

GUANAJUATO
Gobierno del Estado

DISEÑO DE UN PCB EN EL SOFTWARE ALTIUM DESIGNER

IECA®
Instituto Estatal de Capacitación

ieca.guanajuato.gob.mx

   @IECAGuanajuato

FLUJO DE DISEÑO DE PCB's

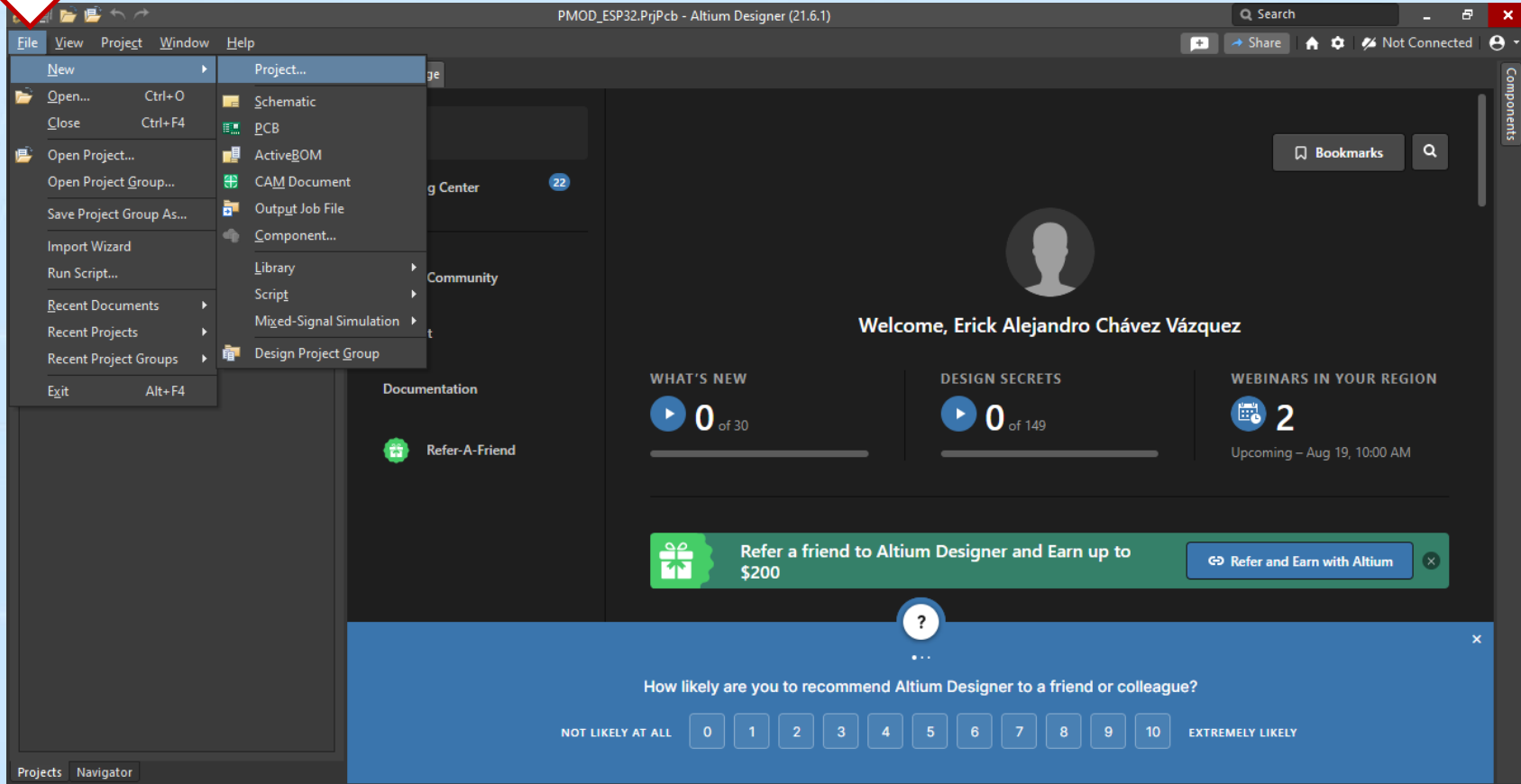


FLUJO DE DISEÑO EN ALTIUM DESIGNER

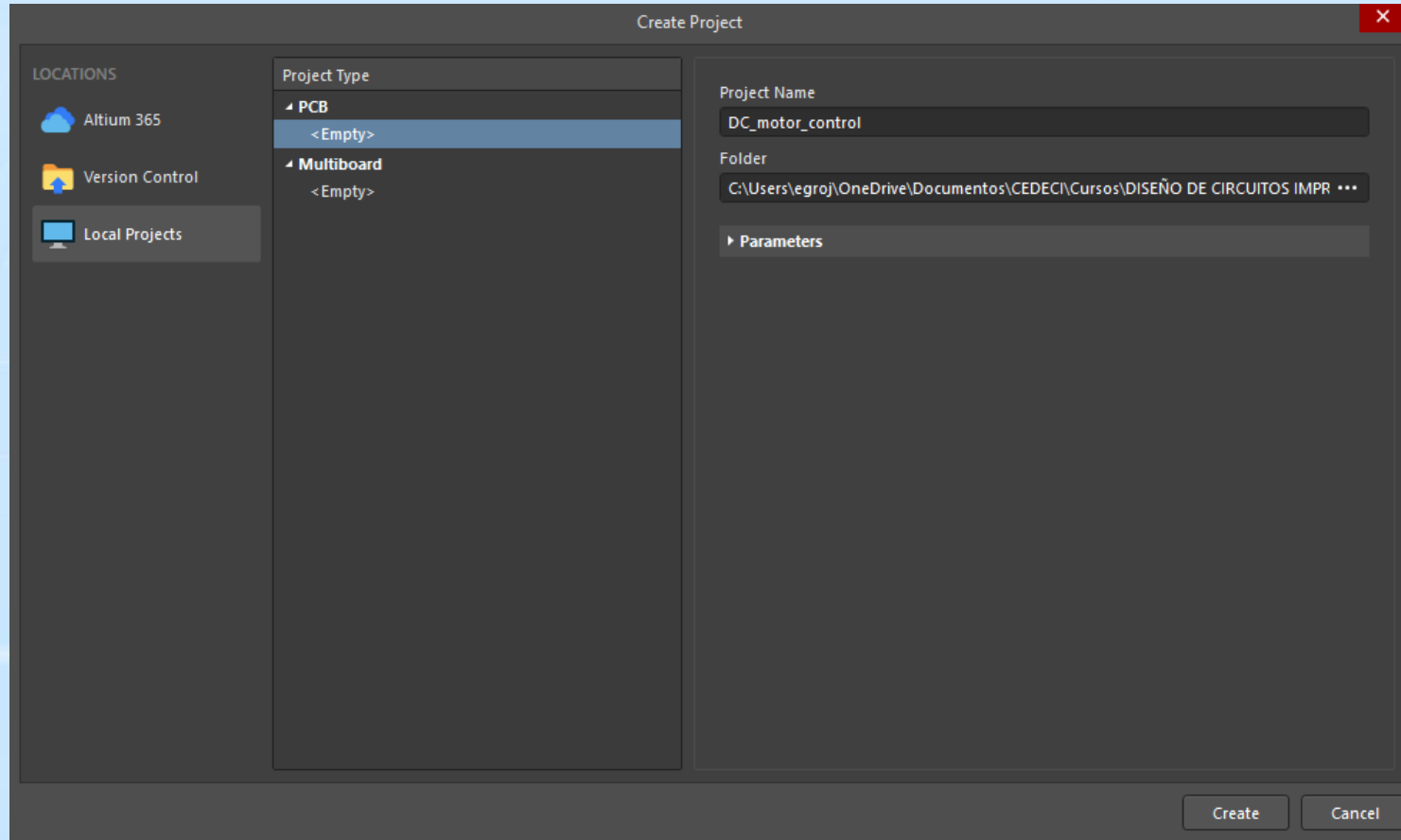


Click

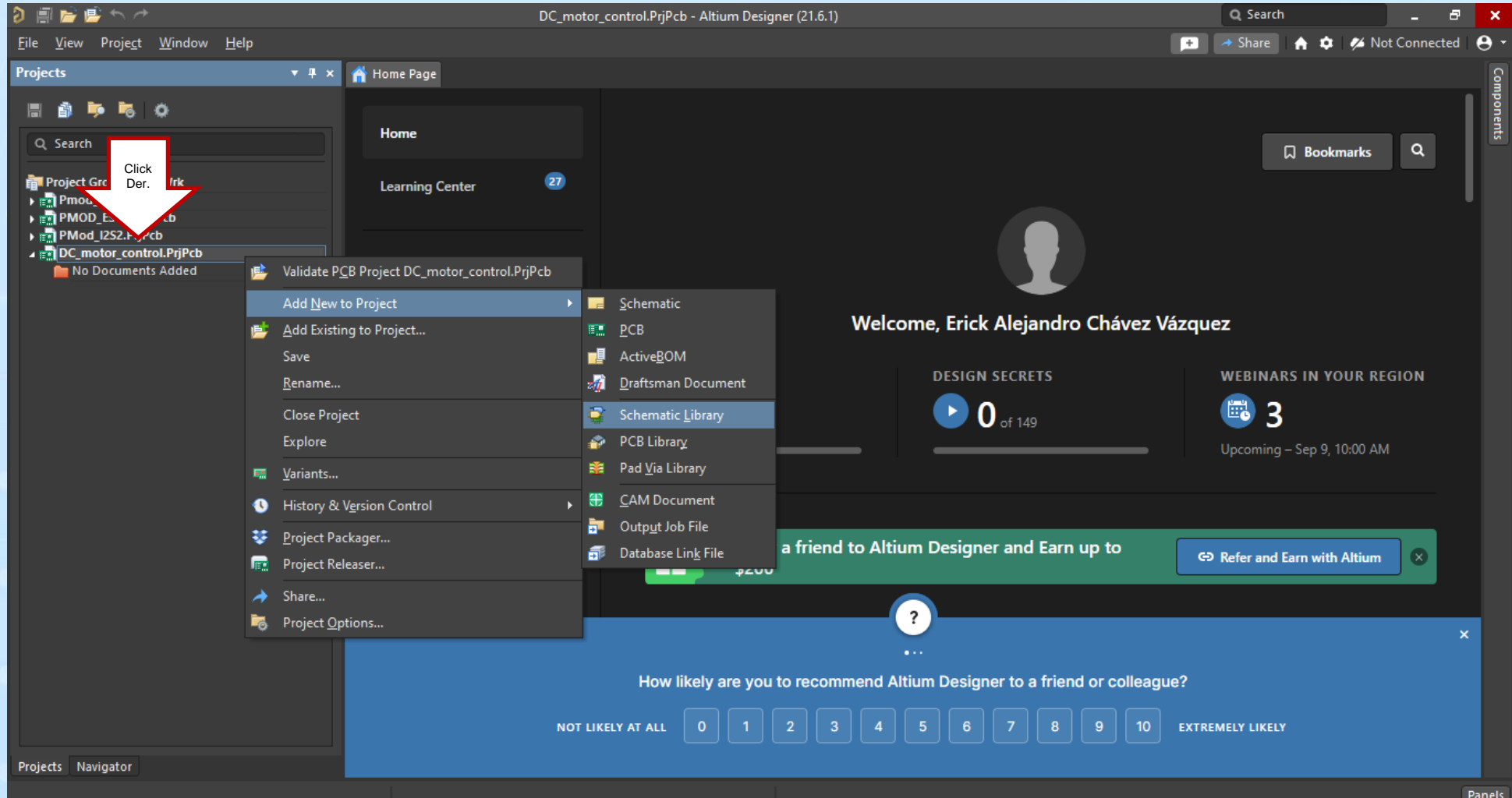
Crear un nuevo Proyecto



Asignación de nombre al proyecto y ubicación de los archivos



Creación de librería de símbolos



Altium Designer (21.6.1) interface showing the 'Projects' panel and a context menu for 'DC_motor_control.PrjPcb'.

The 'Projects' panel on the left shows a list of projects, including 'DC_motor_control.PrjPcb'. A red arrow points to the 'Click Der.' button in the 'Projects' panel.

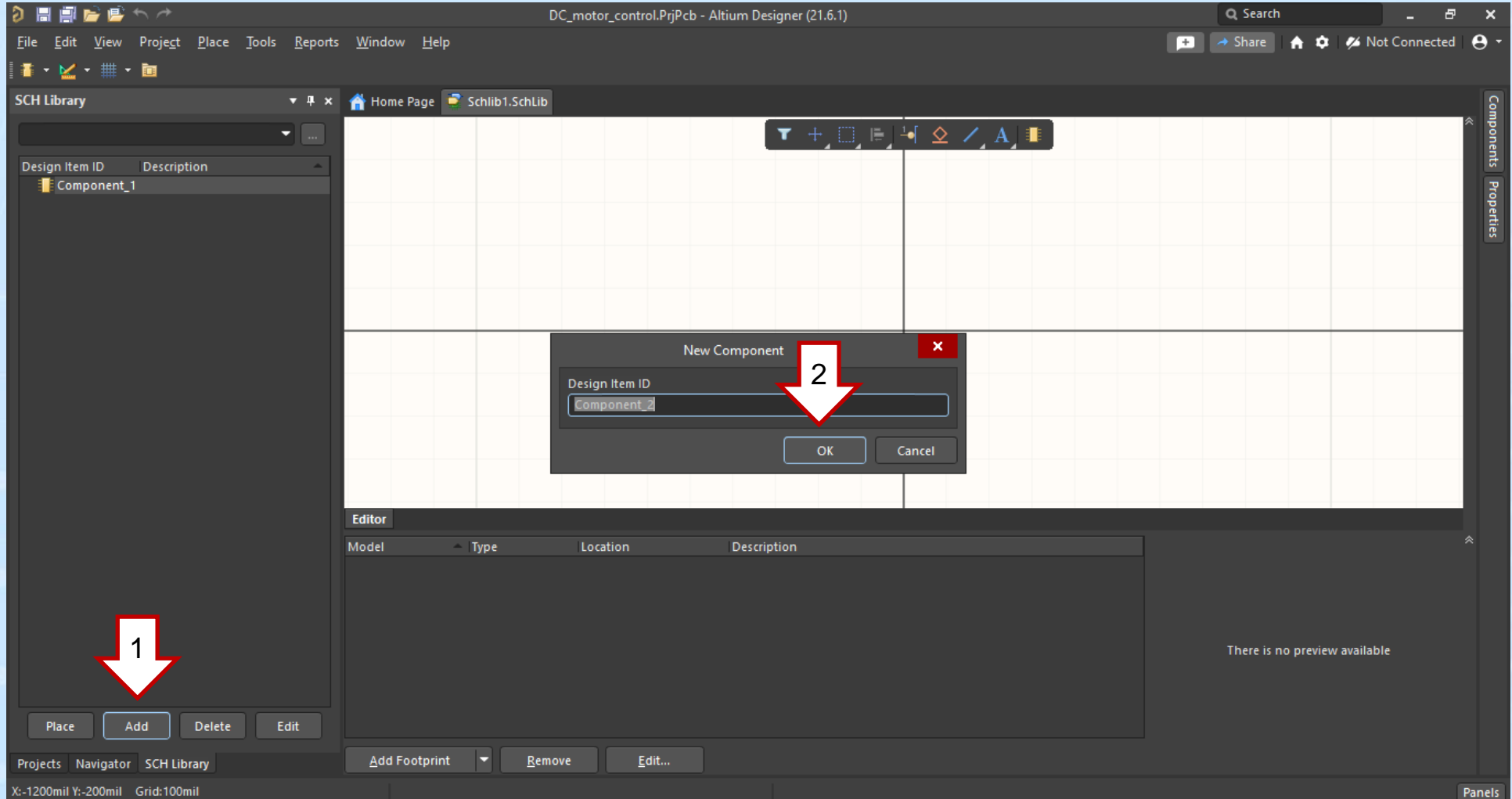
The context menu for 'DC_motor_control.PrjPcb' is open, showing options:

- Validate PCB Project DC_motor_control.PrjPcb
- Add New to Project
 - Schematic
 - PCB
 - ActiveBOM
 - Draftsman Document
 - Schematic Library
 - PCB Library
 - Pad Via Library
 - CAM Document
 - Output Job File
 - Database Link File
- Add Existing to Project...
- Save
- Rename...
- Close Project
- Explore
- Variants...
- History & Version Control
- Project Packager...
- Project Releaser...
- Share...
- Project Options...

The main workspace displays a 'Welcome, Erick Alejandro Chávez Vázquez' message and a 'DESIGN SECRETS' section showing 0 of 149 secrets. A 'WEBINARS IN YOUR REGION' section shows 3 upcoming webinars on Sep 9, 10:00 AM.

A survey at the bottom asks: 'How likely are you to recommend Altium Designer to a friend or colleague?' with a scale from 0 (NOT LIKELY AT ALL) to 10 (EXTREMELY LIKELY).

Pasos para la creación de un símbolo



The screenshot shows the Altium Designer (21.6.1) interface. The main workspace is a grid. A 'New Component' dialog box is open in the center, with a red arrow labeled '2' pointing to the 'Design Item ID' field, which contains 'Component_2'. Below the dialog, the 'SCH Library' panel is visible on the left, showing a table with one row: 'Component_1'. A red arrow labeled '1' points to the 'Add' button at the bottom of this panel. The 'Editor' panel at the bottom right shows a table with columns: Model, Type, Location, and Description. The status bar at the bottom indicates 'X: 1200mil Y: 200mil Grid: 100mil'.

DC_motor_control.PrjPcb - Altium Designer (21.6.1)

File Edit View Project Place Tools Reports Window Help

SCH Library

Design Item ID Description

Design Item ID	Description
Component_1	

Place Add Delete Edit

Projects Navigator SCH Library

Add Footprint Remove Edit...

X: 1200mil Y: 200mil Grid: 100mil

There is no preview available

Components Properties

Search

Share Not Connected

New Component

Design Item ID

Component_2

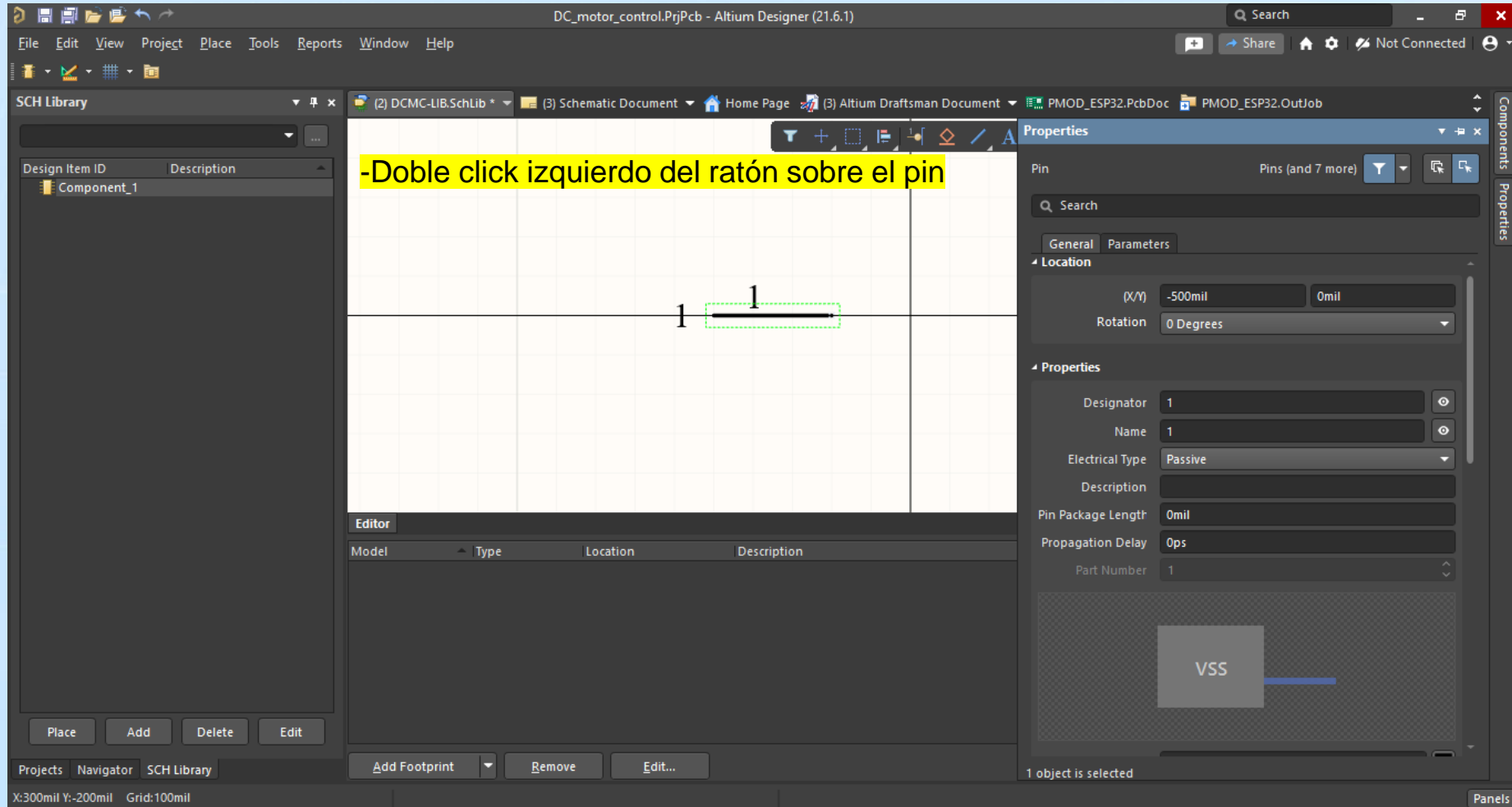
OK Cancel

2

1

The screenshot displays the Altium Designer (21.6.1) interface. The main workspace is a grid-based area for schematic capture. On the left, the 'SCH Library' panel shows a table with columns 'Design Item ID' and 'Description', containing one entry: 'Component_1'. Below this is a 'Place' button and a list of actions: 'Add', 'Delete', and 'Edit'. The top menu bar includes 'File', 'Edit', 'View', 'Project', 'Place', 'Tools', 'Reports', 'Window', and 'Help'. The 'Place' menu is open, showing options like 'Pin', 'Arc', 'Full Circle', 'Elliptical Arc', 'Ellipse', 'Line', 'Rectangle', 'Round Rectangle', 'Polygon', 'Bezier', 'Text String', 'Text Frame', and 'Graphic...'. A secondary context menu is also open, showing 'Find Similar Objects...', 'Clear Filter', 'Shift+C', 'Place', 'Tools', 'View', 'Cut', 'Copy', 'Paste', 'Preferences...', and 'Supplier Links...'. The bottom status bar shows 'X: 1300mil Y: 400mil Grid: 100mil'.

Creación de un PIN



The screenshot displays the Altium Designer (21.6.1) interface for a project named "DC_motor_control.PrjPcb". The main workspace shows a schematic document with a grid. A horizontal line representing a pin is centered on the grid, with a green dashed selection box around it. The number "1" is placed at both ends of the line. A yellow text box with the instruction "-Doble click izquierdo del ratón sobre el pin" is overlaid on the top left of the workspace.

SCH Library

Design Item ID	Description
Component_1	

Properties

Pin Pins (and 7 more)

General Parameters

Location

(X/Y) -500mil 0mil
Rotation 0 Degrees

Properties

Designator 1
Name 1
Electrical Type Passive
Description
Pin Package Length 0mil
Propagation Delay 0ps
Part Number 1

Editor

Model	Type	Location	Description
-------	------	----------	-------------

VSS

1 object is selected

Place Add Delete Edit

Projects Navigator SCH Library

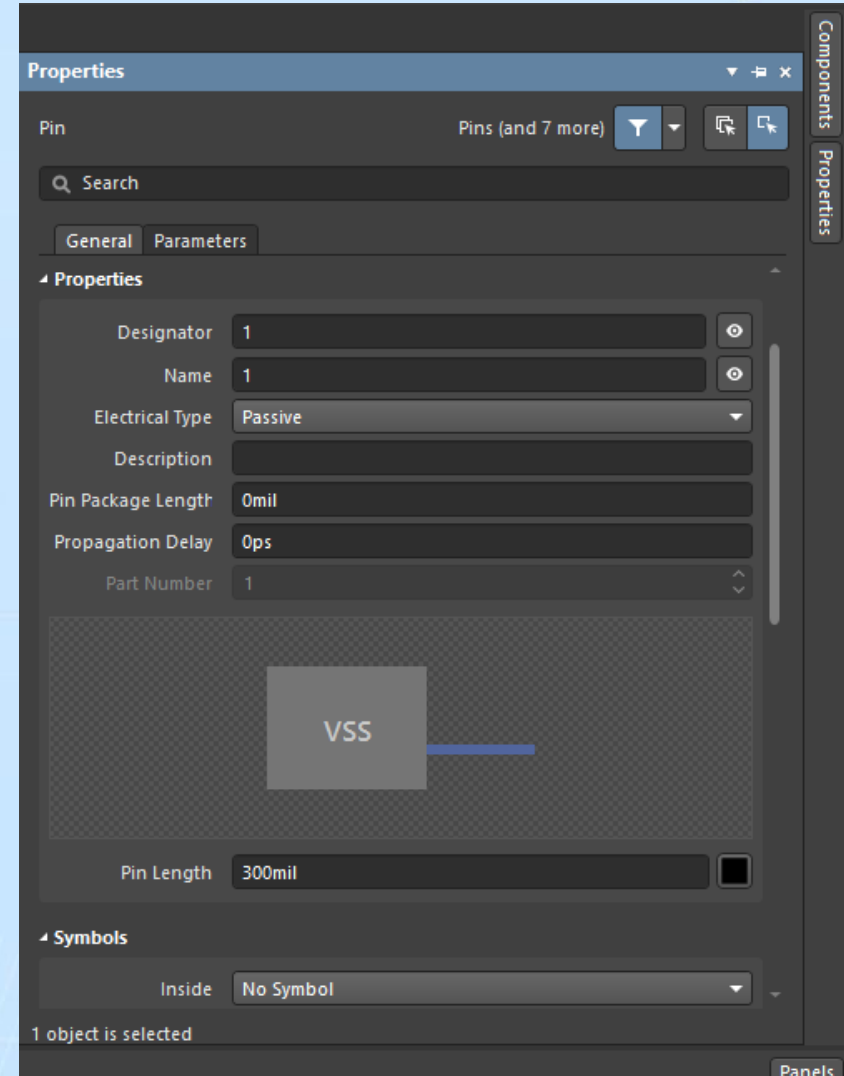
Add Footprint Remove Edit...

X:300mil Y:200mil Grid:100mil

Panels

Propiedades de un PIN

- Designator
- Name
- Electrical Type:
 - Passive
 - Input
 - Output
- Description
- Pin Package Length
- Propagation Delay
- Pin Length
- Symbols:
 - Inside
 - Inside Edge
 - Outside Edge



The screenshot shows the 'Properties' panel for a 'Pin' component. The panel has tabs for 'General' and 'Parameters'. The 'General' tab is active, showing the following properties:

- Designator:** 1
- Name:** 1
- Electrical Type:** Passive
- Description:**
- Pin Package Length:** 0mil
- Propagation Delay:** 0ps
- Part Number:** 1

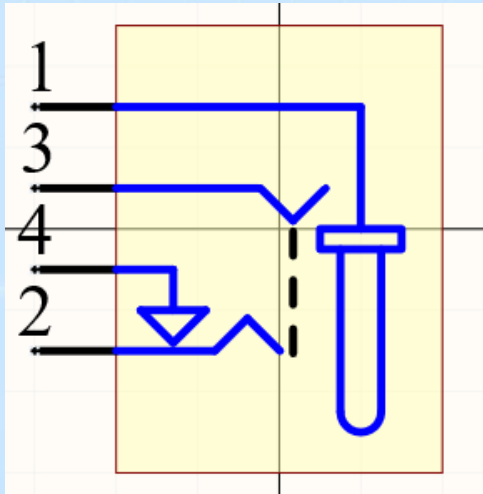
Below the properties is a visual representation of the pin, showing a rectangular pad labeled 'VSS' with a horizontal line extending from its right side. Below this visual is the **Pin Length** property, set to 300mil.

At the bottom of the panel is the **Symbols** section, which includes a dropdown menu for 'Inside' set to 'No Symbol'. The status bar at the bottom indicates '1 object is selected'.

Creación de un Símbolos

Contenido de un símbolo:

- Agregar pines
- Agregar dibujo del cuerpo del símbolo
- Agregar atributos del símbolo que ayudarán a crear el BOM



Properties

Component Pins (and 7 more)

Search

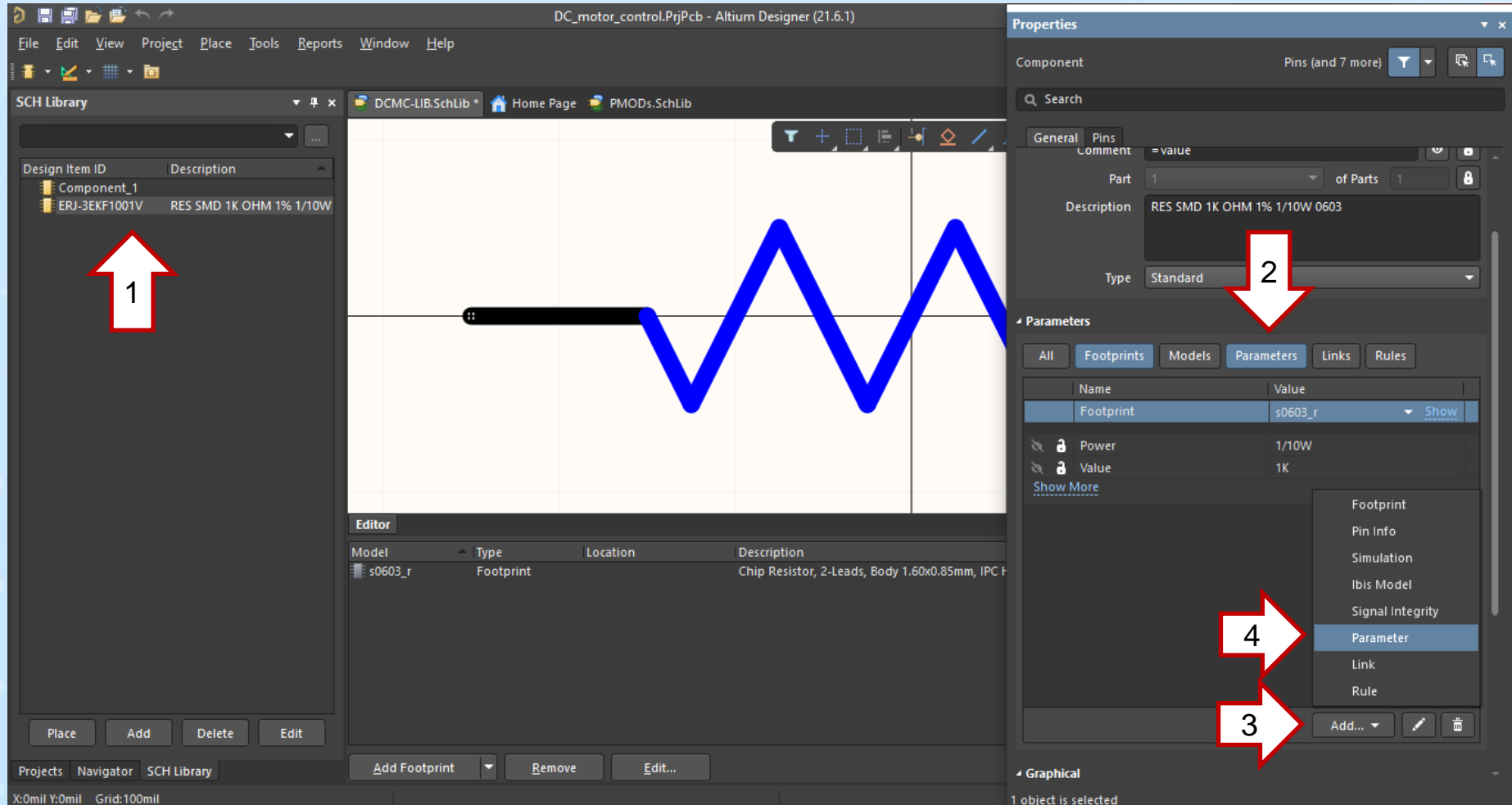
General Pins

Name	Value
Footprint	CONN SJ-3524-S... Show
Manufacturer	CUI Devices
Supplier 1	DigiKey
Supplier Part Number 1	CP-3524SJBECT-ND
Manufacturer Part Number	SJ-3524-SMT-TR-BE
Operating temperature	-25°C ~ 85°C
Mounting type	PTH
Package / Case	*
Type	Connectors
Lead free status / RoHS stat...	OK / OK
Case / EIA	*
Value	*
Power	1 A
Tolerance	*

[Show Less](#)

Add...

Propiedades de símbolos



DC_motor_control.PrjPcb - Altium Designer (21.6.1)

File Edit View Project Place Tools Reports Window Help

SCH Library

Design Item ID Description

Component_1

ERJ-3EKF1001V RES SMD 1K OHM 1% 1/10W

1

DCMC-LIB.SchLib Home Page PMODs.SchLib

Editor

Model	Type	Location	Description
s0603_r	Footprint		Chip Resistor, 2-Leads, Body 1.60x0.85mm, IPC

Properties

Component Pins (and 7 more)

Search

General Pins

Comment = value

Part 1 of Parts 1

Description RES SMD 1K OHM 1% 1/10W 0603

Type Standard

2

Parameters

All Footprints Models Parameters Links Rules

Name	Value
Footprint	s0603_r Show
Power	1/10W
Value	1K

Show More

Footprint

Pin Info

Simulation

Ibis Model

Signal Integrity

Parameter

Link

Rule

3

4

Graphical

1 object is selected