LTspice

We use LTspice to simulate the circuits.
It can be downloaded for free:
https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html

configuration

OSS library

To make it easy all necessary models are summarized in the OSS.lib
Just copy it into the project folder and add the following command in LTspice (press „S“):

```
.include OSS.lib
```

History OOS.lib

- Version 0.1
- Stand: 02.01.2022

Included models

- Thyyristor MCR706A Source: ON Semiconductor (Now: Littelfuse)
- Diode UF4007 Source: http://ltwiki.org/index.php?title=Standard.dio

Ignition coil

To create an ignition coil, you create a transformator. Add to inductivities (press L) L1 and L2 and then create a directive (press S):

```
K1 L1 L2 0.7
```

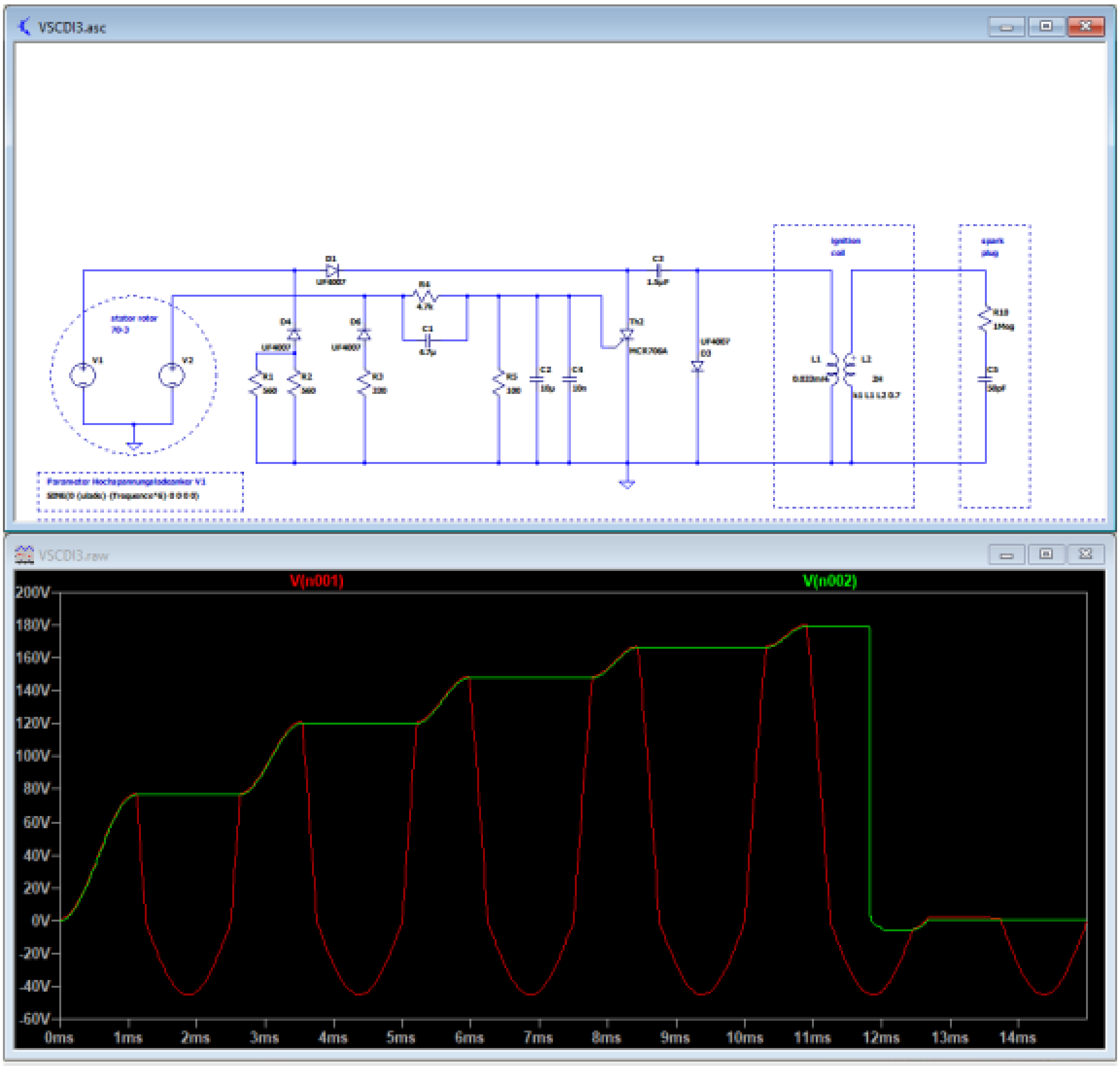
This creates the transformator K1 with L1 and L2. And the last number is the efficiency. 1.0 is an ideal transformator with no loss and 0.7 is a good approximation for a standard ignition coil.

Note

LTSpice is not case-sensitive, so for the unit prefix „Mega“ you need to write 1000k or meg, because m/M stands for milli.

Simulation

This is a first version of a LTspice simulation. It shows the CDI at the Vape 70-3.
The red curve is the voltage of the high voltage coil V1 and the green curve is the voltage of C3.



Its not perfect yet but it works.