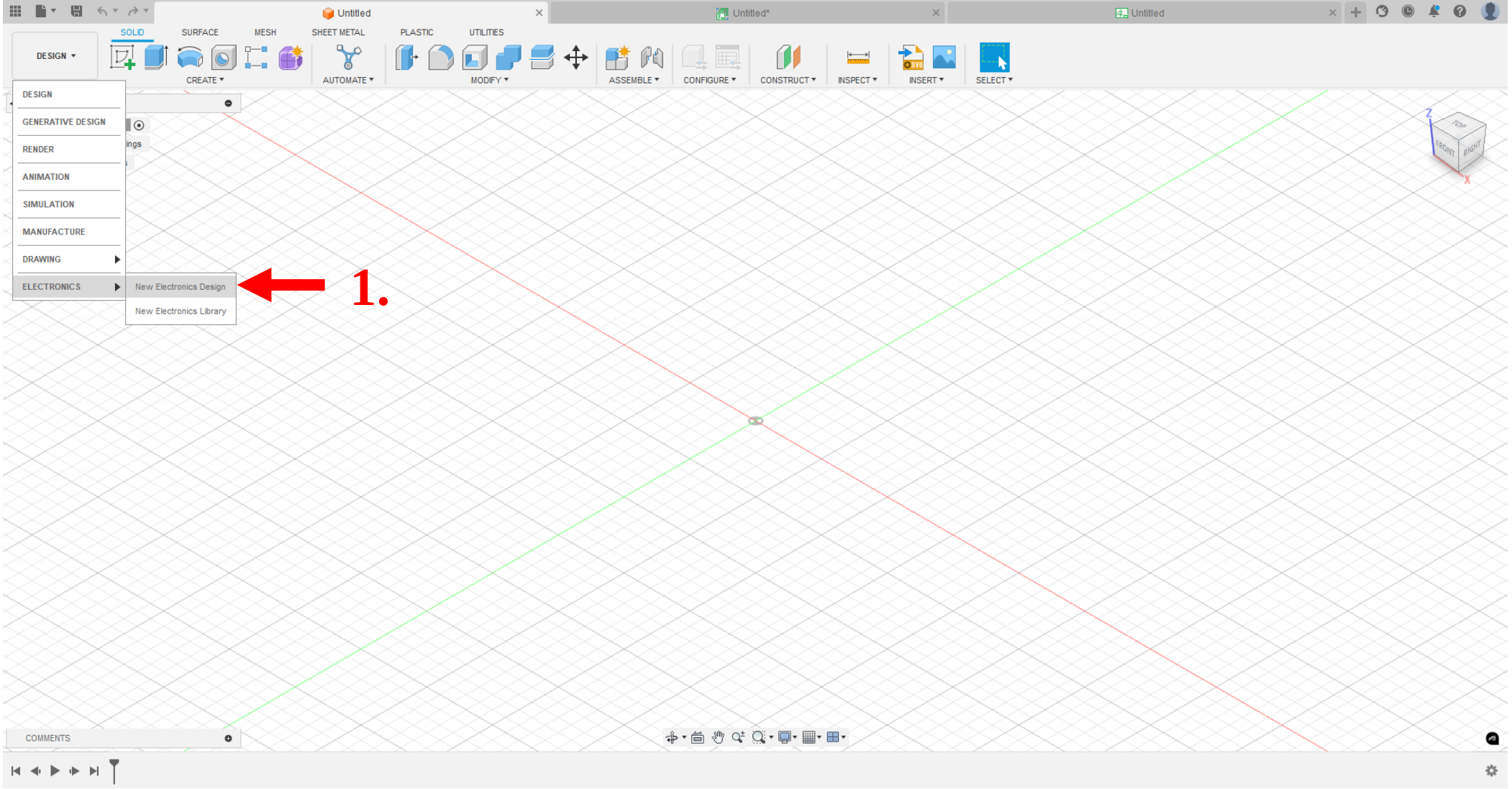
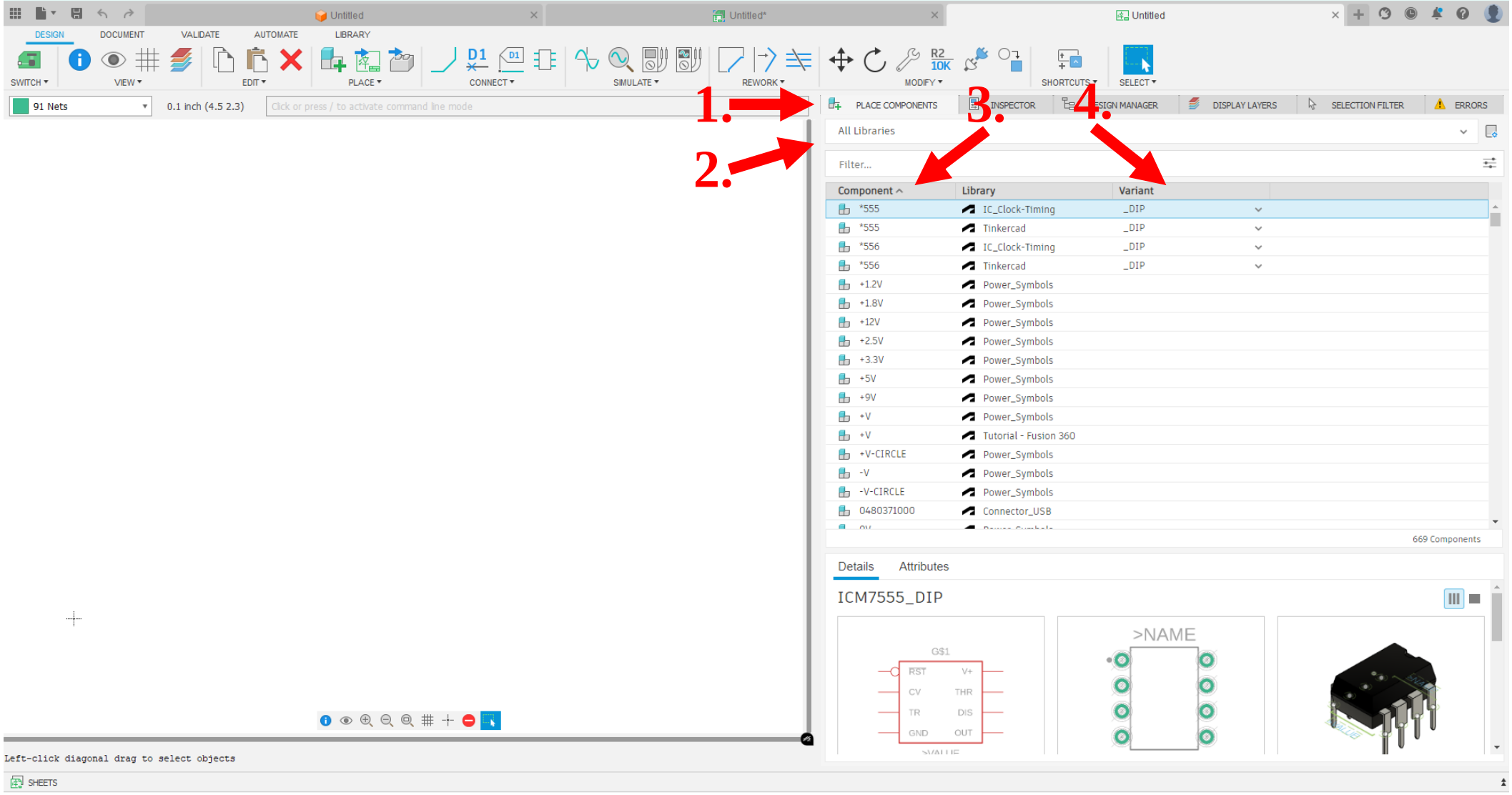
Introduction

Printed Circuit Board (PCB) design is a key stage in the development of electronic circuits. Similar to mechanics, where CAD models allow for the precise development of components, CAD systems for electronics allow PCB layouts to be designed, optimized and visualized in 3D space. Modern CAD tools for PCBs allow for the integration of electronic and mechanical models, which is particularly important in mechatronics, where electronic systems must be matched to the physical structure of the device. This allows you to detect potential collisions between electronic components and the housing early, optimize component placement, and prepare a production-ready model.

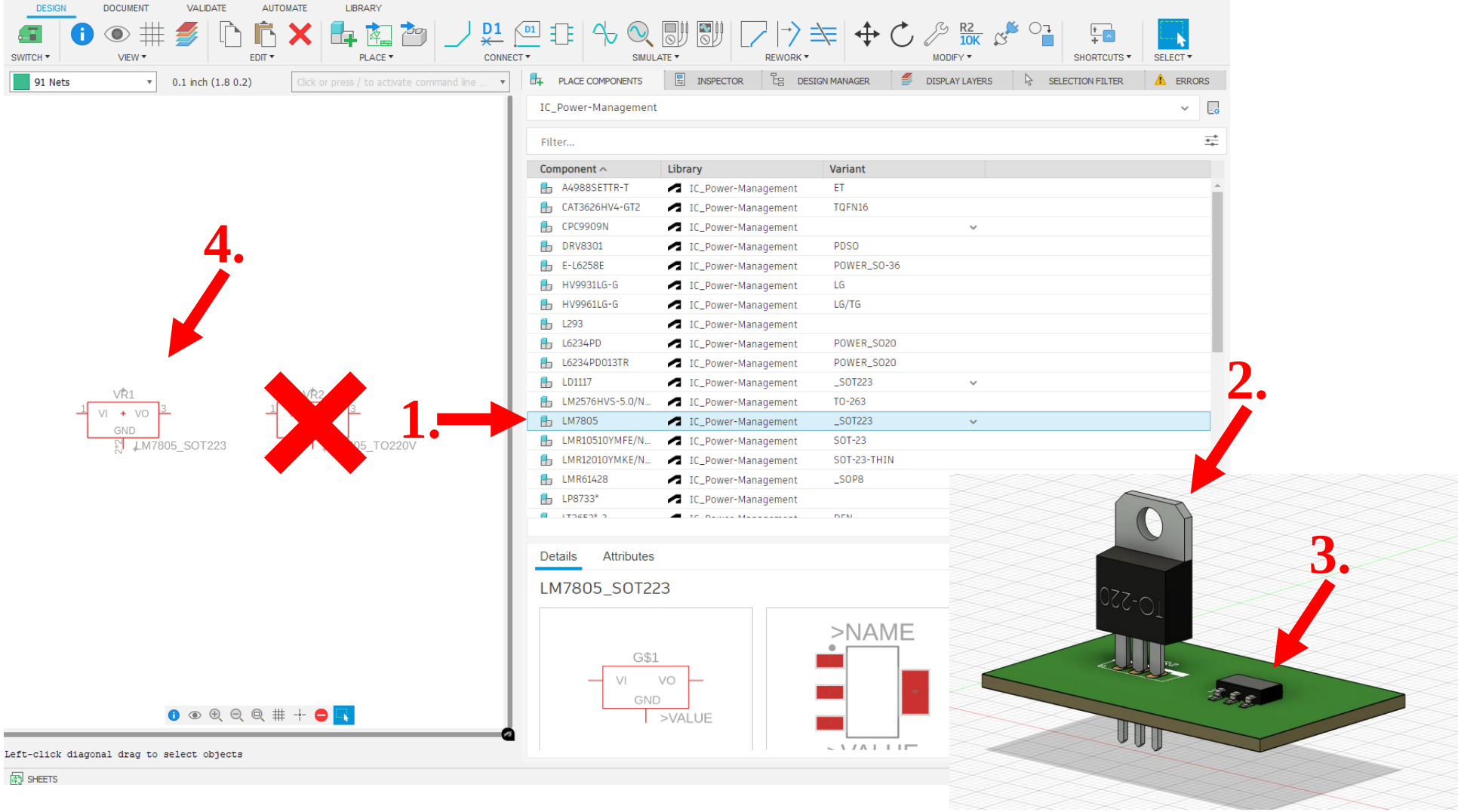
The purpose of this exercise is to learn about the PCB design process from scratch, including the placement of electronic components on the board and the preparation for integration with mechanical systems.



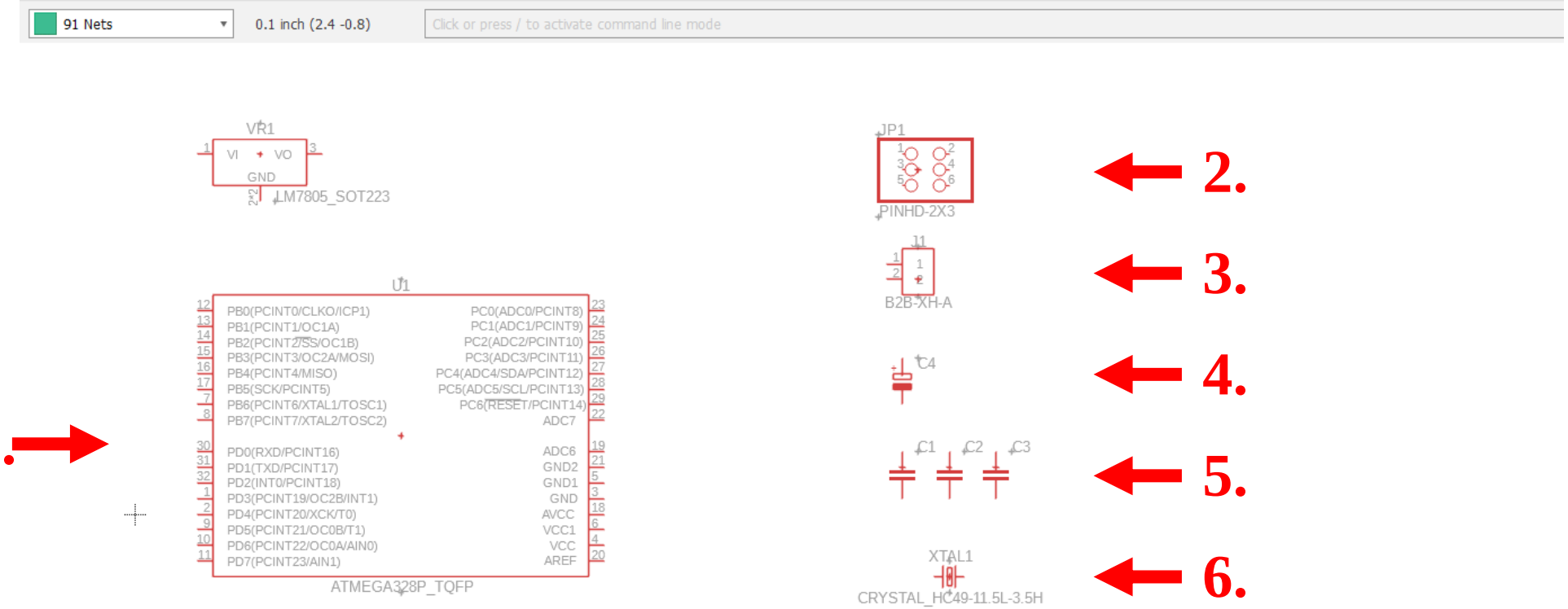
1. Start Autodesk Fusion and select New Electronics Design from Electronics mode



1. From the "place components" tab, it is possible to drag elements onto the electrical diagram
2. The drop-down list allows you to filter elements according to the library they come from.
3. The name of the components is in the first column.
4. Some of the elements with their own 3D models are available in several variants.

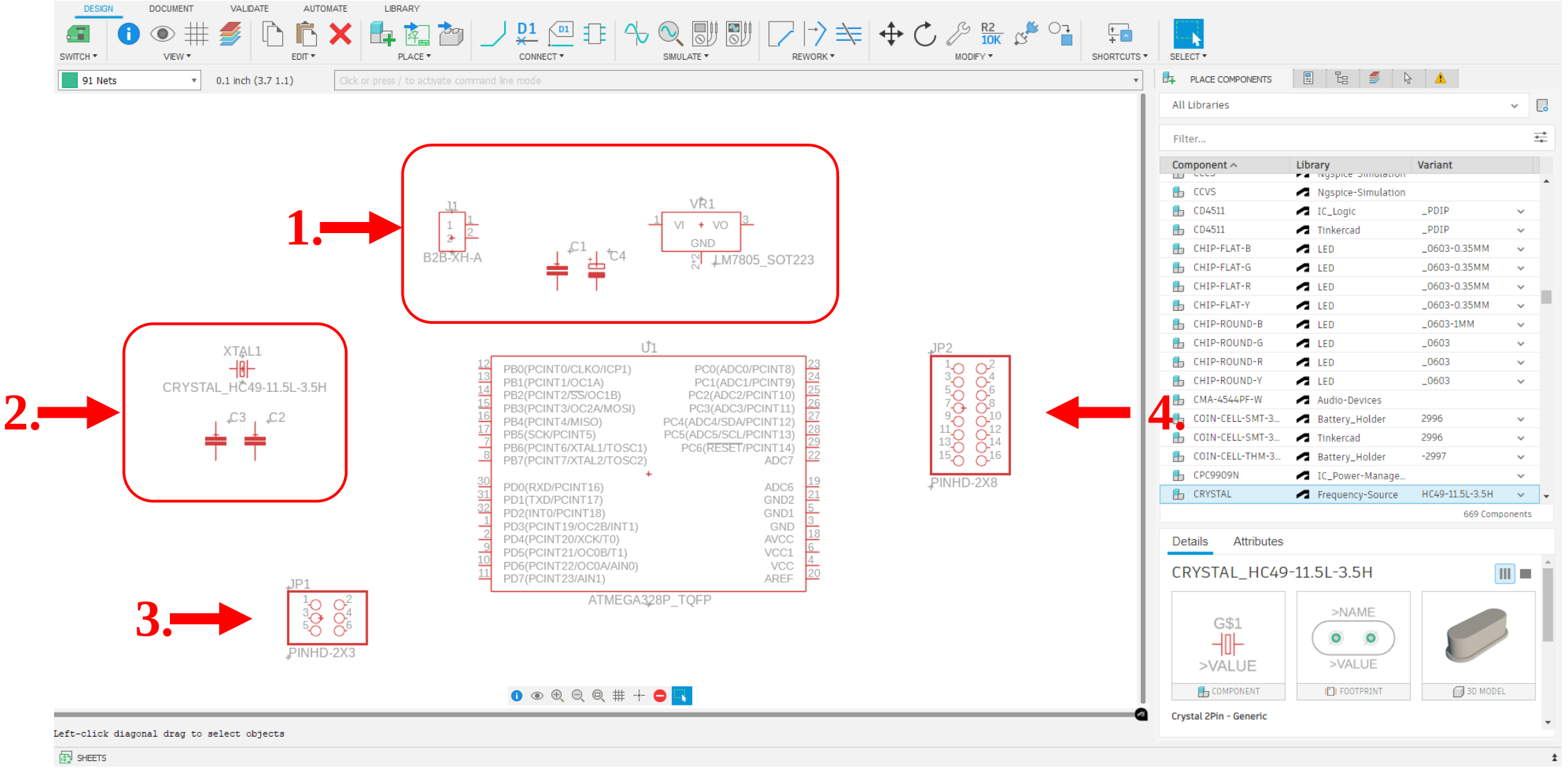


1. LM7805 voltage stabilizer is available in 2 variants:
2. TO-220 through-hole package (THT) – the legs of the system go through the board.
3. SOT-223 surface mount (SMD).
4. Only put the SMD variant in the schematic.



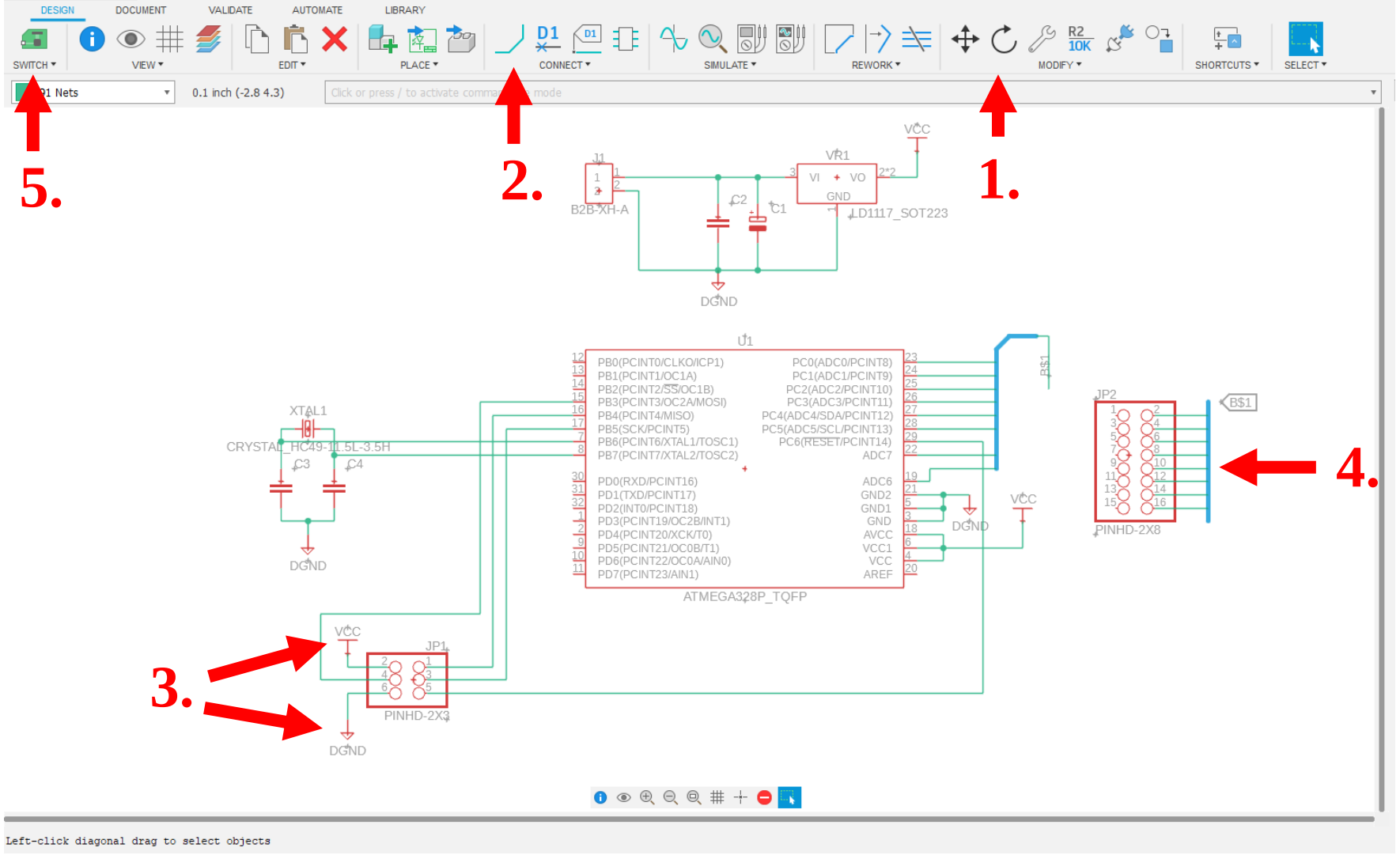
Add additional components:

1. Atmega328P variant of TQFP
2. 2x3 and 2x8 PINHD plugs
3. Power plug B2B-XH-A
4. Electrolytic capacitor in 11mm diameter housing
5. 3 capacitors type CHIP-0603
6. HC49-11.5L-3.5H Crystal Oscillator

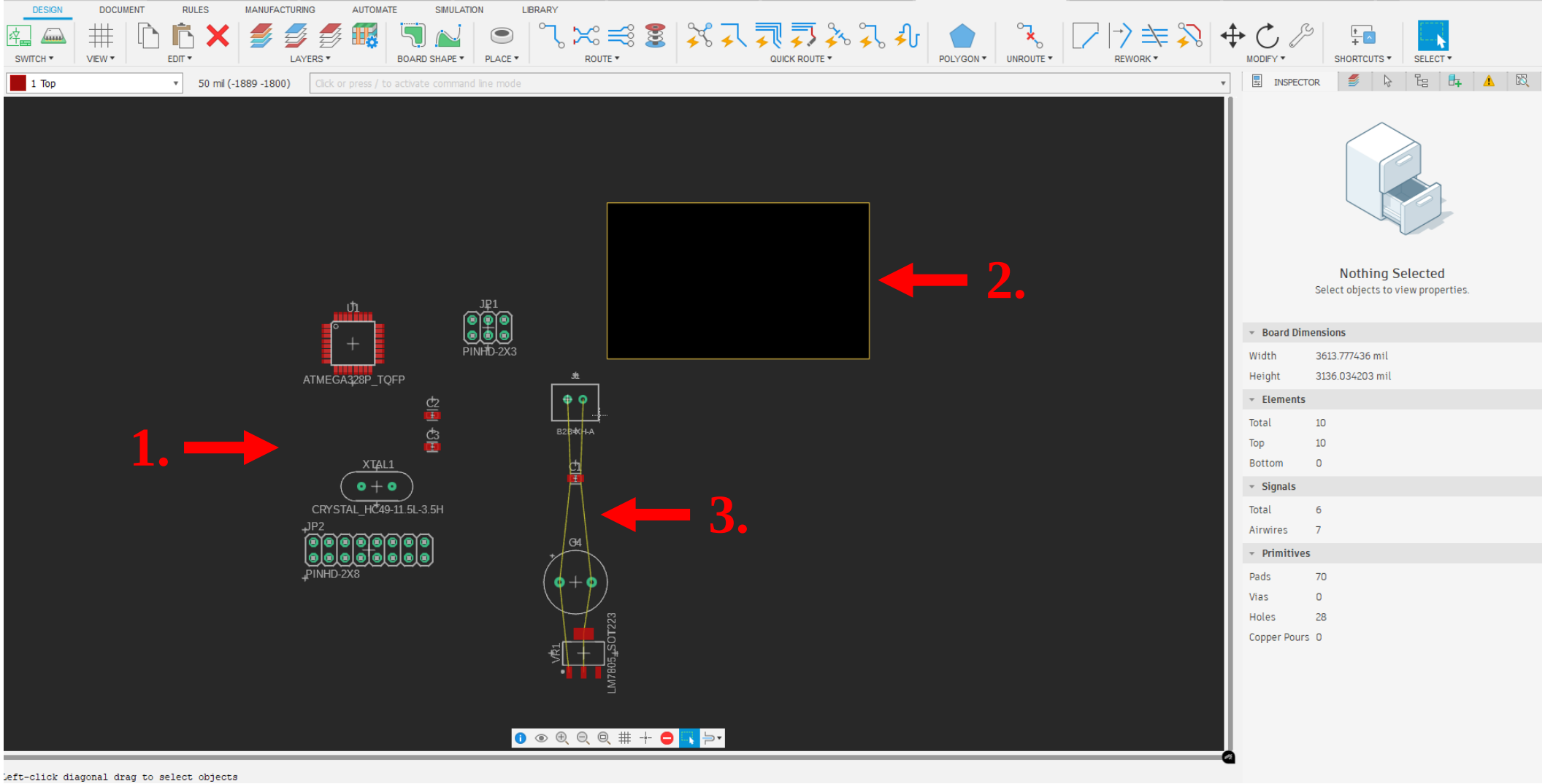


Pre-disassemble the components that make up the subassemblies:

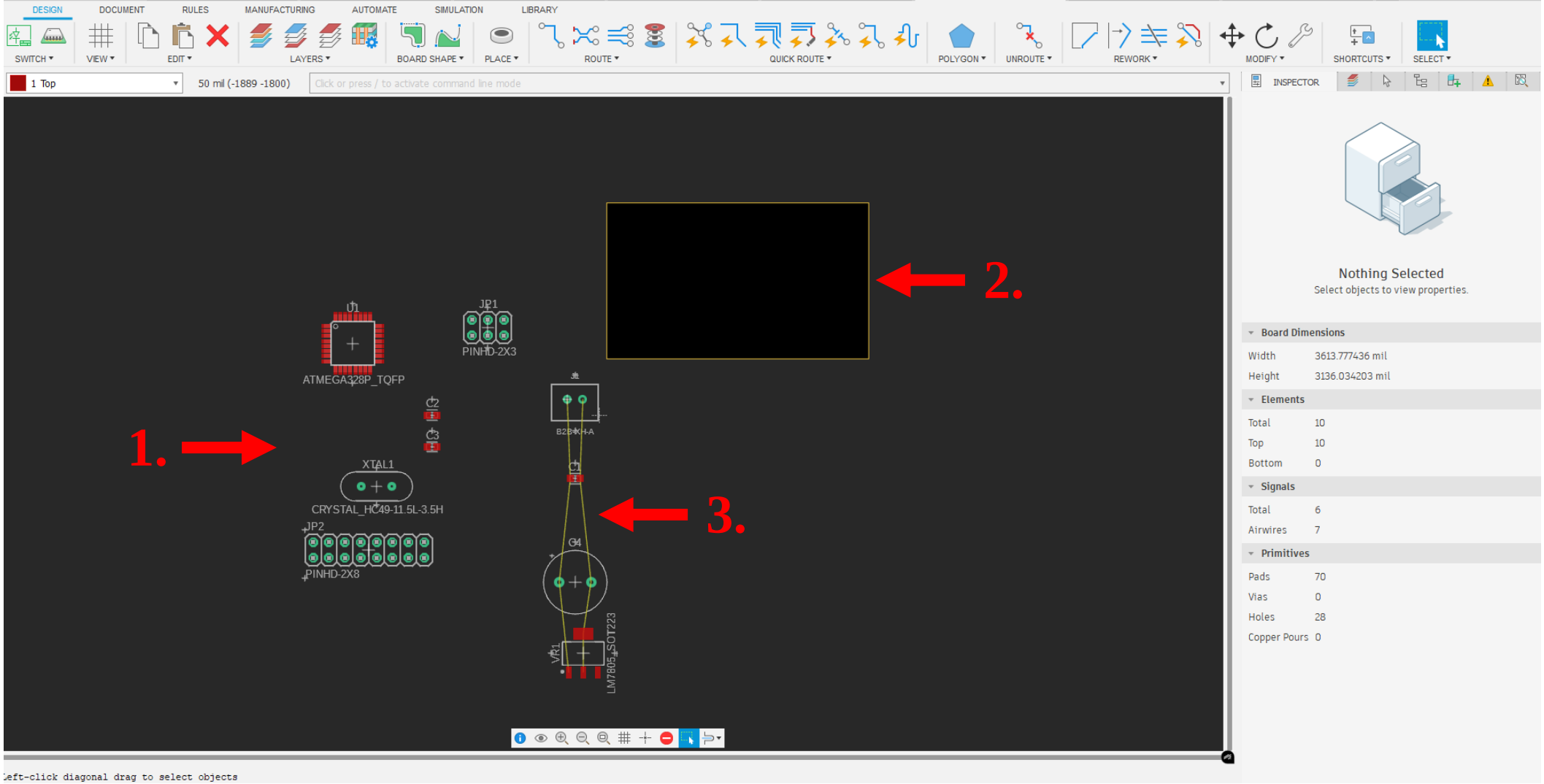
1. Power generation
2. Clock Signal Generation
3. ISP plug (for microcontroller programming)
4. Microcontroller Output Plug



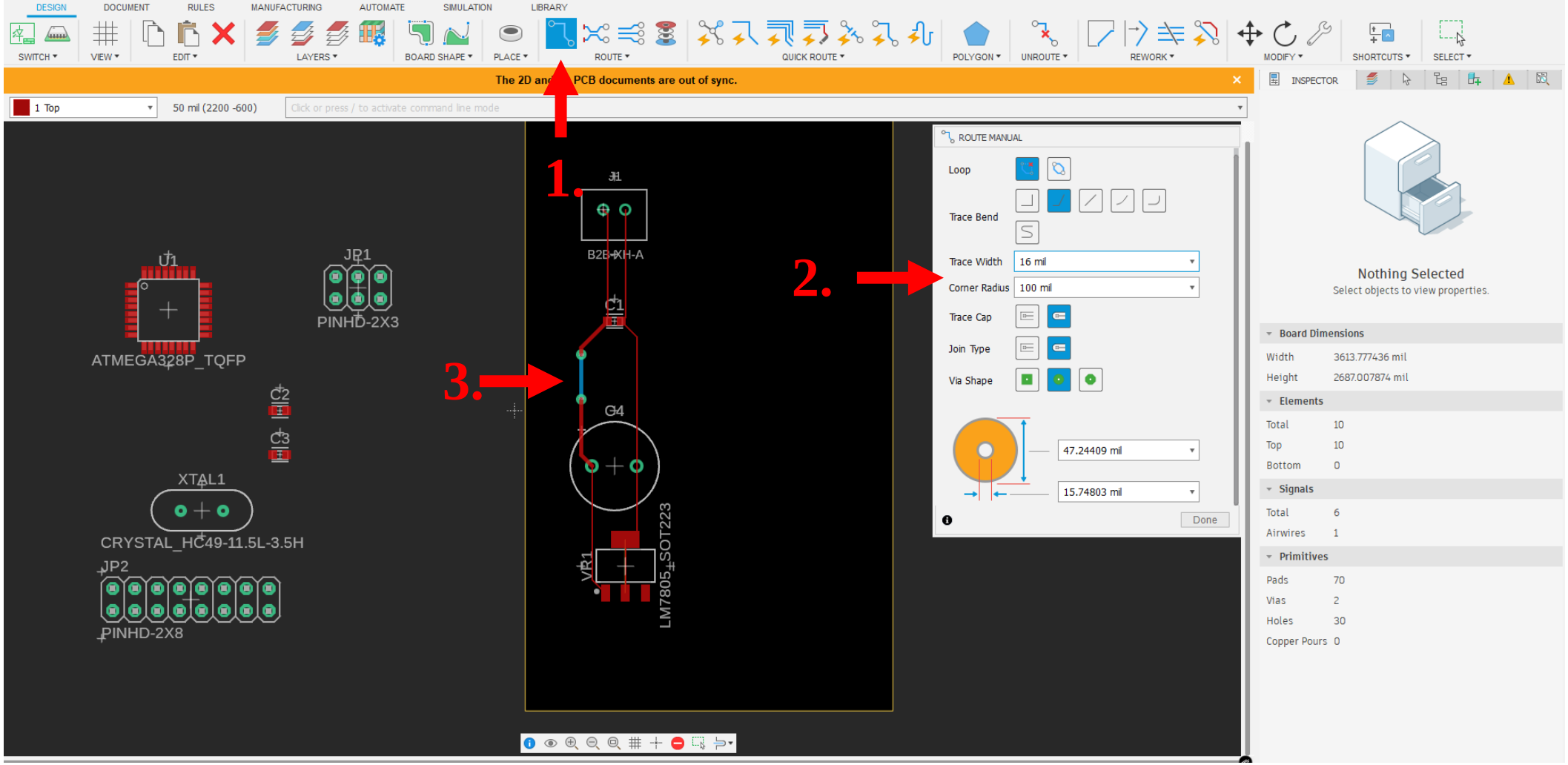
1. Rotate the components that require it.
2. Draw the connections in the schematic.
3. To make the diagram easier to read, you can add symbols for power sources and zero potential.
4. You can use the bus tool to run multiple parallel wires.
5. When the schematic is ready, go to the PCB view



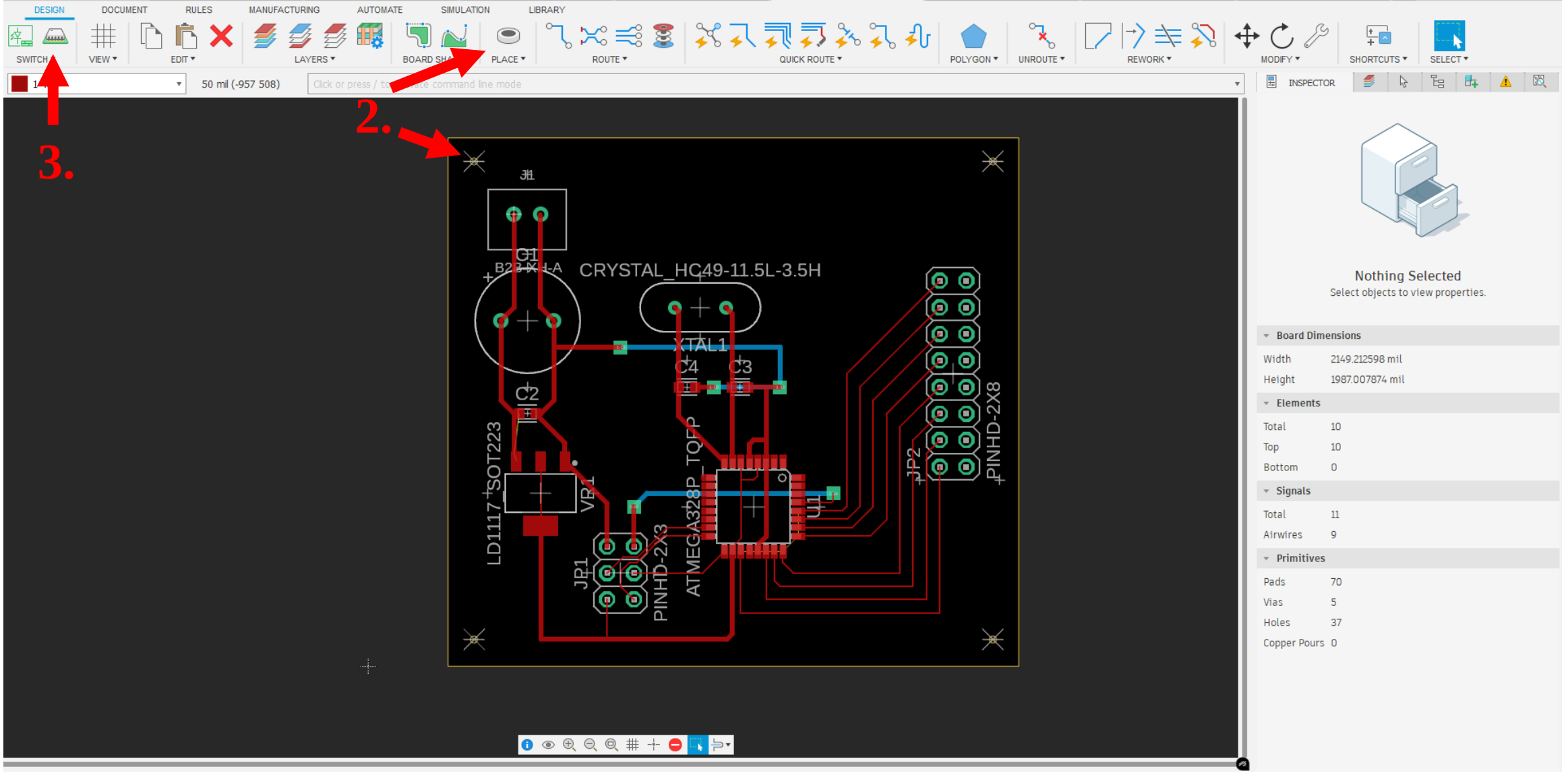
1. Physical models of the components added to the schematic appeared on the PCB. Components in through-hole packages (THTs) have pads marked in green. You can attach a path to them on each layer. The pads of SMD components are marked in red and are only available on the top layer.
2. The black box defines the boundaries of the board, inside which all components must be placed. The dimensions of the board can be customized.



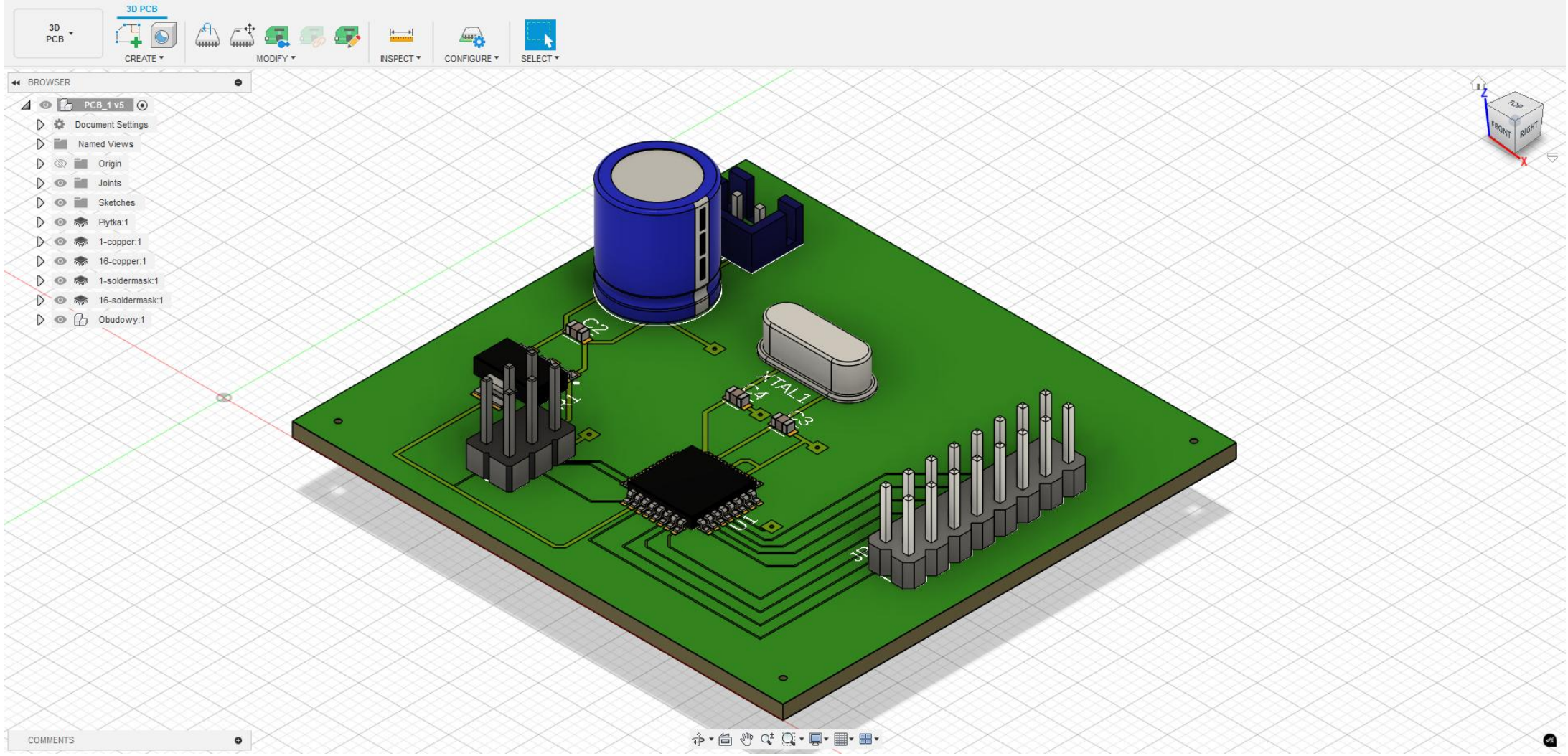
3. Connections from the wiring diagram in this view are marked with yellow lines. They are not physical connections, but only hints as to which connections should be connected to each other.



1. Add paths between the appropriate connections. Paths cannot intersect within a single layer.
2. You can customize path properties. Make sure the path with the negative GND potential is wider than the others.
3. To allow the intersection of 2 tracks, make a via by clicking the SPACE button while running the track and confirm with clicking. A track of a different color is located in a different layer.



1. Arrange all components on the board and connect all required connections.
2. Place mounting holes in the corners of the board.
3. Once the project is complete, generate the 3D model.



The finished design can be exported as an assembly, and its exact dimensions can be taken into account in the modeling of other elements of the product.