**Circuit Simulation Tool**

User’s Manual

*Group Members:*

* Mahmoud Youssri Sayed
* Khaled Sabry Ibrahim
* Mohammed Magdy Khalil
* Hady Maher
* Mahmoud Othman Adas

*Project overview:*

A circuit simulation tool designed to solve any circuit. The user inputs the circuit details to the simulation and it calculates the current, voltage, power, maximum resistance or maximum power depending on the user’s needs.

*How to use:*

These are some clear simple steps to help you use the simulation:

1. Open the simulation
2. Input the circuit node by node
   1. Each node is inserted by entering each directly connected element to it. The element can be a resistance(R), a current source (J) or a voltage source (E). Each element has a type (R, J or E), an ID and a value. The element is inserted in the format Type ID Value... Examples:

R1 15 E2 4 J1 2 R3 40

* 1. The elements must be inserted in valid units. The units are Ohm for resistance, Ampere for current sources and Volt for voltage sources.

1. After inserting each node press ‘x’.
2. After inserting the whole circuit press ‘xx’.
3. Press ‘I’ to calculate the current, ‘V’ to calculate the voltage difference, ‘P’ to calculate the power, ‘R’ to calculate the maximum resistance and ‘M’ to calculate the maximum power.
4. Input the required element in the format Type ID

Examples: R2 E3

1. To do any further calculation, repeat the steps (5-6).
2. To exit the simulation press x.

Program flow