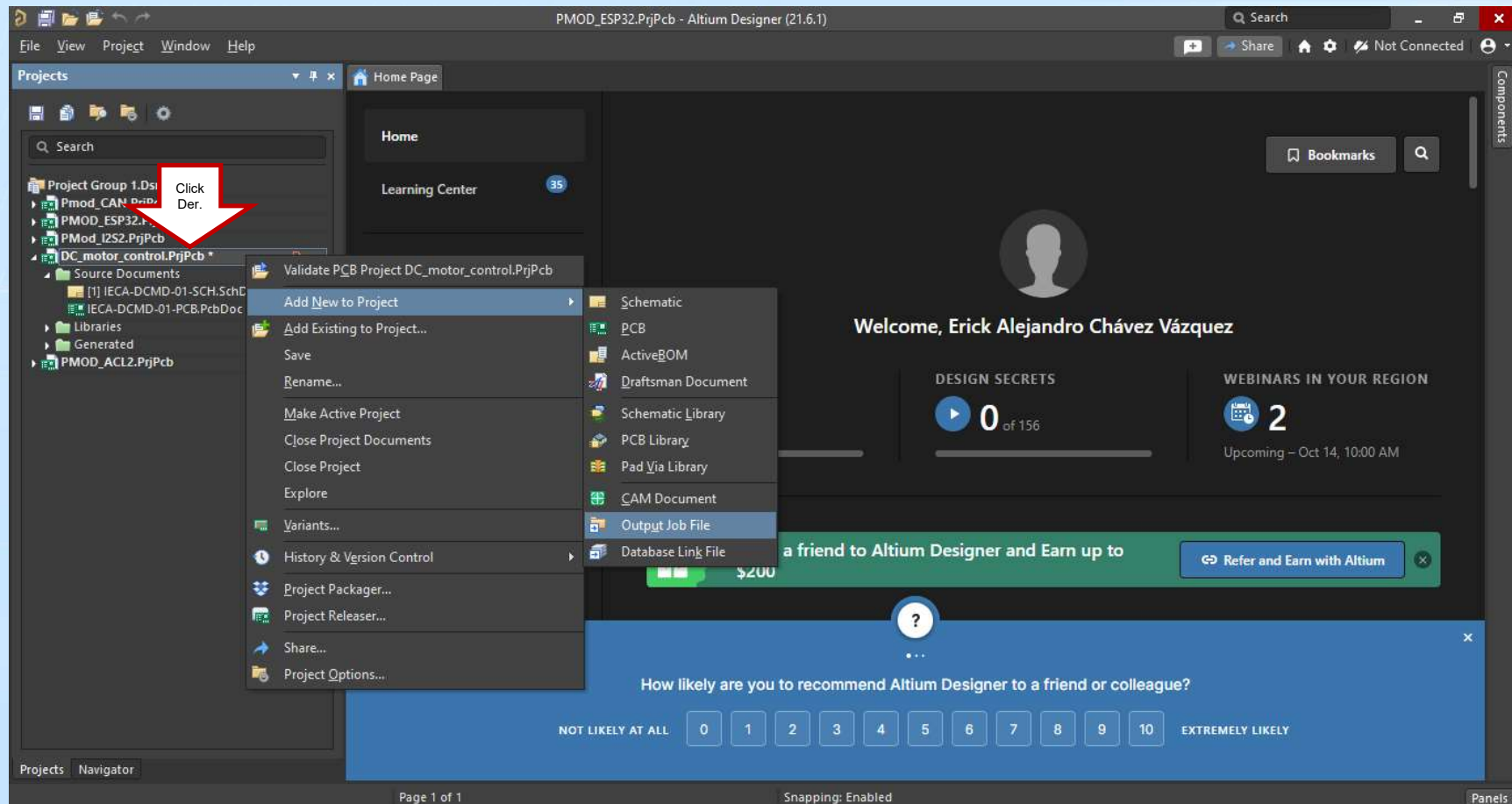


Creación de “Output Job” File para generar documentos de fabricación



The screenshot displays the Altium Designer (21.6.1) interface. The 'Projects' panel on the left shows a project tree with 'DC_motor_control.PriPcb' selected. A red arrow points to the 'Click Der.' button next to it. The 'File' menu is open, and the 'Add New to Project' option is highlighted, which has opened a sub-menu. In this sub-menu, the 'Output Job File' option is selected. The main workspace shows a 'Welcome' message for Erick Alejandro Chávez Vázquez, along with 'DESIGN SECRETS' (0 of 156) and 'WEBINARS IN YOUR REGION' (2 upcoming). A survey banner at the bottom asks 'How likely are you to recommend Altium Designer to a friend or colleague?' with a scale from 0 to 10.

DC_motor_control.PrjPcb - Altium Designer (21.6.1)

File Edit View Project Tools Window Help

Search [Share] [Home] [Not Connected]

Projects

Home Page Job1.OutJob

Search

Project Group 1.DsnWrk

- Pmod_CAN.PrjPcb
- PMOD_ESP32.PrjPcb
- PMOD_I2S2.PrjPcb
- DC_motor_control.PrjPcb ***
 - Source Documents
 - [1] IECA-DCMD-01-SCH.SchDoc
 - IECA-DCMD-01-PCB.PcbDoc
 - Settings
 - Output Job Files
 - Job1.OutJob**
 - Libraries
 - Generated
- PMOD_ACL2.PrjPcb

Variant Choice

This choice of Variant only applies to outputs generated from within this view. For release of PCB configurations, variant usage is always driven by the choice of variants in the PCB configuration

- ☒ Choose a single variant for the whole output job file [No Variations]
- ☐ Choose a different variant for each output

This variant will be only used when generating outputs and printing hardcopies from here.
If this output job is used for release of PCB configurations then this choice will be ignored and variant/parameter information will be passed to this output job by the PCB Release manager.

Name	Data Source	Output Description	Enabled
Netlist Outputs			
[Add New Netlist Output]			
Simulator Outputs			
[Add New Simulator Output]			
Documentation Outputs			
[Add New Documentation Output]			
Assembly Outputs			
[Add New Assembly Output]			
Fabrication Outputs			
[Add New Fabrication Output]			
Report Outputs			
[Add New Report Output]			
Validation Outputs			
[Add New Validation Output]			
Export Outputs			
[Add New Export Output]			
PostProcess Outputs			
[Add New PostProcess Output]			

Output Containers

Container

[Add New Output Container]

Hard Copy

Job Name

Print Job

Printer Default

[Add New Print Job]

Projects Navigator

Panels

Output Job File

Assembly Outputs

- Add New Assembly Outputs
- Test Point Report> <PCB>
- Pick and place files > <PCB>

Fabrication Outputs

- Add New Fabrication Output
- Gerber Files Top > <PCB>
- Gerber Files Bottom> <PCB>
- NC Drill Files > <PCB>
- Test Point Report> <PCB>

Report Outputs

- Add New Report Output
- Bill of Materials> [Project]
- Bill of Materials> Active BOM

Export Outputs

- Add New Export Output
- Export STEP > <PCB>
- Export VRML > <PCB>

Documentation Outputs

- Add New Documentation Output
- Schematic Print> All SCH Documents
- AddPDF3D > <PCB>
- Add Draftsman> <Draftsman para fabricación *.PCBDwf>
- Add Draftsman> <Draftsman para ensamble *.PCBDwf>
- Add Draftsman> <Draftsman de configuración *.PCBDwf>

New Folder > Test Point Report File
New Folder > Centroid File

New Folder > XGERBERS
New Folder > XGERBERS
New Folder > DRILLS
New Folder > Neutral File

New Folder > BOM
New Folder > BOM

New Folder > 3D STP
New Folder > 3D STP

New PDF > SCHEMATIC
New PDF > 3D
New PDF > FAB
New PDF > ASSY
New PDF > CFG

Archivo para puntos de prueba (Test Point)

The screenshot shows the Altium Designer (21.6.1) interface. The 'Variant Choice' dialog is open, and the 'Outputs' list is visible. The following steps are indicated by red arrows and numbers:

- 1**: Click on 'Test Point Report' in the 'Outputs' list.
- 2**: Click on 'IECA-DCMD-01-PCB.PcbDoc' in the 'Output Description' column.
- 3**: Click on 'New PDF' in the 'Add New Output Container' dialog.
- 4**: Click on 'New Folder Structure' in the 'Add New Output Container' dialog.

The 'Variant Choice' dialog contains the following text:

Variant Choice
This choice of Variant only applies to outputs generated from within this view. For release of PCB configurations, variant usage is always driven by the choice of variants in the PCB configuration.

☒ Choose a single variant for the whole output/job file [No Variations]
☐ Choose a different variant for each output

This variant will be only used when generating outputs and printing hardcopies from here. If this output job is used for release of PCB configurations then this choice will be ignored and variant/parameter information will be passed to this output job by the PCB Release manager.

Name	Data Source	Output Description	Enabled
Netlist Outputs			
[Add New Netlist Output]			
Simulation Outputs			
[Add New Simulation Output]			
Assembly Outputs			
[Add New Assembly Output]			
Fabrication Outputs			
[Add New Fabrication Output]			
Test Point Report		[PCB Document]	
[Add New Test Point Report]			
Validation Outputs			
[Add New Validation Output]			
Export Outputs			
[Add New Export Output]			
PostProcess Outputs			
[Add New PostProcess Output]			

The 'Add New Output Container' dialog shows the following options:

- New PDF
- New Video
- New Folder Structure

The 'Hard Copy' section shows the following options:

- Job Name
- Printer: Default

The 'Print Job' section shows the following options:

- [Add New Print Job]

Vincular archivo con su carpeta

The screenshot shows the Altium Designer (21.6.1) interface. The 'Variant Choice' dialog is open, displaying options for variant selection. The 'Outputs' table is visible, listing various output types and their descriptions. A red arrow labeled '1' points to the 'Test Point' output, and a red arrow labeled '2' points to the 'Test Point Report' output in the table.

Variant Choice

This choice of Variant only applies to outputs generated from within this view. For release of PCB configurations, variant usage is always driven by the choice of variants in the PCB configuration.

- ☒ Choose a single variant for the whole output/job file [No Variations]
- ☐ Choose a different variant for each output

This variant will be only used when generating outputs and printing hardcopies from here. If this output job is used for release of PCB configurations then this choice will be ignored and variant/parameter information will be passed to this output job by the PCB Release manager.

Name	Data Source	Output Description	Enabled
Netlist Outputs			
[Add New Netlist (
Simulator Outputs			
[Add New Simulat			
Documentation Outp			
[Add New Documr			
Assembly Outputs			
Test Point Report [PCB Document]		Test Point Report	1
[Add New Assemb			
Fabrication Outputs			
[Add New Fabrical			
Report Outputs			
[Add New Report			
Validation Outputs			
[Add New Validati			
Export Outputs			
[Add New Export (
PostProcess Outputs			
[Add New PostPro			

Outputs

Test Point [Change or Remove]

Release managem
Path in server
Local Path

Generate content →
Generate and publish →

[Add New Output Container]

Hard Copy
Job Name

Print Job

Printer Default

[Add New Print Job]

Archivo para la ubicación de los componentes (Pick and Place)

The screenshot shows the Altium Designer (21.6.1) interface. The 'Variant Choice' dialog is open, and the 'Outputs' list is visible. The 'Job1.OutJob' is selected in the 'Projects' panel. The 'Outputs' list shows the following items:

Name	Data Source	Output Description	Enabled
Netlist Outputs			
[Add New Netlist Output]			
Simulation Outputs			
[Add New Simulation Output]			
Assembly Outputs			
[Add New Assembly Output]			
Assembly Drawings			
Generates pick and place files		[PCB Document]	
Test Point Report		IECA-DCMD-01-PCB.PcbDoc	
Validation Outputs			
[Add New Validation Output]			
Export Outputs			
[Add New Export Output]			
PostProcess Outputs			
[Add New PostProcess Output]			

The 'Variant Choice' dialog has two options:

- ☒ Choose a single variant for the whole output/job file [No Variations]
- ☐ Choose a different variant for each output

The 'Hard Copy' section shows the 'Print Job' with the 'Printer' set to 'Default'.

Red arrows and numbers indicate the steps to generate pick and place files:

- Click on the 'Generates pick and place files' output.
- Click on the 'IECA-DCMD-01-PCB.PcbDoc' output.
- Click on the 'New PDF' button in the 'Hard Copy' section.
- Click on the 'New Folder Structure' button in the 'Hard Copy' section.

Vincular archivo con su carpeta

The screenshot shows the Altium Designer (21.6.1) interface. The 'Variant Choice' dialog is open, displaying options for variant selection. The 'Outputs' table lists various output types, and the 'Output Containers' section shows the 'Pick and Place' button. A red arrow labeled '1' points to the 'Pick and Place' button, and a red arrow labeled '2' points to the 'Generates pick and place files' row in the 'Outputs' table.

Variant Choice

This choice of Variant only applies to outputs generated from within this view. For release of PCB configurations, variant usage is always driven by the choice of variants in the PCB configuration.

- ☒ Choose a single variant for the whole output/job file [No Variations]
- ☐ Choose a different variant for each output

This variant will be only used when generating outputs and printing hardcopies from here. If this output job is used for release of PCB configurations then this choice will be ignored and variant/parameter information will be passed to this output job by the PCB Release manager.

Name	Data Source	Output Description	Enabled
Netlist Outputs			
[Add New Netlist C			
Simulator Outputs			
[Add New Simulat			
Documentation Outp			
[Add New Docume			
Assembly Outputs			
Generates pick an [PCB Document]		Generates pick and place files	1
[Add New Assemb			
Fabrication Outputs			
[Add New Fabrica			
Report Outputs			
[Add New Report			
Validation Outputs			
[Add New Validati			
Export Outputs			
[Add New Export C			
PostProcess Outputs			
[Add New PostPro			

Output Containers

Pick and Place [Change](#) or [Remove](#)

- Release manageme
- Path in server
- Local Path

[Generate content →](#)

[Generate and publish →](#)

[Add New Output Container]

Hard Copy

Job Name

Print Job

Printer Default

[Add New Print Job]

Archivos en formato Gerber

The screenshot displays the Altium Designer (21.6.1) interface. The 'Variant Choice' dialog box is open, showing a table of output types and their associated files. The 'Gerber Files' row is highlighted, and a context menu is open over it, showing options like 'Report Board Stack', 'Composite Drill Guide', 'Drill Drawings', 'Final', 'Gerber Files', 'Gerber X2 Files', 'IPC-2581', 'Mask Set', 'NC Drill Files', 'ODB++ Files', 'Power-Plane Set', and 'Test Point Report'. A red arrow labeled '1' points to the 'Gerber Files' row in the table. Another red arrow labeled '2' points to the 'Gerber Files' option in the context menu. A third red arrow labeled '3' points to the 'Click Der.' button. A fourth red arrow labeled '4' points to the 'New Folder Structure' option in the context menu. The background shows the Altium Designer workspace with a project tree on the left and a panel on the right.

Configuración de archivos Gerber

The screenshot shows the Altium Designer interface with the Gerber Setup dialog box open. The dialog box has four tabs: General, Layers, Drill Drawing, and Apertures. The Layers tab is selected, showing a table of layers to plot and mechanical layers to add to all plots.

Annotation 1: Points to the 'Configure...' button in the 'Gerber Files Top' section of the Projects panel.

Annotation 2: Points to the 'Layers' tab in the Gerber Setup dialog box.

Annotation 3: Points to the 'Mechanical Layers(s) to Add to All ...' table in the Layers tab.

Annotation 4: Points to the 'OK' button at the bottom right of the Gerber Setup dialog box.

Table 1: Layers To Plot

Ext...	Layer Name	Plot	Mirror
<input type="checkbox"/>	Top Overlay	<input checked="" type="checkbox"/>	<input type="checkbox"/>
<input type="checkbox"/>	Top Paste	<input checked="" type="checkbox"/>	<input type="checkbox"/>
<input type="checkbox"/>	Top Solder	<input checked="" type="checkbox"/>	<input type="checkbox"/>
<input type="checkbox"/>	Top Layer	<input checked="" type="checkbox"/>	<input type="checkbox"/>
<input type="checkbox"/>	Bottom Layer	<input type="checkbox"/>	<input type="checkbox"/>
<input type="checkbox"/>	Bottom Solder	<input type="checkbox"/>	<input type="checkbox"/>
<input type="checkbox"/>	Bottom Paste	<input type="checkbox"/>	<input type="checkbox"/>
<input type="checkbox"/>	Bottom Overlay	<input type="checkbox"/>	<input type="checkbox"/>
<input type="checkbox"/>	Mechanical 1	<input type="checkbox"/>	<input type="checkbox"/>
<input type="checkbox"/>	Mechanical 2	<input type="checkbox"/>	<input type="checkbox"/>
<input type="checkbox"/>	Mechanical 3	<input type="checkbox"/>	<input type="checkbox"/>
<input type="checkbox"/>	Top Assembly	<input type="checkbox"/>	<input type="checkbox"/>
<input type="checkbox"/>	Bottom Assembly	<input type="checkbox"/>	<input type="checkbox"/>
<input type="checkbox"/>	Top Courtyard	<input type="checkbox"/>	<input type="checkbox"/>
<input type="checkbox"/>	Bottom Courtyard	<input type="checkbox"/>	<input type="checkbox"/>
<input type="checkbox"/>	Top Designator	<input type="checkbox"/>	<input type="checkbox"/>
<input type="checkbox"/>	Bottom Designator	<input type="checkbox"/>	<input type="checkbox"/>
<input type="checkbox"/>	Mechanical 13	<input type="checkbox"/>	<input type="checkbox"/>
<input type="checkbox"/>	Mechanical 15	<input type="checkbox"/>	<input type="checkbox"/>
<input type="checkbox"/>	Keep-Out Layer	<input type="checkbox"/>	<input type="checkbox"/>
<input type="checkbox"/>	Top Pad Master	<input type="checkbox"/>	<input type="checkbox"/>

Table 2: Mechanical Layers(s) to Add to All ...

Layer Name	Plot
Mechanical 1	<input type="checkbox"/>
Mechanical 2	<input type="checkbox"/>
Mechanical 3	<input type="checkbox"/>
Top Assembly	<input type="checkbox"/>
Bottom Assembly	<input type="checkbox"/>
Top Courtyard	<input type="checkbox"/>
Bottom Courtyard	<input type="checkbox"/>
Top Designator	<input type="checkbox"/>
Bottom Designator	<input type="checkbox"/>
Mechanical 13	<input type="checkbox"/>
Mechanical 15	<input type="checkbox"/>

Vincular archivo con su carpeta

DC_motor_control.PrjPcb - Altium Designer (21.6.1)

File Edit View Project Tools Window Help

Projects

Search

Project Group 1.DsnWrk

- Pmod_CAN.PrjPcb
- Pmod_ESP32.PrjPcb
- Pmod_I2S2.PrjPcb
- DC_motor_control.PrjPcb *
- Source Documents
 - [1] IECA-DCMD-01-SCH.SchDoc
 - IECA-DCMD-01-PCB.PcbDoc
- Settings
 - Output Job Files
 - Job1.OutJob *
 - Libraries
 - Generated
- Pmod_ACL2.PrjPcb

Variant Choice

This choice of Variant only applies to outputs generated from within this view. For release of PCB configurations, variant usage is always driven by the choice of variants in the PCB configuration

☒ Choose a single variant for the whole output/job file [No Variations]

☐ Choose a different variant for each output

This variant will be only used when generating outputs and printing hardcopies from here. If this output job is used for release of PCB configurations then this choice will be ignored and variant/parameter information will be passed to this output job by the PCB Release manager.

Name	Data Source	Output Description	Enabled
Netlist Outputs			
[Add New Netlist (
Simulator Outputs			
[Add New Simulat			
Documentation Outp			
[Add New Documr			
Assembly Outputs			
[Add New Assemb			
Fabrication Outputs			
Gerber Files Top IECA-DCMD-01-PCB.PcbDoc	Gerber Files	1	
Gerber Files Botto IECA-DCMD-01-PCB.PcbDoc	Gerber Files	2	
[Add New Fabrical			
Report Outputs			
[Add New Report			
Validation Outputs			
[Add New Validati			
Export Outputs			
[Add New Export (
PostProcess Outputs			
[Add New PostPro			

Outputs

Gerber

Release managemr

Path in server

Local Path

Change or Remove

Generate content →

Generate and publish →

[Add New Output Container]

Hard Copy

Job Name

Print Job

Printer

Default

[Add New Print Job]

Projects Navigator

Panels

Archivos de perforaciones (Drill Files)

The screenshot shows the Altium Designer 21.6.1 interface. The 'Variant Choice' dialog is open, and the 'Outputs' list is visible. The following steps are indicated by red arrows and numbers:

- 1**: Click on 'NC Drill Files' in the 'Outputs' list.
- 2**: Click on 'IECA-DCMD-01-PCB.PcbDoc' in the 'Outputs' list.
- 3**: Click on 'New PDF' in the 'Add New Output' dialog.
- 4**: Click on 'New Folder Structure' in the 'Add New Output' dialog.

The 'Variant Choice' dialog includes the following text:

Variant Choice
This choice of Variant only applies to outputs generated from within this view. For release of PCB configurations, variant usage is always driven by the choice of variants in the PCB configuration.

Choose a single variant for the whole output job file [No Variations]

for each output

used when generating outputs and printing hardcopies from here.
for release of PCB configurations then this choice will be ignored and variant/parameter information will be passed to this output job by the

Outputs

Name	Output Description	Enabled
[Add New Output]		
[PCB Document]		
IECA-DCMD-01-PCB.PcbDoc		

Hard Copy
Job Name

Print Job
Printer Default

Configuración de archivos de perforaciones

DC_motor_control.PrjPcb - Altium Designer (21.6.1)

File Edit View Project Tools Window Help

Projects

Search

Project Group 1.DsnWrk

- Pmod_CAN.PrjPcb
- Pmod_ESP32.PrjPcb
- Pmod_I2S2.PrjPcb
- DC_motor_control.PrjPcb *
- Source Documents
 - [1] IECA-DCMD-01-SCH.SchDoc
 - IECA-DCMD-01-PCB.PcbDoc
- Settings
 - Output Job Files
 - Job1.OutJob *
 - Libraries
 - Generated
- PMOD_ACL2.PrjPcb

Variant Choice

This choice of Variant only applies to outputs generated from within this view. For release of PCB configuration

Choose a single variant for the whole output file [No Variation]

☐ Cut Ctrl+X
☐ Copy Ctrl+C
☐ Paste Ctrl+V
☐ Duplicate Ctrl+D
☐ Delete Del
☐ Enable All
☐ Disable All
☐ Enable Selected Ctrl+TECLA DE ADICION
☐ Disable Selected Ctrl+TECLA DE SUSTRACCION
☐ Configure... Alt+Enter
☐ Document Options

NC Drill Setup

NC Drill Format

Specify the units and format to be used in the NC Drill output files.

This controls the units (inches or millimeters), and the number of digits before and after the decimal point.

Units: ☒ Inches ☐ Millimeters
 Format: ☐ 2:3 ☐ 2:4 ☒ 2:5

The number format should be set to suit the requirements of your design. The 2:3 format has a 1 mil resolution, 2:4 has a 0.1 mil resolution, and 2:5 has a 0.01 mil resolution.

If you are using one of the higher resolutions you should check that the PCB manufacturer supports that format. The 2:4 and 2:5 formats only need to be chosen if there are holes on a grid finer than 1 mil.

Leading/Trailing Zeroes: ☐ Keep leading and trailing zeroes ☐ Suppress leading zeroes ☒ Suppress trailing zeroes

Coordinate Positions: ☐ Reference to absolute origin ☒ Reference to relative origin

Other

- ☒ Optimize change location commands
- ☒ Generate separate NC Drill files for plated & non-plated holes
- ☐ Use drilled slot command (G85)
- ☒ Generate Board Edge Rout Paths

Rout Tool Dia: 10mil

☐ Generate EIA Binary Drill File (.DRL)

OK Cancel

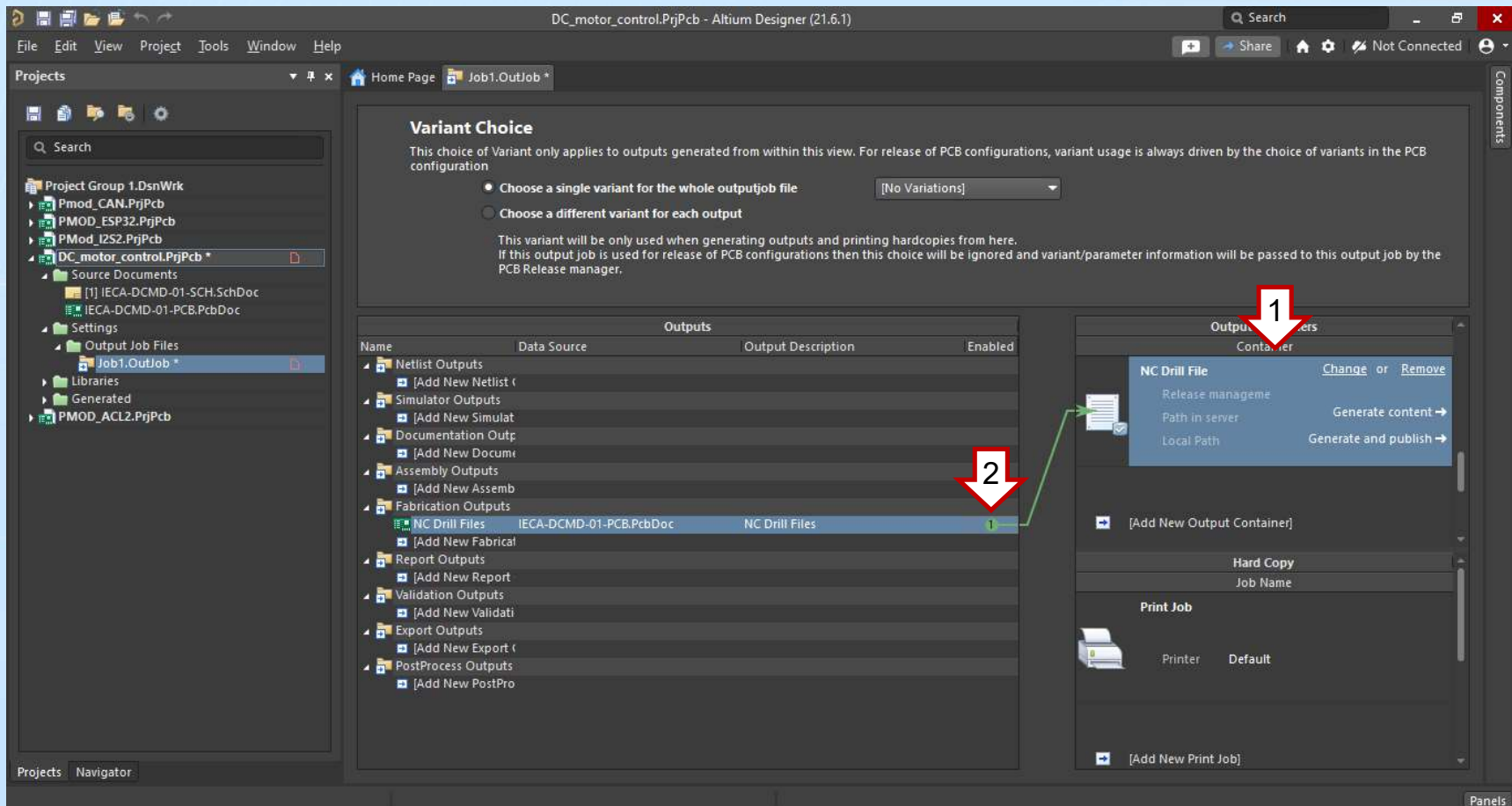
Click Der.

1

2

3

Vincular archivo con su carpeta



DC_motor_control.PrjPcb - Altium Designer (21.6.1)

File Edit View Project Tools Window Help

Projects Home Page Job1.OutJob *

Search

Project Group 1.DsnWrk

- Pmod_CAN.PrjPcb
- PMod_ESP32.PrjPcb
- PMod_I2S2.PrjPcb
- DC_motor_control.PrjPcb *
- Source Documents
 - [1] IECA-DCMD-01-SCH.SchDoc
 - IECA-DCMD-01-PCB.PcbDoc
- Settings
 - Output Job Files
 - Job1.OutJob *
 - Libraries
 - Generated
- PMod_ACL2.PrjPcb

Variant Choice

This choice of Variant only applies to outputs generated from within this view. For release of PCB configurations, variant usage is always driven by the choice of variants in the PCB configuration

- Choose a single variant for the whole output/job file [No Variations]
- Choose a different variant for each output

This variant will be only used when generating outputs and printing hardcopies from here. If this output job is used for release of PCB configurations then this choice will be ignored and variant/parameter information will be passed to this output job by the PCB Release manager.

Name	Data Source	Output Description	Enabled
Netlist Outputs			
[Add New Netlist]			
Simulator Outputs			
[Add New Simulator]			
Documentation Outputs			
[Add New Documentation]			
Assembly Outputs			
[Add New Assembly]			
Fabrication Outputs			
NC Drill Files	IECA-DCMD-01-PCB.PcbDoc	NC Drill Files	1
[Add New Fabrication]			
Report Outputs			
[Add New Report]			
Validation Outputs			
[Add New Validation]			
Export Outputs			
[Add New Export]			
PostProcess Outputs			
[Add New PostProcess]			

Output Containers

NC Drill File [Change or Remove]

Release manager

Path in server Generate content →

Local Path Generate and publish →

[Add New Output Container]

Hard Copy

Job Name

Print Job

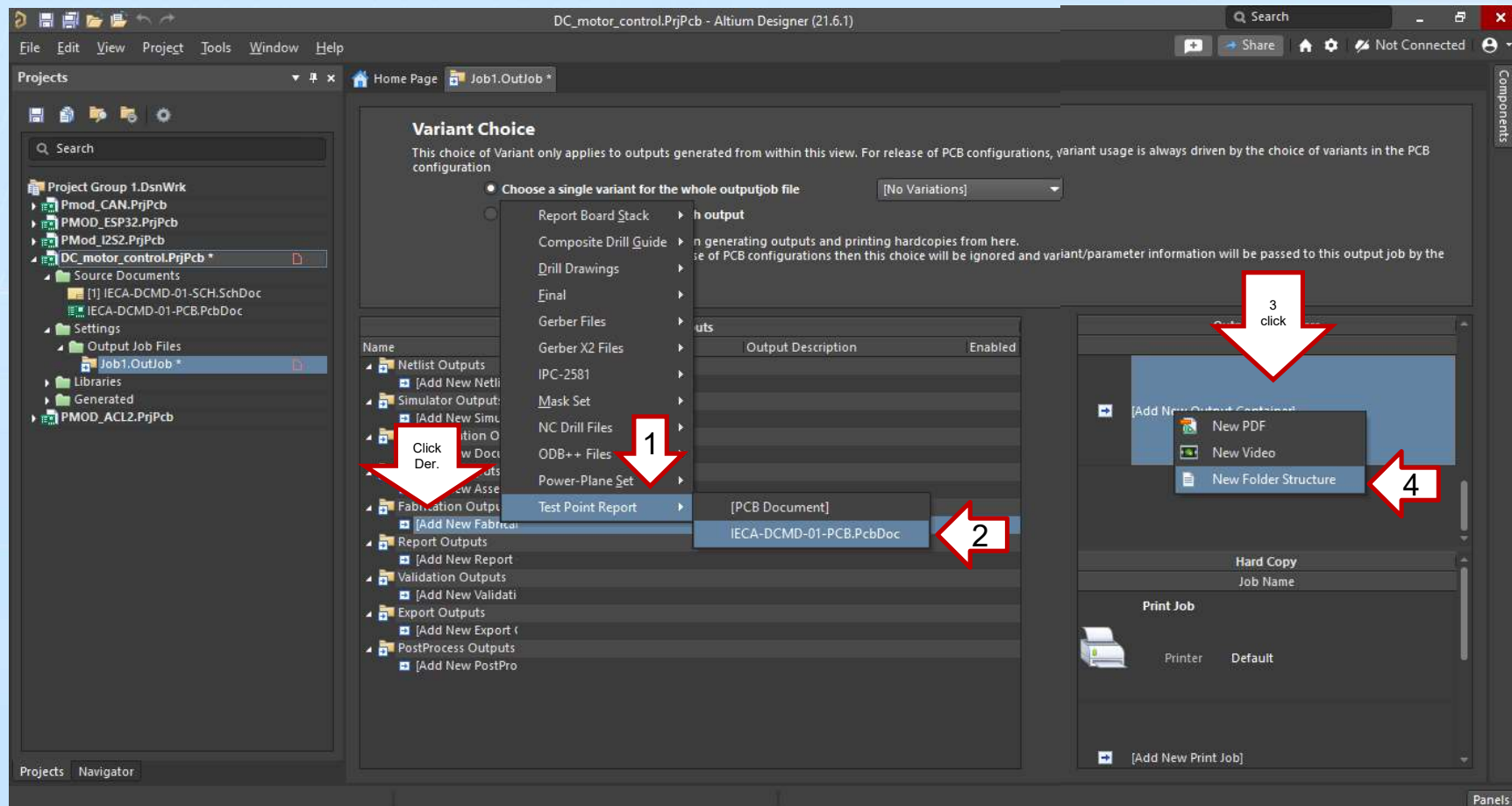
Printer Default

[Add New Print Job]

Projects Navigator

Panels

Archivo para prueba eléctrica (Neutral File)



Altium Designer (21.6.1) interface showing the 'Variant Choice' dialog box. The dialog is titled 'Variant Choice' and contains a section 'Choose a single variant for the whole output/job file' with a dropdown menu set to '[No Variations]'. Below this is a table with columns 'Name', 'Output Description', and 'Enabled'. The table lists various output types: Netlist Outputs, Simulator Outputs, Fabrication Outputs, Report Outputs, Validation Outputs, Export Outputs, and PostProcess Outputs. A red arrow labeled '1' points to the 'Test Point Report' row. A red arrow labeled '2' points to the 'IECA-DCMD-01-PCB.PcbDoc' file in the list. A red arrow labeled '3' points to the 'New PDF' button in the 'Add New Output' section. A red arrow labeled '4' points to the 'New Folder Structure' button in the 'Add New Output' section. The 'Projects' panel on the left shows the project hierarchy, including 'DC_motor_control.PrjPcb' and 'Job1.OutJob *'. The 'Components' panel on the right is empty.

Configuración de archivo Neutral File

The screenshot shows the Altium Designer (21.6.1) interface. The 'Variant Choice' dialog is open, and the 'Fabrication Testpoint Setup' dialog is also open. Red arrows and numbers 1, 2, and 3 indicate the steps:

- Click Derive... in the Variant Choice dialog.
- Click the 'Fabrication Testpoint Setup' dialog.
- Click the OK button in the Fabrication Testpoint Setup dialog.

The 'Fabrication Testpoint Setup' dialog has the following settings:

- Report Formats:**
 - ☐ Text
 - ☐ CSV
 - ☒ IPC-D-356A
- Test Point Layers:**
 - ☒ Top layer
 - ☒ Bottom layer
- Units:**
 - ☒ Imperial
 - ☐ Metric
- Coordinate Positions:**
 - ☐ Reference to absolute origin
 - ☒ Reference to relative origin
- IPC-D-356A Options:**
 - ☐ Adjacency Information: 25mil
 - ☒ Board Outline: Mechanical 2
 - ☒ Conductor Traces
 - ☐ Merge Net-Tie Nets

Vincular archivo con su carpeta

Variant Choice

This choice of Variant only applies to outputs generated from within this view. For release of PCB configurations, variant usage is always driven by the choice of variants in the PCB configuration.

- ☒ Choose a single variant for the whole output/job file [No Variations]
- ☐ Choose a different variant for each output

This variant will be only used when generating outputs and printing hardcopies from here. If this output job is used for release of PCB configurations then this choice will be ignored and variant/parameter information will be passed to this output job by the PCB Release manager.

Name	Data Source	Output Description	Enabled
Netlist Outputs			
Simulator Outputs			
Documentation Outputs			
Assembly Outputs			
Fabrication Outputs			
Neutral File	IECA-DCMD-01-PCB.PcbDoc	Test Point Report	1
Report Outputs			
Validation Outputs			
Export Outputs			
PostProcess Outputs			

Outputs

Neutral File [Change](#) or [Remove](#)

Release managem
Path in server
Local Path

[Generate content →](#)
[Generate and publish →](#)

[Add New Output Container]

Hard Copy
Job Name

Print Job

Printer Default

[Add New Print Job]

Lista de materiales (BOM)

Variant Choice

This choice of Variant only applies to outputs generated from within this view. For release of PCB configurations, variant usage is always driven by the choice of variants in the PCB configuration.

- Choose a single variant for the whole output/job file [No Variations]
- Choose a different variant for each output

This variant will be only used when generating outputs and printing hardcopies from here. If this output job is used for release of PCB configurations then this choice will be ignored and variant/parameter information will be passed to this output job by the PCB Release manager.

Name	Data Source	Output Description	Enabled
Netlist Outputs			
Simulator Outputs			
Documentation Outputs			
Reports Outputs			
Validation Outputs			
Export Outputs			
PostProcess Outputs			

1 Click Der.

2 BOM Compare (Requires workspace)

3 click

4 New PDF

5 New Folder Structure

6 Print Job

7 Printer Default

Vincular archivo con su carpeta

DC_motor_control.PrjPcb - Altium Designer (21.6.1)

File Edit View Project Tools Window Help

Projects Home Page Job1.OutJob *

Search

Project Group 1.DsnWrk

- Pmod_CAN.PrjPcb
- PMod_ESP32.PrjPcb
- PMod_I2S2.PrjPcb
- DC_motor_control.PrjPcb *
- Source Documents
 - [1] IECA-DCMD-01-SCH.SchDoc
 - IECA-DCMD-01-PCB.PcbDoc
- Settings
 - Output Job Files
 - Job1.OutJob *
- Libraries
- Generated
- PMod_ACL2.PrjPcb

Variant Choice

This choice of Variant only applies to outputs generated from within this view. For release of PCB configurations, variant usage is always driven by the choice of variants in the PCB configuration

- Choose a single variant for the whole outputjob file [No Variations]
- Choose a different variant for each output

This variant will be only used when generating outputs and printing hardcopies from here.
If this output job is used for release of PCB configurations then this choice will be ignored and variant/parameter information will be passed to this output job by the PCB Release manager.

Name	Data Source	Output Description	Enabled
Netlist Outputs			
[Add New Netlist (
Simulator Outputs			
[Add New Simulat			
Documentation Outp			
[Add New Docume			
Assembly Outputs			
[Add New Assemb			
Fabrication Outputs			
[Add New Fabrica			
Report Outputs			
Bill of Materials (ActiveBOM Document)		Bill of Materials	1
[Add New Report			
Validation Outputs			
[Add New Validati			
Export Outputs			
[Add New Export (
PostProcess Outputs			
[Add New PostPro			

Output Containers

BOM

Release managem

Path in server

Local Path

Change or Remove

Generate content →

Generate and publish →

[Add New Output Container]

Hard Copy

Job Name

Print Job

Printer

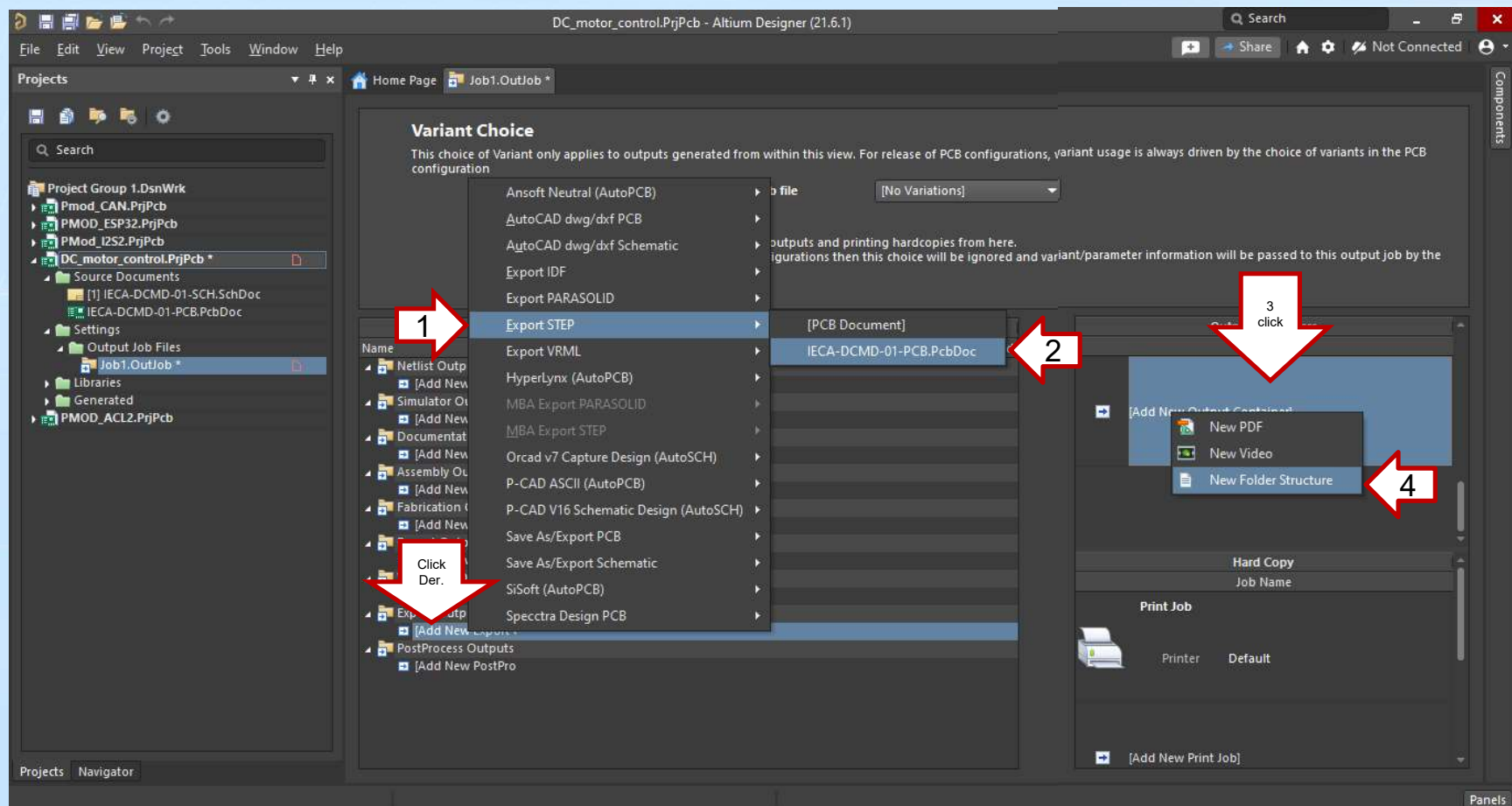
Default

[Add New Print Job]

Projects Navigator

Panels

Archivos 3D



Vincular archivo con su carpeta

DC_motor_control.PrjPcb - Altium Designer (21.6.1)

File Edit View Project Tools Window Help

Search

Share Not Connected

Projects

Home Page Job1.OutJob *

Variant Choice

This choice of Variant only applies to outputs generated from within this view. For release of PCB configurations, variant usage is always driven by the choice of variants in the PCB configuration.

Choose a single variant for the whole output/job file [No Variations]

Choose a different variant for each output

This variant will be only used when generating outputs and printing hardcopies from here. If this output job is used for release of PCB configurations then this choice will be ignored and variant/parameter information will be passed to this output job by the PCB Release manager.

Name	Data Source	Output Description	Enabled
Netlist Outputs			
Simulator Outputs			
Documentation Outputs			
Assembly Outputs			
Fabrication Outputs			
Report Outputs			
Validation Outputs			
Export Outputs			
Export STEP	IECA-DCMD-01-PCB.PcbDoc	Export STEP	
AutoCAD dwg/dxf	IECA-DCMD-01-PCB.PcbDoc	AutoCAD dwg/dxf File PCB	
PostProcess Outputs			

1

2

3D

Release management

Path in server

Local Path

Change or Remove

Generate content →

Generate and publish →

[Add New Output Container]

Hard Copy

Job Name

Print Job

Printer

Default

[Add New Print Job]

Projects Navigator

Panels

Archivo de Esquemático

The screenshot shows the Altium Designer 21.6.1 interface. The 'Variant Choice' dialog is open, and the 'Outputs' table is visible. The 'Project Group 1.DsnWk' is expanded, showing the 'DC_motor_control.PrjPcb' project. The 'Outputs' table lists various output types, including 'Schematic Prints'. A red arrow labeled '1' points to the 'Schematic Prints' row. A red arrow labeled '2' points to the 'IECA-DCMD-01-SCH.SchDoc' file. A red arrow labeled '3' points to the 'Add New Output Container' button. A red arrow labeled '4' points to the 'New PDF' button in the 'Add New Output Container' dialog.

Variant Choice
This choice of Variant only applies to outputs generated from within this view. For release of PCB configuratic variant usage is always driven by the choice of variants in the PCB configuration.

- Choose a single variant for the whole output/job file [No Variations]
- Choose a different variant for each output

This variant will be only used when generating outputs and printing hardcopies from here. If this output job is used for release of PCB configurations then this choice will be ignored anariant/parameter information will be passed to this output job by the PCB Release manager.

Name	Data Source	Output Description	Enabled
Click Der.			
Composite			
PCB 3D Print			
PCB 3D Video			
PCB Prints			
Draftsman			
PCBLIB Prints			
PDF3D			
PDF3D MBA			
Report Prints			
Schematic Prints			
SimView Prints			

Outputs

Output Container

[Add New Output Container]

New PDF

New Video

New Folder Structure

Hard Copy

Job Name

Print Job

Printer Default

[Add New Print Job]

Vincular archivo con su PDF

The screenshot shows the Altium Designer (21.6.1) interface. The 'Variant Choice' dialog is open, displaying options for choosing a single variant for the whole output job file or a different variant for each output. The 'Outputs' table is visible, showing various output types and their descriptions. A red arrow labeled '1' points to the 'Schematic' output, and a red arrow labeled '2' points to the 'Schematic Prints' output.

Name	Data Source	Output Description	Enabled
Netlist Outputs			
Simulator Outputs			
Documentation Outputs			
Schematic Prints	IECA-DCMD-01-SCH.SchDoc	Schematic Prints	1
Assembly Outputs			
Fabrication Outputs			
Report Outputs			
Validation Outputs			
Export Outputs			
PostProcess Outputs			

Archivo de Dibujo de Fabricación, Ensamble y Configuración

The screenshot shows the Altium Designer 21.6.1 interface. The 'Variant Choice' dialog is open, displaying two options: 'Choose a single variant for the whole output job file' (selected) and 'Choose a different variant for each output'. The 'Outputs' list is visible, showing various output types like 'Draftsman', 'PCB 3D Print', 'PDF3D', etc. Red arrows and numbers 1, 2, 3, and 4 indicate the steps to create a new output container and select a variant.

- Click on the 'Add New Output Container' button in the 'Outputs' list.
- Click on the 'Draftsman' option in the 'Add New Output Container' dropdown menu.
- Click on the 'New PDF' button in the 'Add New Output Container' dropdown menu.
- Click on the 'New Folder Structure' button in the 'Add New Output Container' dropdown menu.

Vincular archivo con su PDF

DC_motor_control.PrjPcb - Altium Designer (21.6.1)

File Edit View Project Tools Window Help

Projects Home Page Job1.OutJob * CFG.PCBDwf ASSY.PCBDwf FAB.PCBDwf

Search

Project Group 1.DsnWrk

- Pmod_CAN.PrjPcb
- Pmod_ESP32.PrjPcb
- Pmod_I2S2.PrjPcb
- DC_motor_control.PrjPcb *
- Source Documents
 - [1] IECA-DCMD-01-SCH.SchDoc
 - IECA-DCMD-01-PCB.PcbDoc
 - FAB.PCBDwf
 - ASSY.PCBDwf
 - CFG.PCBDwf
- Settings
 - Output Job Files
 - Job1.OutJob *
 - Libraries
 - Generated
- PMOD_ACL2.PrjPcb

Variant Choice

This choice of Variant only applies to outputs generated from within this view. For release of PCB configurations, variant usage is always driven by the choice of variants in the PCB configuration

☒ Choose a single variant for the whole outputjob file [No Variations]

☐ Choose a different variant for each output

This variant will be only used when generating outputs and printing hardcopies from here.
If this output job is used for release of PCB configurations then this choice will be ignored and variant/parameter information will be passed to this output job by the PCB Release manager.

Name	Data Source	Output Description	Enabled
Netlist Outputs			
[Add New Netlist ...]			
Simulator Outputs			
[Add New Simulat ...]			
Documentation Outp...			
Draftsman	FAB.PCBDwf	Draftsman	<input checked="" type="checkbox"/>
Copy of Draftsman ASSY.PCBDwf		Draftsman	<input checked="" type="checkbox"/>
Copy of Copy of D CFG.PCBDwf		Draftsman	<input checked="" type="checkbox"/>
[Add New Docum...			
Assembly Outputs			
[Add New Assemb ...]			
Fabrication Outputs			
[Add New Fabrica ...]			
Report Outputs			
[Add New Report ...]			
Validation Outputs			
[Add New Validati ...]			
Export Outputs			
[Add New Export ...]			
PostProcess Outputs			
[Add New PostPro ...]			

Converters

FAB [Change](#) or [Remove](#)

Release managemen
Path in server
Local Path

[Generate content →](#)
[Generate and publish →](#)

ASSY [Change](#) or [Remove](#)

Release managemen
Path in server

[Generate content →](#)

Hard Copy

Job Name

[Add New Print Job]

Projects Navigator

Panels

The screenshot displays the Altium Designer (21.6.1) interface. The top menu bar includes File, Edit, View, Project, Tools, Window, and Help. The Projects panel on the left shows a project tree with 'DC_motor_control.PrjPcb' selected. The main workspace shows the 'Variant Choice' dialog, which is titled 'Variant Choice' and contains the following text: 'This choice of Variant only applies to outputs generated from within this view. For release of PCB configurations, variant usage is always driven by the choice of variants in the PCB configuration'. Below this text are two radio buttons: 'Choose a single variant for the whole output job file' (selected) and 'Choose a different variant for each output'. A dropdown menu next to the first radio button shows '[No Variations]'. Below the radio buttons is a paragraph: 'This variant will be only used when generating outputs and printing hardcopies from here. If this output job is used for release of PCB configurations then this choice will be ignored and variant/parameter information will be passed to this output job by the PCB Release manager.' The 'Outputs' panel is visible, showing a list of outputs with columns for Name, Data Source, Output Description, and Enabled. A context menu is open over the 'PDF3D' output, showing options like 'Composite', 'PCB 3D Print', 'PCB 3D Video', 'PCB Prints', 'Draftsman', 'PCBLIB Prints', 'PDF3D', 'PDF3D M8A', 'Report Prints', 'Schematic Prints', and 'SimView Prints'. The 'PDF3D' option is selected, and a sub-menu is open showing 'IECA-DCMD-01-PCB.PcbDoc'. The 'Print Job' section is also visible, showing a 'Printer' set to 'Default'.

Vincular archivo con su PDF

DC_motor_control.PrjPcb - Altium Designer (21.6.1)

File Edit View Project Tools Window Help

Projects

Search

Project Group 1.DsnWrk

- Pmod_CAN.PrjPcb
- Pmod_ESP32.PrjPcb
- Pmod_I2S2.PrjPcb
- DC_motor_control.PrjPcb *
- Source Documents
 - [1] IECA-DCMD-01-SCH.SchDoc
 - FAB.PCBDwf
 - ASSY.PCBDwf
 - CFG.PCBDwf
- Settings
 - Output Job Files
 - Job1.OutJob *
 - Libraries
 - Generated
- Pmod_ACL2.PrjPcb

Variant Choice

This choice of Variant only applies to outputs generated from within this view. For release of PCB configurations, variant usage is always driven by the choice of variants in the PCB configuration

☒ Choose a single variant for the whole output/job file [No Variations]

☐ Choose a different variant for each output

This variant will be only used when generating outputs and printing hardcopies from here. If this output job is used for release of PCB configurations then this choice will be ignored and variant/parameter information will be passed to this output job by the PCB Release manager.

Name	Data Source	Output Description	Enabled
Netlist Outputs			
Simulator Outputs			
Documentation Outputs			
PDF3D	IECA-DCMD-01-PCB.PcbDoc	PDF3D	1
Assembly Outputs			
Fabrication Outputs			
Report Outputs			
Validation Outputs			
Export Outputs			
PostProcess Outputs			

Outputs

3D

Release manage

Path in server

Local Path

Generate content →

Generate and publish →

[Add New Output Container]

Hard Copy





















Job Name

[Add New Print Job]

Projects Navigator

Panels

Generación De Documentos De Fabricación

Nombre	Fecha de modificación	Tipo	Tamaño
 IECA-ESP32-01-PCB-3D STEP.step	29/09/2021 09:20 a. m.	Archivo STEP	3,345 KB
 IECA-ESP32-01-PCB-3D.PDF	29/09/2021 09:15 a. m.	Adobe Acrobat D...	802 KB
 IECA-ESP32-01-PCB-ART(01).GTO	29/09/2021 09:18 a. m.	CAMtastic Top Ov...	20 KB
 IECA-ESP32-01-PCB-ART(02).GTP	29/09/2021 09:18 a. m.	CAMtastic Top Pa...	11 KB
 IECA-ESP32-01-PCB-ART(03).GTS	29/09/2021 09:18 a. m.	CAMtastic Top Sol...	13 KB
 IECA-ESP32-01-PCB-ART(04).GTL	29/09/2021 09:18 a. m.	CAMtastic Top La...	62 KB
 IECA-ESP32-01-PCB-ART(05).GBL	29/09/2021 09:18 a. m.	CAMtastic Botto...	48 KB
 IECA-ESP32-01-PCB-ART(06).GBS	29/09/2021 09:18 a. m.	CAMtastic Botto...	11 KB
 IECA-ESP32-01-PCB-ART(07).GBP	29/09/2021 09:18 a. m.	CAMtastic Botto...	9 KB
 IECA-ESP32-01-PCB-ART(08).GBO	29/09/2021 09:18 a. m.	CAMtastic Botto...	144 KB
 IECA-ESP32-01-PCB-ART(09).GM2	29/09/2021 09:18 a. m.	CAMtastic Mecha...	9 KB
 IECA-ESP32-01-PCB-ASSY.PDF	29/09/2021 09:24 a. m.	Adobe Acrobat D...	176 KB
 IECA-ESP32-01-PCB-BoardEdgeRout.TXT	29/09/2021 09:17 a. m.	Documento de te...	1 KB
 IECA-ESP32-01-PCB-BOM.xlsx	20/09/2021 02:21 p. m.	Hoja de cálculo d...	13 KB
 IECA-ESP32-01-PCB-CFG.PDF	29/09/2021 09:25 a. m.	Adobe Acrobat D...	80 KB
 IECA-ESP32-01-PCB-FAB.PDF	29/09/2021 09:20 a. m.	Adobe Acrobat D...	194 KB
 IECA-ESP32-01-PCB-Neutral File.ipc	29/09/2021 09:17 a. m.	Archivo IPC	45 KB
 IECA-ESP32-01-PCB-PICK PLACE.txt	29/09/2021 09:20 a. m.	Documento de te...	6 KB
 IECA-ESP32-01-PCB-Plated.TXT	29/09/2021 09:17 a. m.	Documento de te...	1 KB
 IECA-ESP32-01-PCB-Test Point.csv	29/09/2021 09:17 a. m.	Archivo de valores...	7 KB