



# KNX SIMULATOR

## Contents

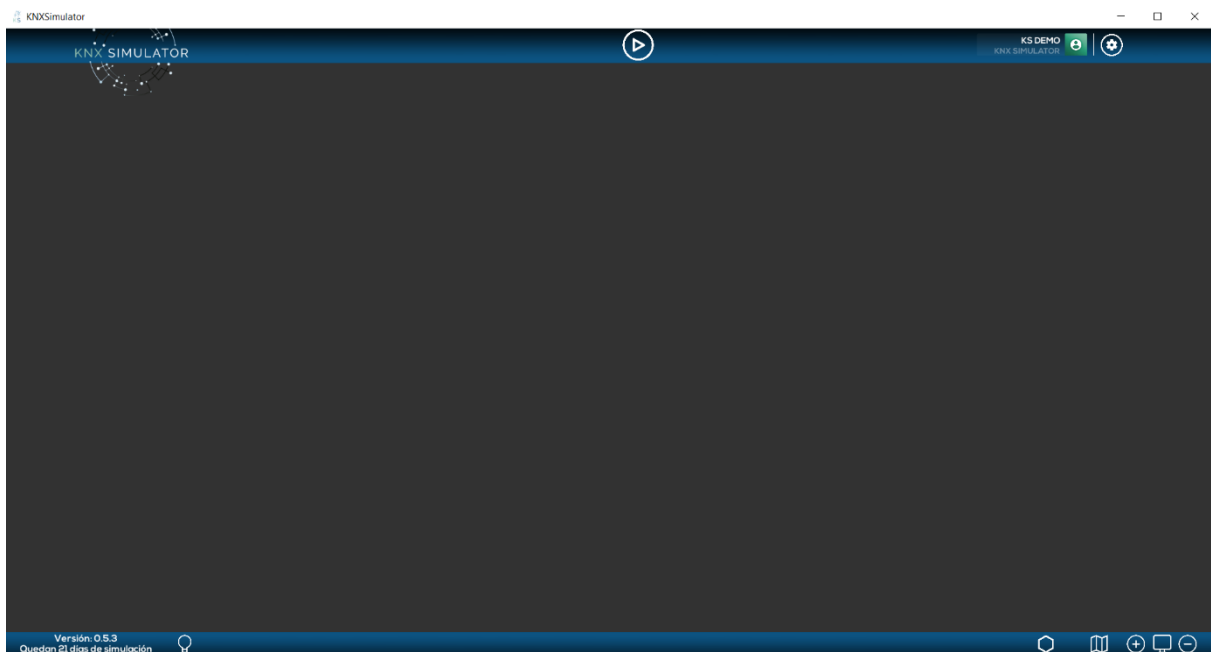
|            |                                     |           |
|------------|-------------------------------------|-----------|
| <b>1</b>   | <b>Introduction .....</b>           | <b>3</b>  |
| <b>2</b>   | <b>Turn on/off light bulb .....</b> | <b>4</b>  |
| <b>2.1</b> | <b>Design: Select devices .....</b> | <b>5</b>  |
| <b>2.2</b> | <b>Wire devices .....</b>           | <b>7</b>  |
| <b>2.3</b> | <b>Commissioning .....</b>          | <b>8</b>  |
| <b>2.4</b> | <b>Save Project .....</b>           | <b>10</b> |

## 1 Introduction

In KNX Simulator you can turn on/off light bulbs with devices that simulate conventional electrical devices.

Although in this example we are going to use only one light bulb, we can carry out circuits with multiple light bulbs in series and/or parallel.

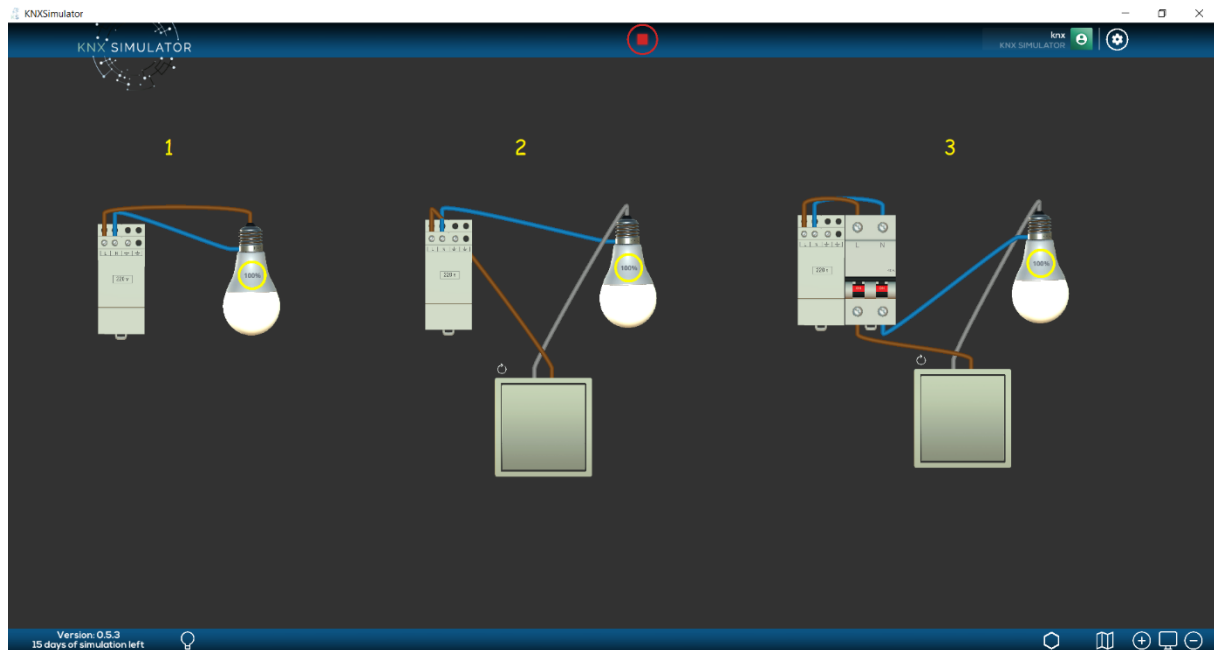
Once we have introduced the login details which we have received in our email account (user, password and code), we enter the simulation room (**Graphic 1**).



**Graphic 1: Simulation Room**

The simulation room is our working area. Here, we will make the design, wiring and commissioning of our projects.

## 2 Turn on/off light bulb



**Graphic 2: turn on/off light bulb circuit**

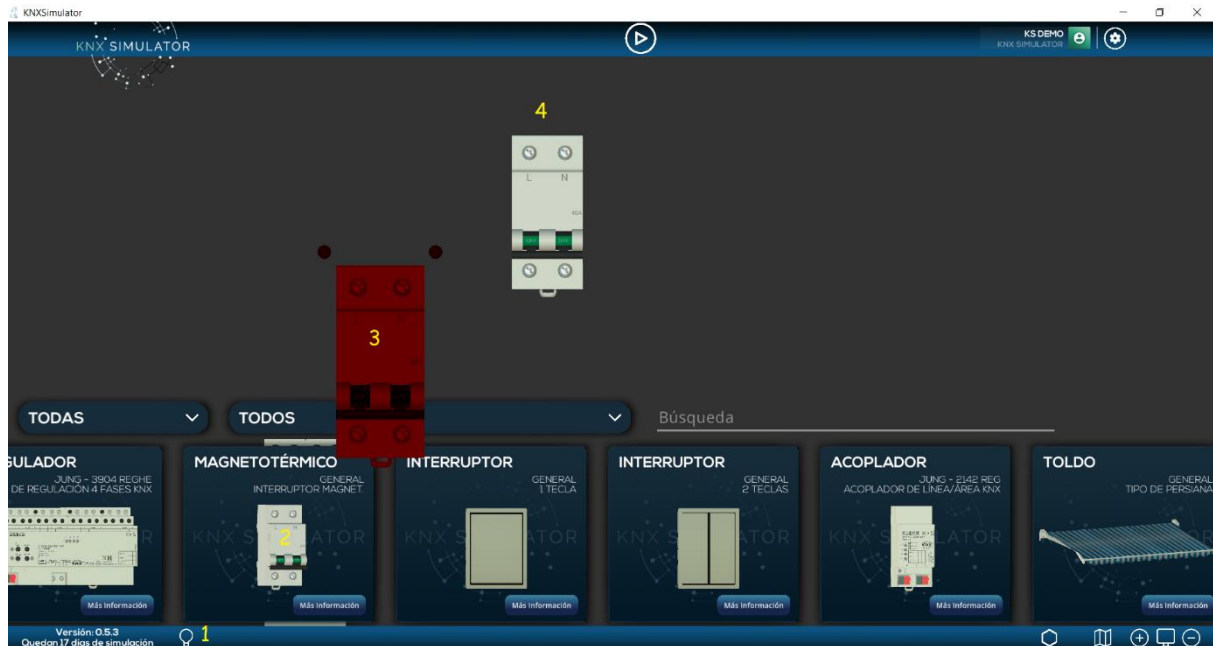
In **Graphic 2** we observe three types of conventional circuits to turn on/off one light bulb (see yellow numerical sequence).

1. By directly moving phase and neutral wires from the power supply to the light bulb.
2. By adding one push-button between the power supply and the light bulb.
3. By adding one circuit breaker just by the power supply output and moving phase and neutral wires from the circuit breaker outputs to the push-button and light bulb, respectively.

In this project, we are going to carry out a circuit step by step as that marked as figure 3 in **Graphic 2**.

## 2.1 Design: Select devices

First of all, we must select the devices from the catalogue to the working area in order to carry out the circuit.



**Graphic 3: Devices (catalogue)**

We follow the yellow numerical sequence in **Graphic 3** to select and drag the circuit's devices: turn on/off.

1. We press on the Devices icon at the bottom bar.
2. A catalogue with the devices of the simulator will open. We press on the device we are going to use in the installation, and...
3. Without dropping it...
4. We drag it to the working area.

It is also possible to select it by simply clicking on its image. In this case, it will remain behind the boxes that contain the devices and won't be seen until we close the catalogue.

To **close the catalogue**, we must press on the **Devices** icon (Graphic 3, figure 1) located at the bottom bar.

Once we have started to drag to the working area, we can already adjust their size so that more components can be easily introduced. The size of the devices can be adjusted within the working area by using our mouse wheel, or by using the zoom in (+) zoom out (-) icons, located at the bottom bar to the right.

We repeat this operation of dragging the devices to the working area until we have all the necessary components for the circuit.

It is possible that the component that we want to select and move to the working area is not visible within the boxes of the catalogue, but we can find it in three different ways:

- By directly writing its name on Search.
- By moving the devices of the catalogue. We press on the blue box of one of the devices and without dropping it, we move it to the right or to the left so that a horizontal movement is produced.
- By using the filters.



**Graphic 4:** Suggested devices to turn on/off the light bulb.

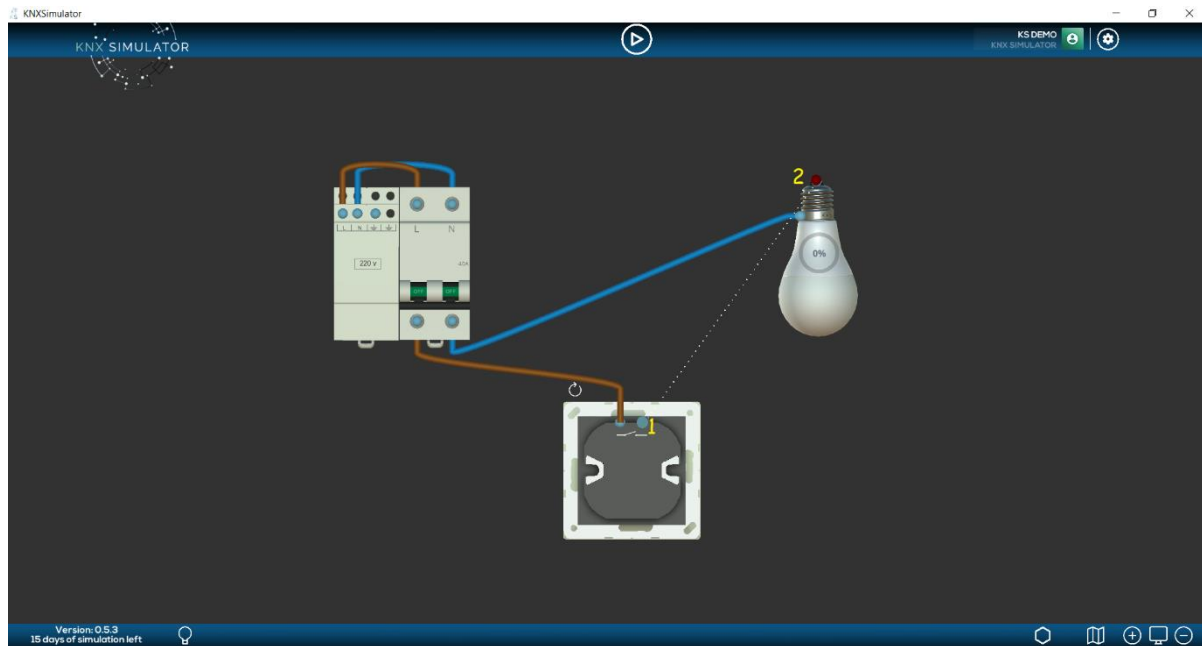
As follows, we observe the suggested devices for this practice, shown in **Graphic 4**:

1. Power Supply. It is a 220V power supply.
2. Circuit breaker.
3. Push-button. 1 gang.
4. Light bulb.

Once all suggested devices are in the working area, we begin to wire them.

**IMPORTANT!** We can add devices at any time to the working area except when we are simulating (PLAY activated). We must press STOP before opening the catalogue of devices.

## 2.2 Wire devices



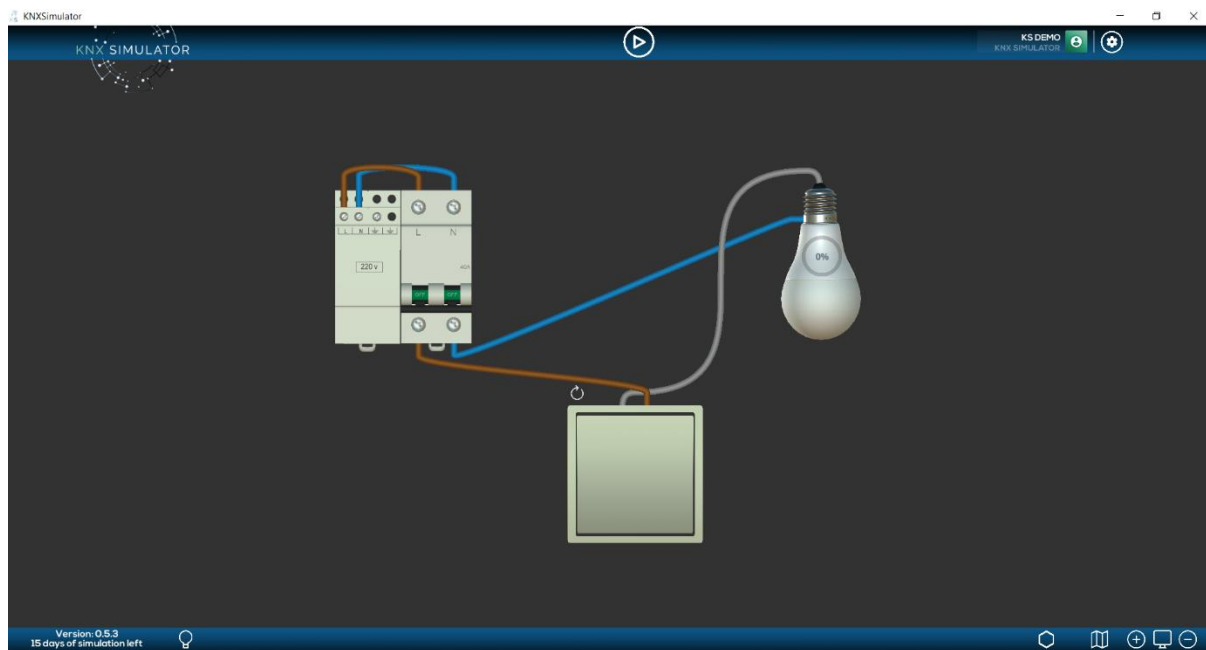
**Graphic 5: Wire devices**

Para cablear los dispositivos en la zona de trabajo seguimos la secuencia de números (en color amarillo) del **Gráfico 5**.

In order to wire the devices in the working area we follow the yellow numerical sequence in **Graphic 5**.

1. We press on one of the nodes of connection of the device to be wired...
2. And we keep pressing until we reach the desired node (entry node will turn to dark red colour).

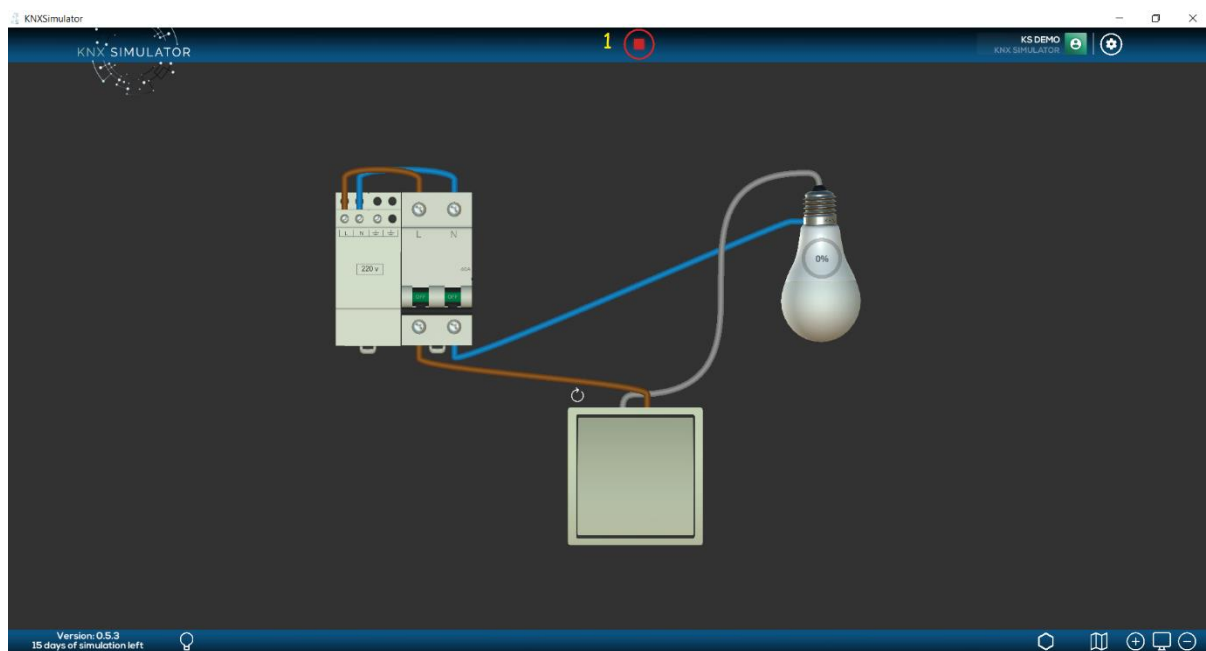
**IMPORTANT!** It is not possible to wire devices while we are simulating (PLAY activated). We must press STOP before we begin to wire.



**Graphic 6: Design and wire completed**

Once all the devices are correctly wired, as shown in **Graphic 6**, we can begin the simulation.

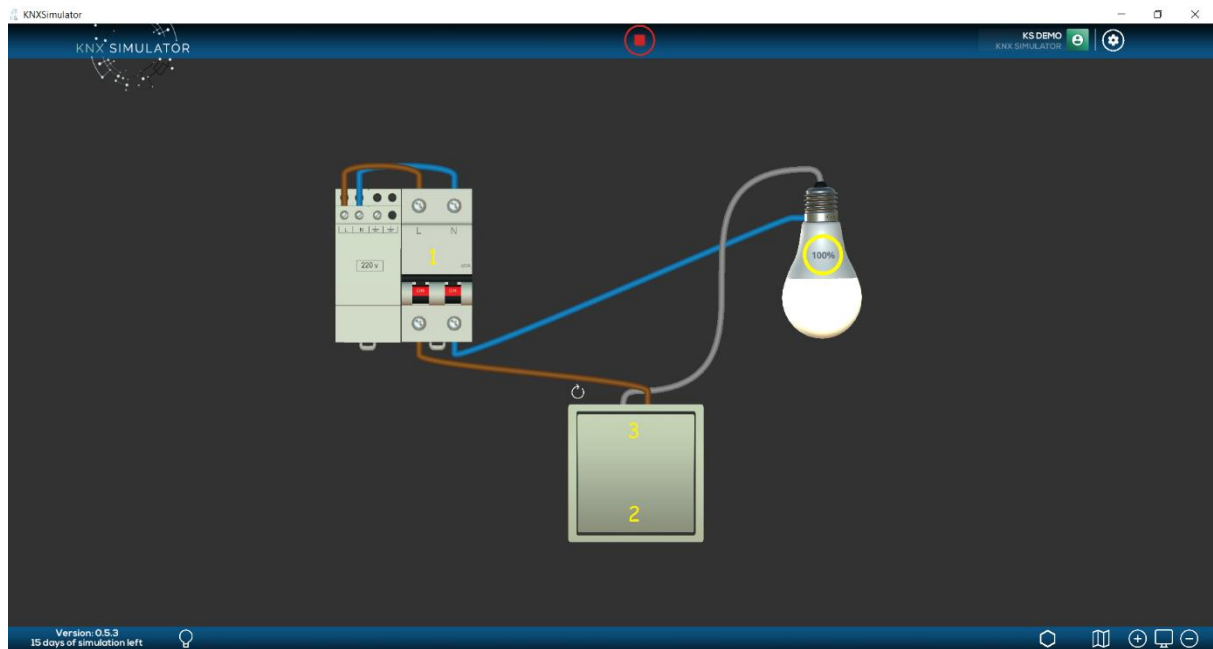
## 2.3 Commissioning



**Graphic 7: PLAY in KNX Simulator**

In order to commission a conventional electrical circuit, it is only necessary to press PLAY (icon will change to a red coloured STOP). See **Graphic 7**, yellow figure (1).





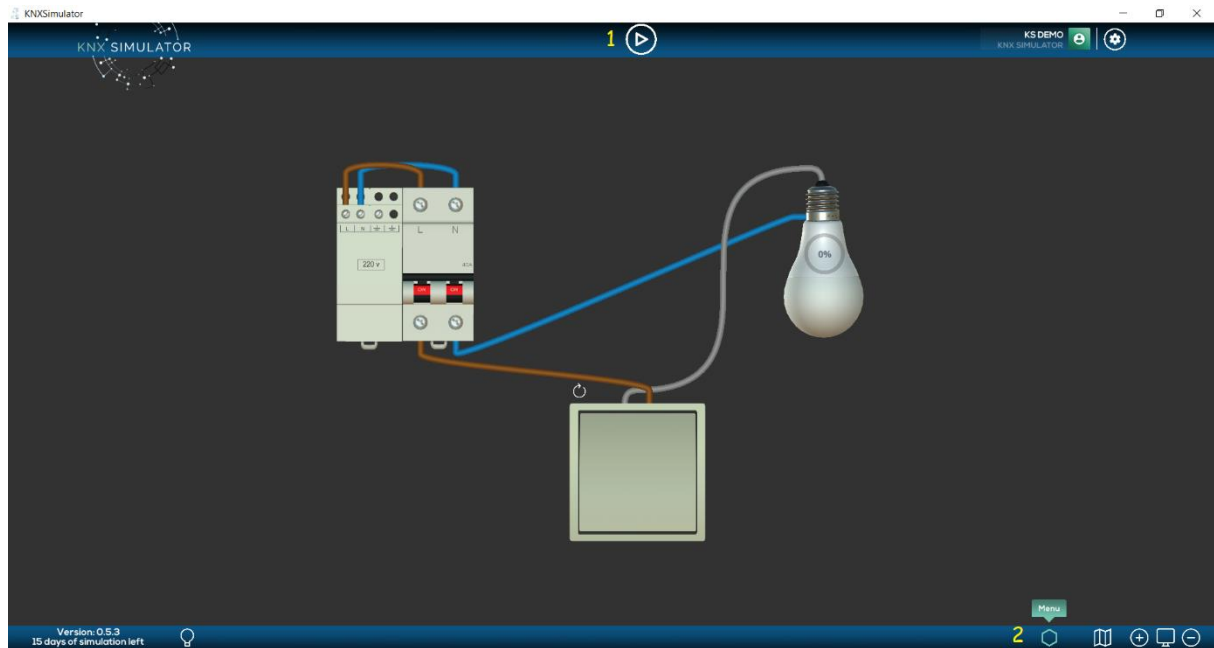
**Graphic 8: Simulation**

**Graphic 8** shows the simulation sequence: turn on/off light bulb.

1. We turn on the circuit breaker of the installation so that the passage of current is allowed (it is NOT mandatory to use it, suggested device).
2. We press on the bottom surface of the push-button and the light bulb turns on.
3. We press on the top surface of the push-button and the light bulb turns off.

## 2.4 Save Project

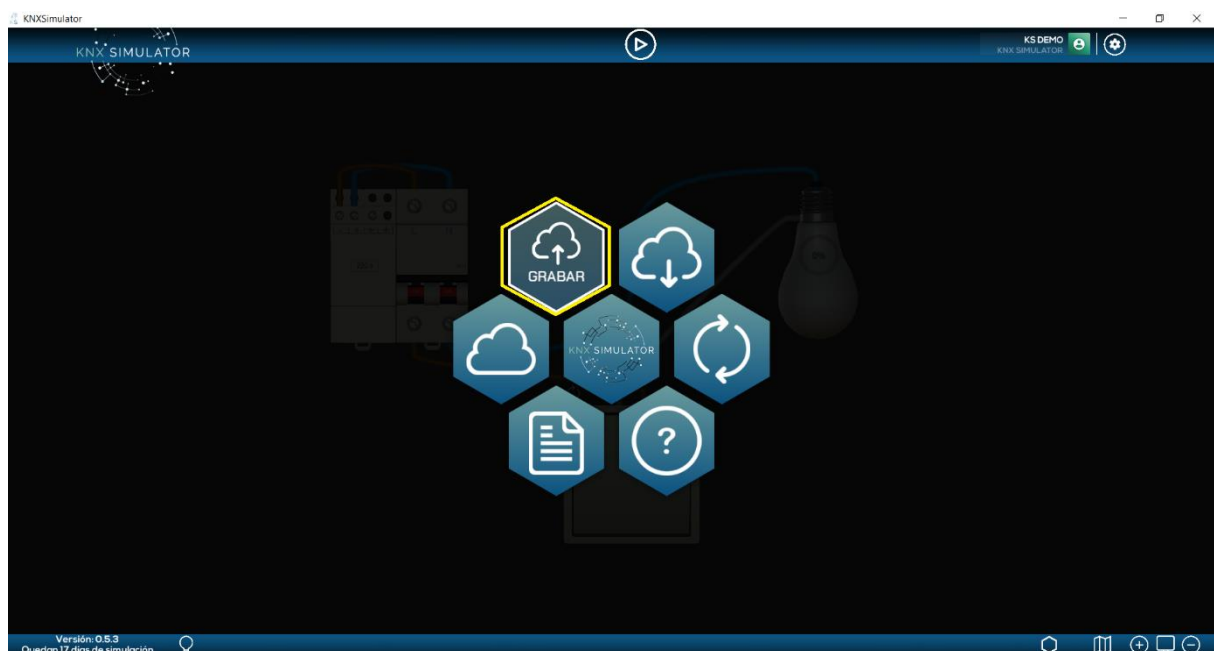
Once we have checked that our circuit functions correctly, we save it.



**Graphic 9: Access to Menu**

**Graphic 9** shows the sequence to accessing the Menu.

1. We press STOP. The icon will turn to white colour.
2. We press on the Menu icon, located at the bottom bar.

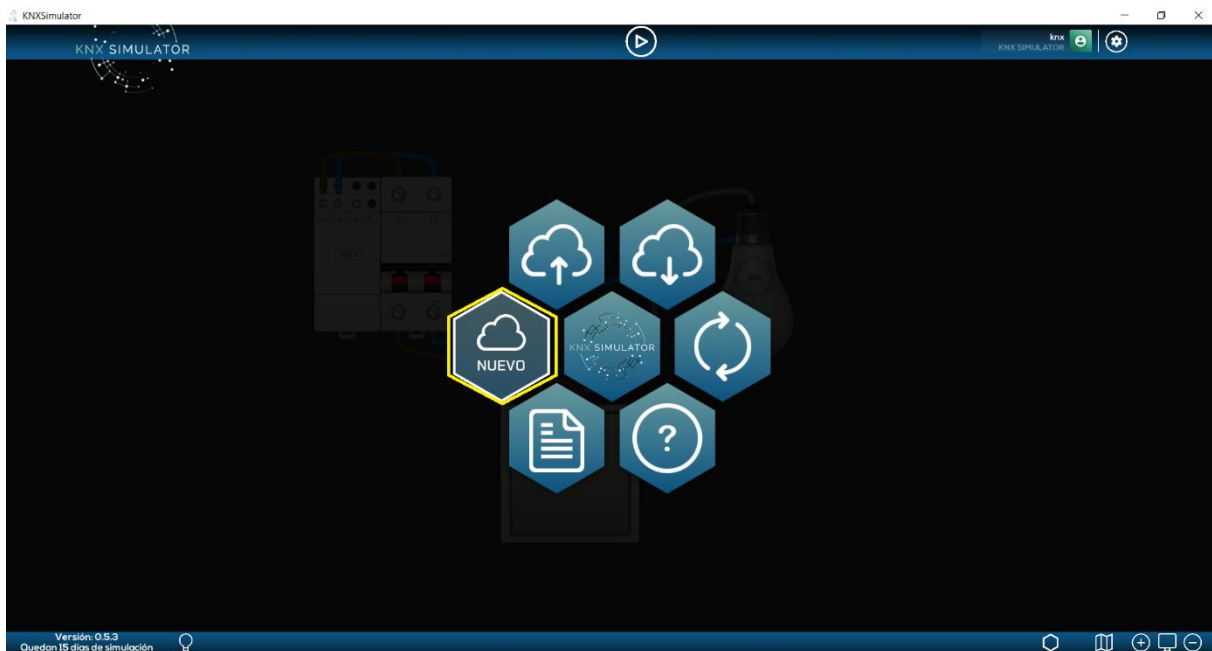


**Graphic 10: Save project**

We can **SAVE** the project by pressing on the corresponding icon (**Graphic 10**, marked in yellow).

A window will pop-up, in which we will indicate the name and location of the project within our computer. The file with the saved project will have the (.ks.) extension.

Once we have saved the circuit, we can go back to the installation and continue simulating – by pressing on the black area (outside the Menu options) – or we can start a **NEW** project.



**Graphic 11: New project**

After pressing on **NEW** (**Graphic 11**, marked in yellow), all devices that may be within the working area will be deleted to foster the beginning of a new project.