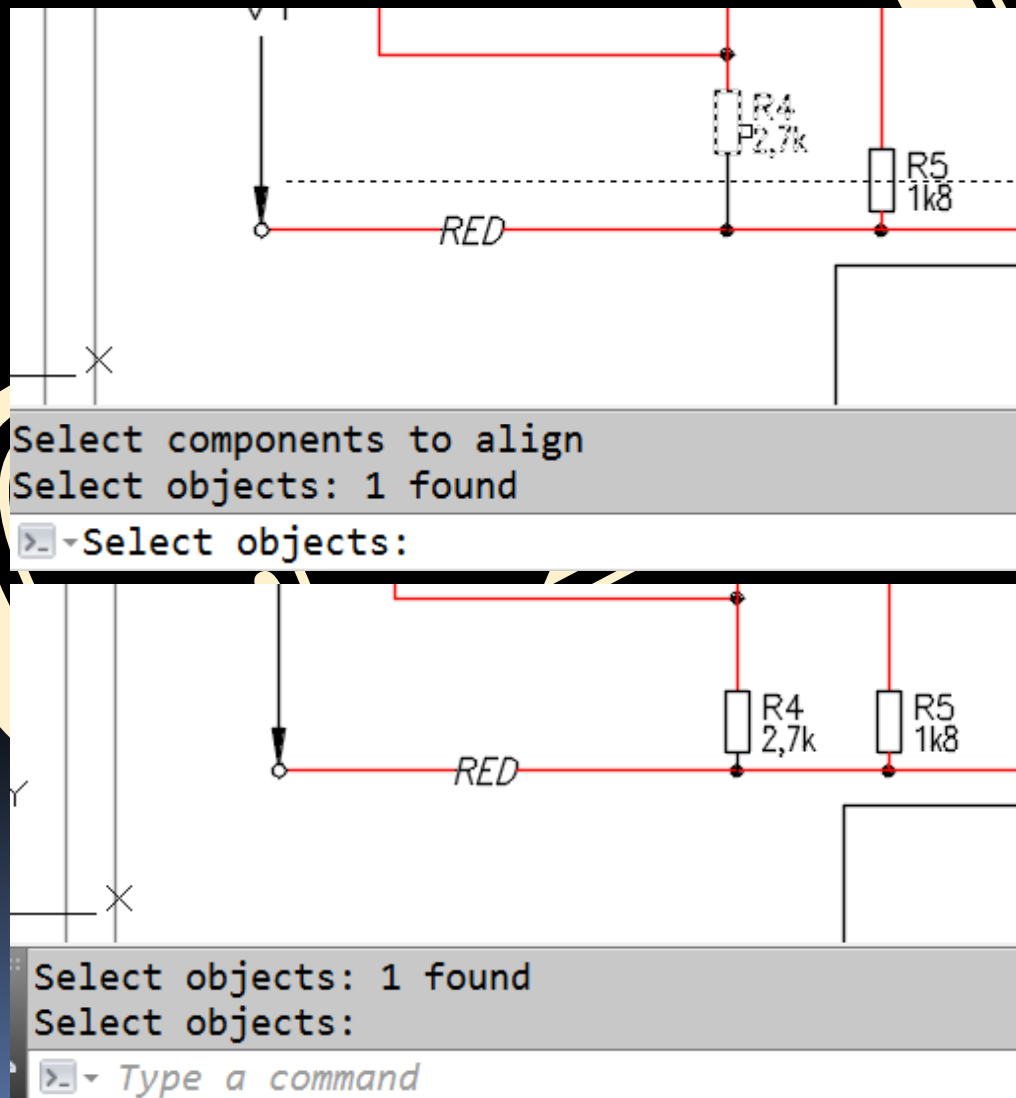
Afişul Align , panel Transform Component, Ribbon Schematic

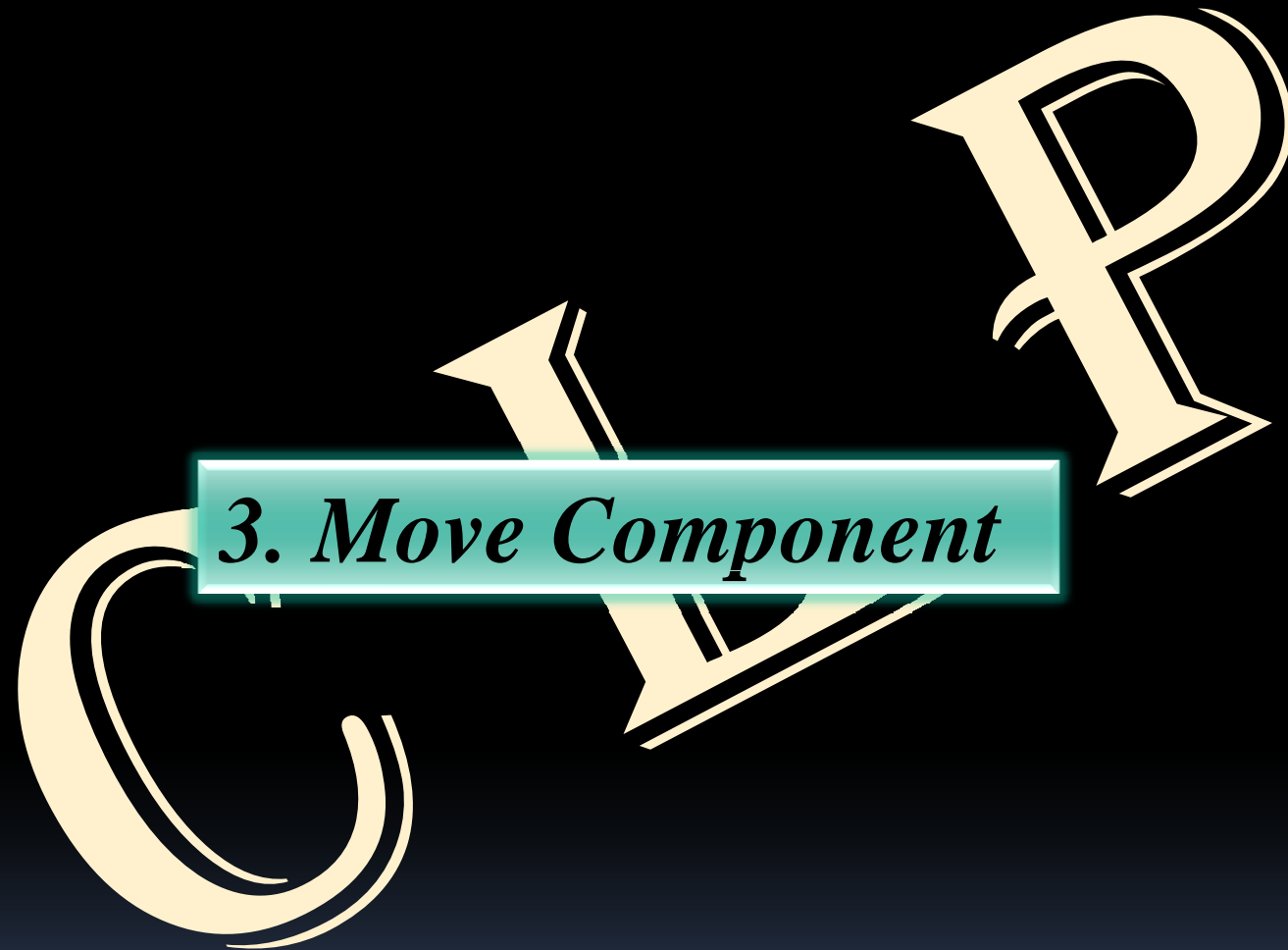


Selectați o entitate ca referință!



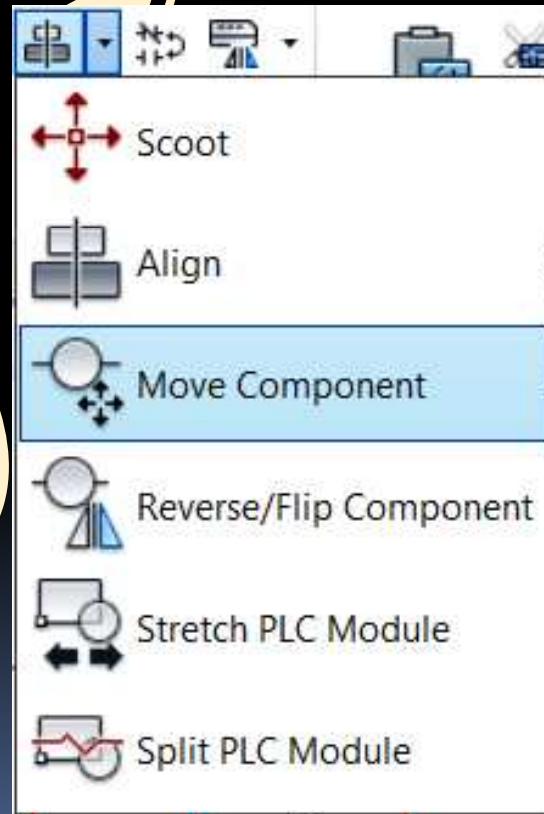
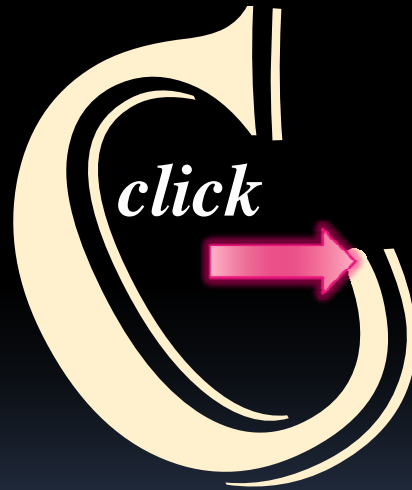
Selectați componentele care urmează să fie aliniate!

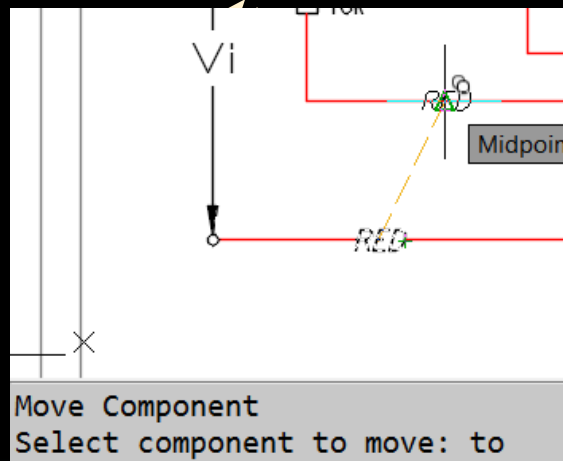
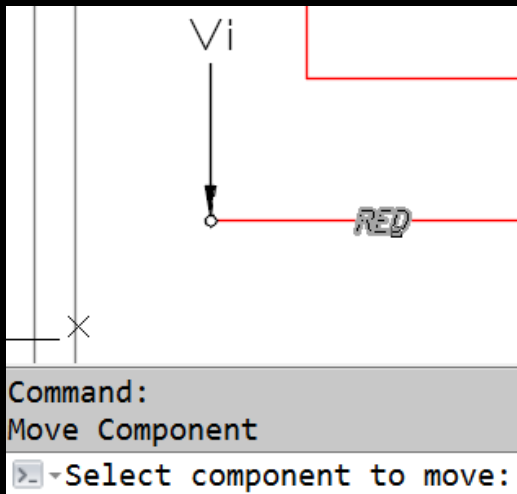




3. Move Component

Afişul Move Component, panel Transform Component, Ribbon Schematic

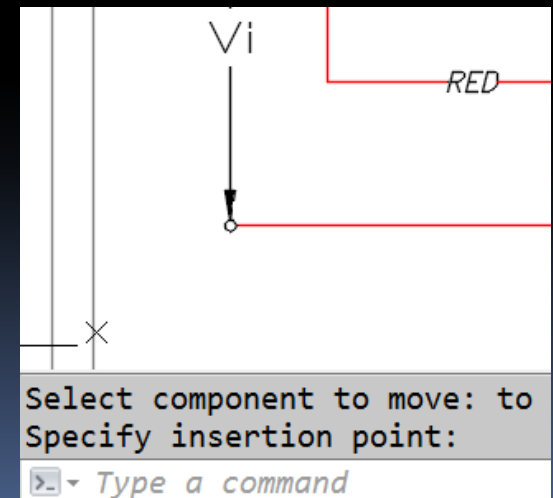




P

C

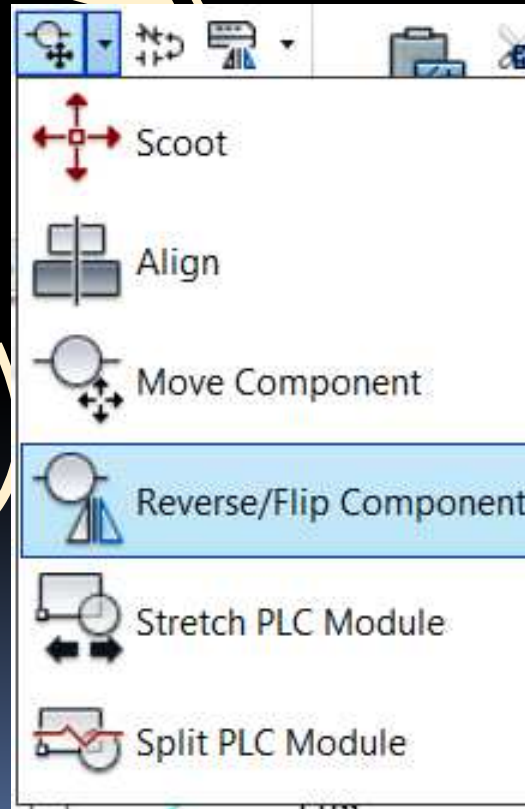
Se mută componentele într-un circuit, fără a crea spații în circuit !



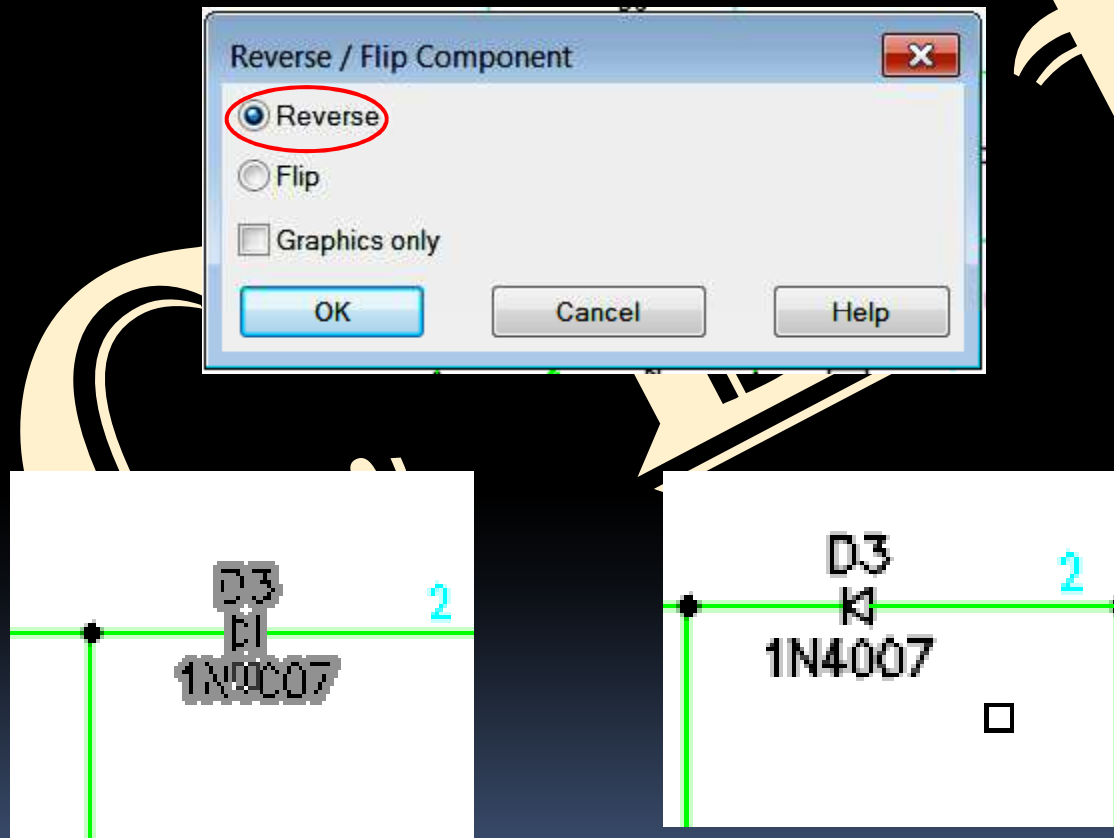
The image features a dark blue gradient background. In the center, the letters 'C', 'V', and 'P' are rendered in a large, stylized, yellow font with a white outline. A teal banner with a white border is positioned horizontally across the middle of the letters. The banner contains the text '4. Reversing or Flipping' in a black, italicized serif font.

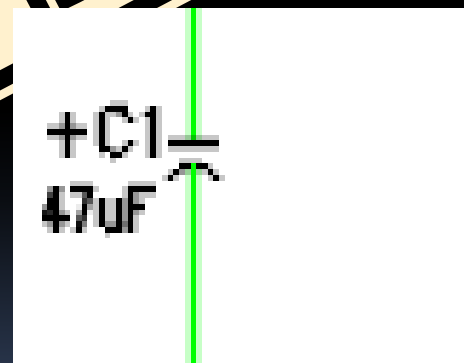
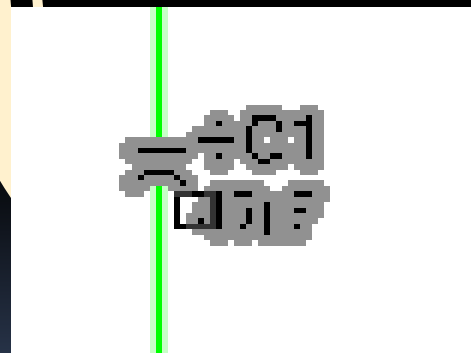
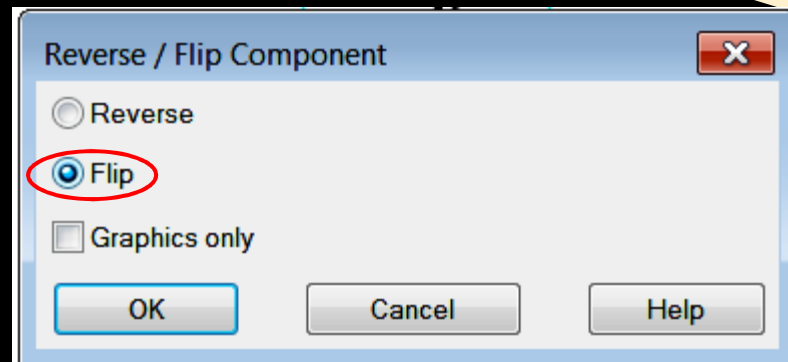
4. Reversing or Flipping

Afişul Reverse/Flip Component, panel Transform Component, Ribbon Schematic



Caseta de dialog Reverse/Flip Component







XII. Eitarea atributelor

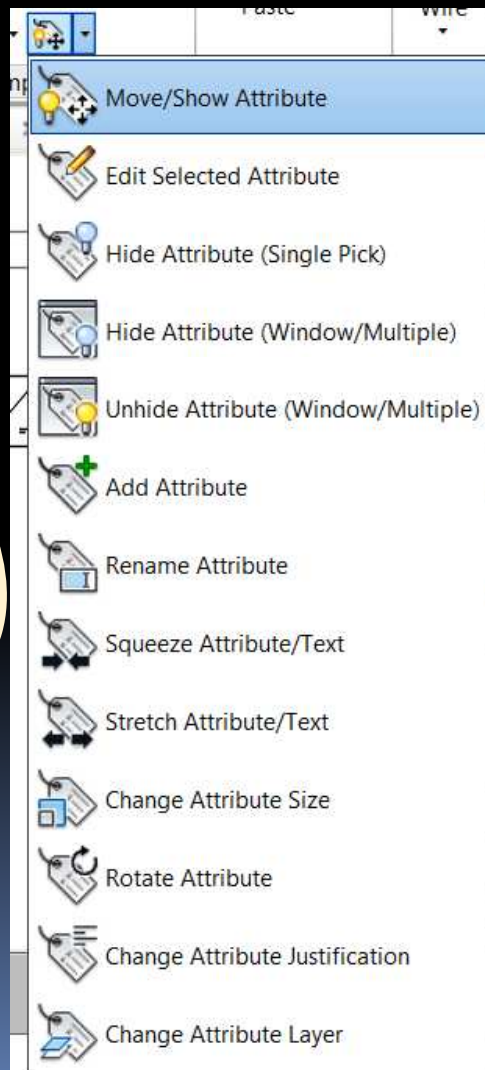
The image features three large, stylized yellow letters: 'C' on the left, 'L' in the center, and 'P' on the right. Each letter has a white outline and a drop shadow. The 'L' is rotated 45 degrees counter-clockwise. The 'C' and 'P' are upright. The letters are positioned around a central text block. The background is a gradient from black at the top to blue at the bottom.

Un atribut este folosit pentru a
face legătura dintre block-uri și
informația reală a cuvântului.



Edit Attribute

Afișul Reverse/Flip Component , panel Move/Show Attribute, Ribbon Schematic



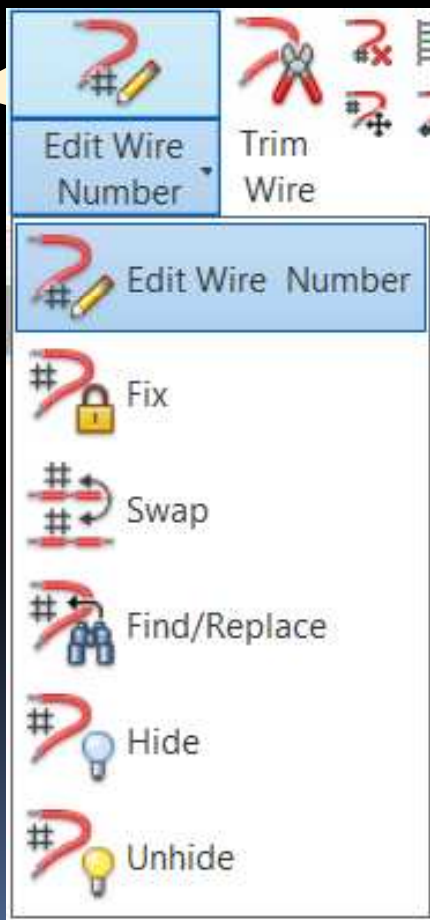
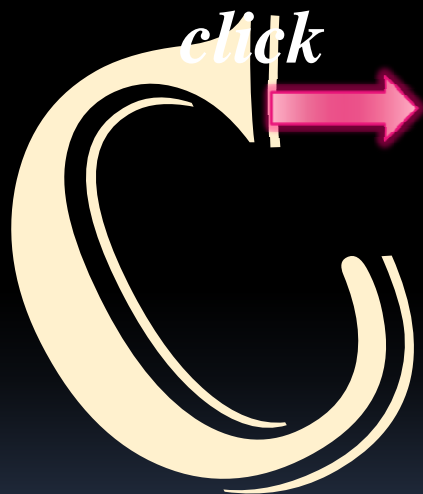


XII. Editarea firelor



1. Edit Wire Number

Afişul Edit Wire Numbers, panel Edit Wires/Wire Numbers, Ribbon Schematic



The image shows a circuit diagram with several components and wires. A command window is overlaid on the diagram. The circuit includes a transformer labeled '1FAZA' with input '1' and output '2'. A resistor 'R3 1K' is connected to wire '2' and wire '4'. A diode 'D2 RED' is connected to wire '4' and wire '5'. A wire labeled 'BLK' is connected to wire '5' and wire '6'. A wire labeled 'BLK_4AWI' is connected to wire '6' and wire '7'. A capacitor 'C5 680uF' is connected to wire '6' and wire '7'. A capacitor 'C1 22nF' is connected to wire '6' and wire '8'. A resistor 'R1' is connected to wire '8' and wire '9'. A component 'CN1 RCA' is connected to wire '9' and wire '10'. A component '1VP' is connected to wire '10' and wire '11'. A component 'INV' is connected to wire '11' and wire '12'. A component 'INV1' is connected to wire '12' and wire '13'. A component 'OUT1' is connected to wire '13' and wire '14'. A component 'D1' is connected to wire '14' and wire '15'. The command window contains the following text:

```
Command:  
Edit/Fixed/Un-fix tag  
-> Select wire or component for "Fixed" (Z=Zoom extents):
```

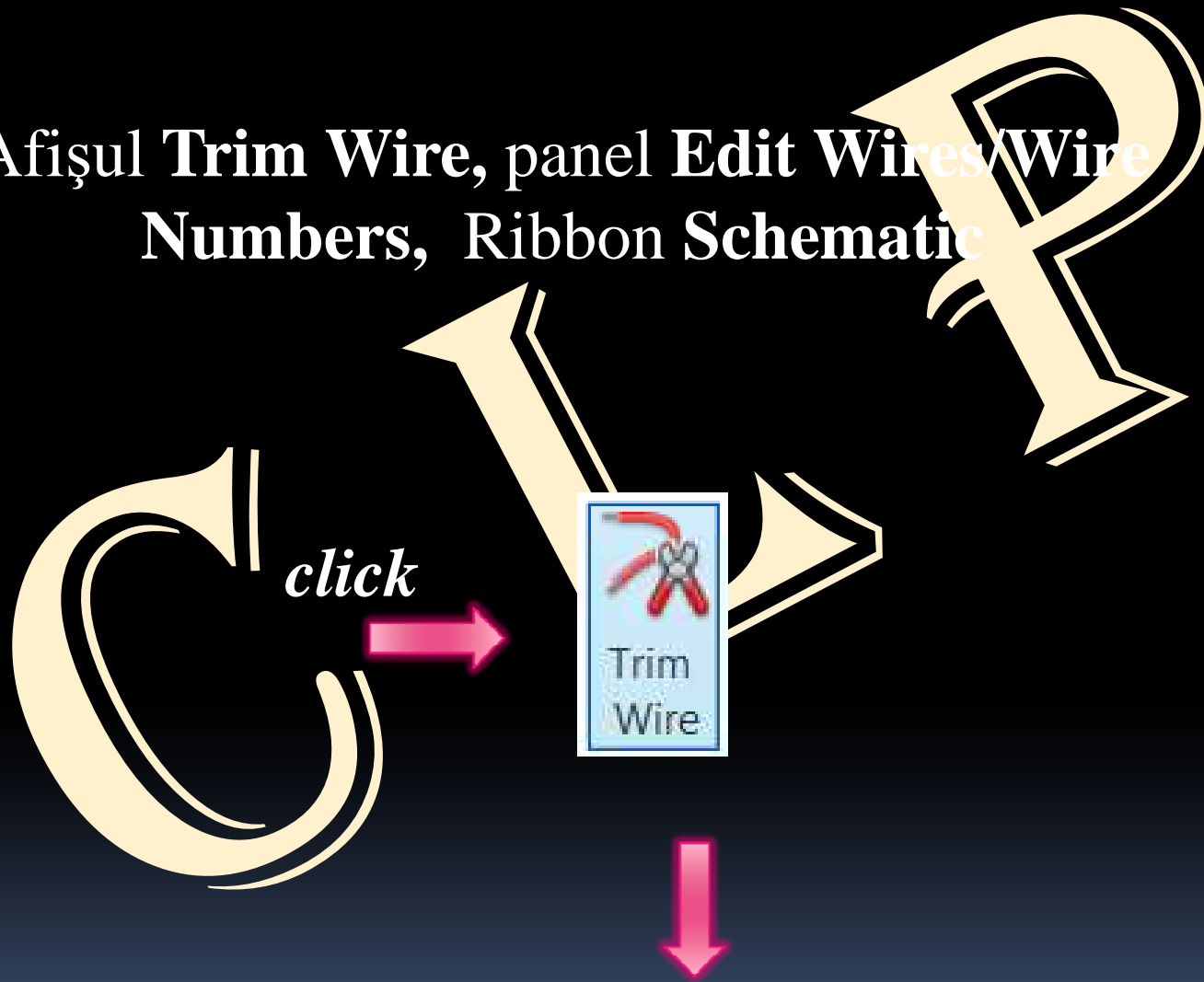
Caseta de dialog Modify/Fix/UnFix

The screenshot shows the 'Modify / Fix / UnFix' dialog box. The 'Wire number' field contains the number '3'. The 'Pick' button is highlighted. The 'similar text' checkbox is checked. The 'currently set to NORMAL' text is displayed. The 'Visible' radio button is selected. The 'Make it Fixed' checkbox is checked. The 'Hidden' radio button is unselected. The 'Zoom', 'OK', 'Cancel', and 'Help' buttons are visible at the bottom.

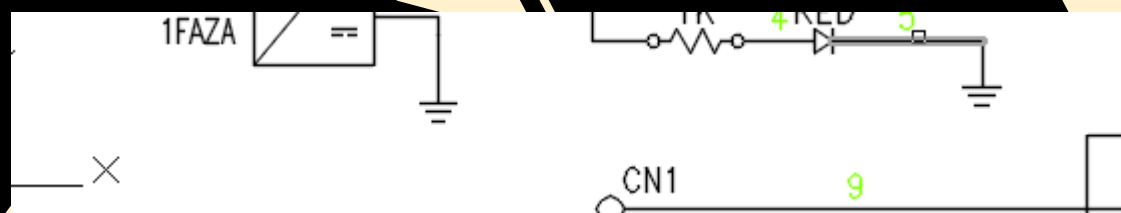
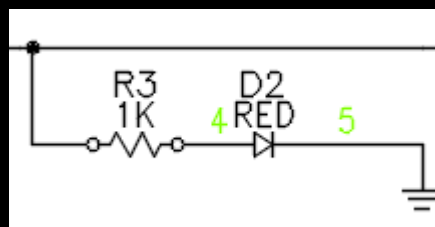


2. Trim Wire

Afişul Trim Wire, panel Edit Wires/Wire Numbers, Ribbon Schematic

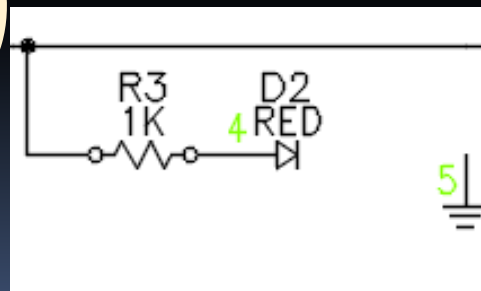


ÎNAINTE



```
Fence/Crossing/Zext/<Select wire to TRIM>:  
Fence/Crossing/Zext/Undo/<Select wire to TRIM>: u  
Fence/Crossing/Zext/Redo/<Select wire to TRIM>:
```

DUPĂ



The image features three large, stylized yellow letters: 'C' on the left, 'V' in the center, and 'P' on the right. The letters are rendered in a classic, slightly gothic font with a double-line outline. A horizontal blue rectangular box with a white border is superimposed over the middle of the 'V'. Inside this box, the text '3. Wire Editing' is written in a black, italicized serif font. The background is a dark blue gradient that transitions from black at the top to a lighter blue at the bottom.

3. Wire Editing

Afişul Wire Editing, panel Edit Wires/Wire Numbers, Ribbon Schematic

