

# 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

- Thyristor 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 transformer. Add to inductivities (press L) L1 and L2 and then create a directive (press S):

```
K1 L1 L2 0.7
```

This creates the transformer K1 with L1 and L2. And the last number is the efficiency. 1.0 is an ideal transformer 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.

From:

<https://www.opensimspark.org/> - **OpenSimSpark**

Permanent link:

<https://www.opensimspark.org/ltspice?rev=1641121688>

Last update: **2022/01/02 12:08**

