



Add-in created in COMSOL Multiphysics 6.4

# Circuit Extractor

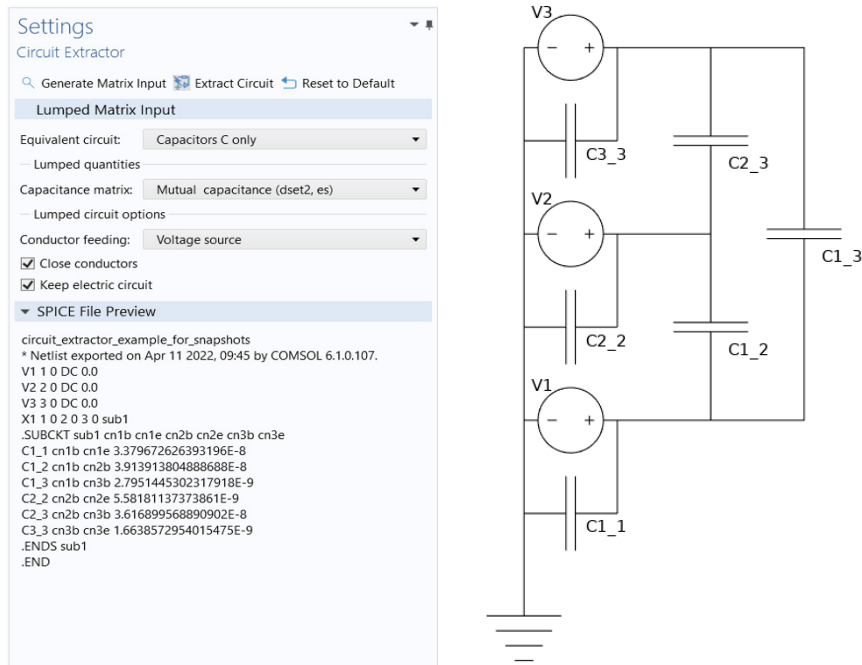
## *Introduction*

---

One of the objectives of electromagnetic simulation is often the extraction of lumped quantities that can be used in electrical circuit simulations. The Circuit Extractor add-in offers an easy way of directly converting the result of electromagnetic simulations into an Electrical Circuit physics interface, and subsequently into a SPICE file. The supported physics interfaces for making such an extraction are Electric Currents (for resistances), Electrostatics (for capacitances), Electrostatics, Boundary Elements (for capacitances), Magnetic Fields Currents Only (for resistances, inductances, and impedances), and Magnetic and Electric Fields (for resistances, inductances, and impedances). Multiple physics interfaces and solutions can contribute to the same extracted circuit.

In order to use the Circuit Extractor add-in, a *Stationary Source Sweep* or a *Frequency Domain Source Sweep* must be solved for the physics of interest. Matrices that are generated from solving such studies are then used for generating the equivalent circuit. After solving, you only need to select the **Equivalent Circuit**, select the **Evaluation Group** corresponding to the desired solution, and press the **Extract Circuit** button. This will open

a window for saving the extracted SPICE file. An example of the user interface is shown in [Figure 1](#), along with the corresponding circuit.



*Figure 1: The user interface of the add-in (left) and the corresponding circuit (right).*

---

**Add-in Library path:** ACDC\_Module/circuit\_extractor

---

### *Circuit Extractor*

The **Settings** window toolbar for this add-in contains three buttons. The **Generate Matrix Input** button generates all possible lumped matrices from the existing solutions. These matrices are placed in **Evaluation Group** features under **Results**. The size of these matrices is controlled by the source sweep study solved in the model, which by default scans over the terminals that are defined at the physics level. The order of the rows and columns is the same as the numbering of the feeding elements in the original physics interface. This numbering is also shared by the extracted circuit. The **Extract Circuit** button runs the

add-in as specified by the given settings, and saves the output. Finally, the **Reset to Default** button resets all settings in the add-in to their default values.

### **LUMPED MATRIX INPUT**

This section contains the user input. In the **Equivalent Circuit** drop down menu, you can select which equivalent circuit analogy should be used for the model. The options are **Resistors R only**, **Inductors L only**, **Capacitors C only**, **R and L (series, diagonal R)**, **R and C (parallel, diagonal R)**, **R, L, and C (parallel C, diagonal R)**, as well as **Impedances Z**. Based on the choice, different lumped inputs must be specified in the following sections.

Under **Lumped quantities**, the drop down menu(s) can be used to select which matrices from the study results should be used for the circuit extraction. Depending on which equivalent circuit analogy is chosen above, different options become available.

Finally, under **Lumped circuit options**, there are others settings that can be used to modify the extracted circuit. In particular, the **Conductor feeding** drop down menu allows you to specify if each conductor should have a **Voltage source**, a **Current source**, or **No feeding element**. Furthermore, the **Close conductors** checkbox determines whether or not it should be assumed that each conductor has both connections closed onto the same ground. If checked, the circuit becomes closed since no extra elements are needed. The **Keep Electrical Circuit** checkbox instead determines if the Electrical Circuit physics interface should be kept after the circuit extraction has finished. If unchecked, only the SPICE file is saved.

### **SPICE FILE PREVIEW**

This section offers information on the status of the add-in. In particular, when the circuit extraction has finished (except for the case with only impedances), it shows the contents of the extracted circuit in the SPICE format.