**Circuit solver**

This project aims to create LTspice simulated circuit from hand-drawn circuits for the students who don’t know how to use simulators.

In this project, we are using ML and DL to recognize a circuit drawn on paper and create its simulation on LTspice. It will recognize each component of a circuit and create its simulation.

* **Technical points:**-

We have used segmentation using an algorithm for identification of open line elements capacitor, battery, Ground, etc, ML for identification of resistors and other continuous elements, DL for the identification of the values associated with each component.

* **Execution**:-

We first scan the hand-drawn circuit image and then with the help of our model, we recognize the different components present in the circuit and simulate the same with the help of LTSpice Python API to generate the output waveform/graph.

* **Beneficial point :**

Students can learn various techniques of circuit solving as learning to work with simulation software is still an essential skill despite this system being available which does not require any prerequisite knowledge of such software.