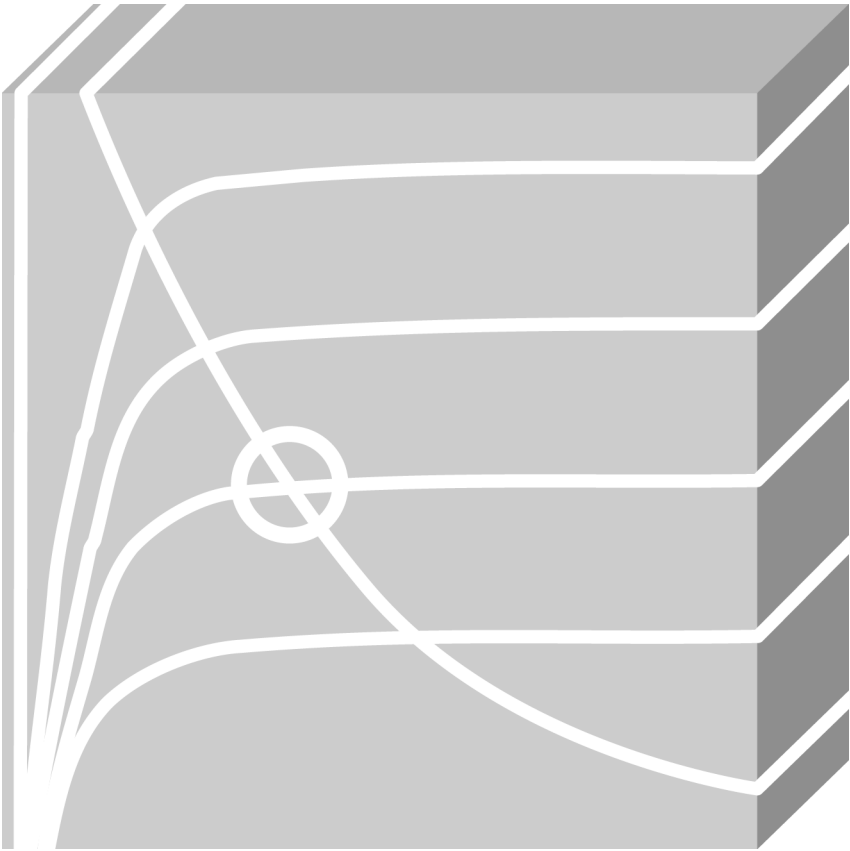


# Using CheckPoint Restart



**Table of Contents**

- 1 Introduction ..... 3**
- 2 Demo Project..... 3**
- 3 Using CheckPoint Restart..... 4**
  - 3.1 Specify the CheckPoints..... 4
  - 3.1.1 Check Point Location ..... 5
  - 3.1.2 Check Point Time ..... 5
  - 3.1.3 Save CheckPoint States at ..... 5
  - 3.2 Run the Initial Analysis ..... 6
  - 3.3 Specify the Restart Point ..... 7
  - 3.4 Restart the Analysis..... 8
- 4 Bibliography ..... 8**

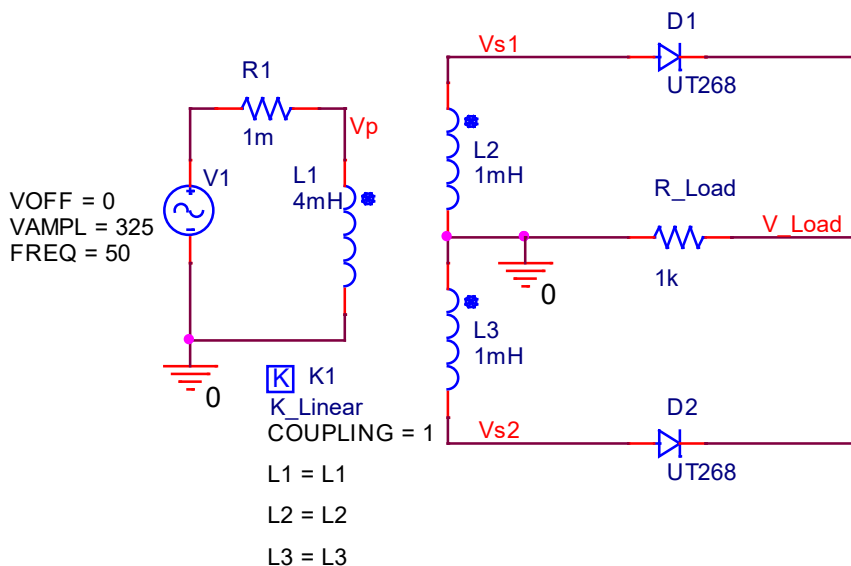
## 1 Introduction

You can save the state of a transient simulation at different moments as CheckPoints. You can specify any of the CheckPoints as a restart point and rerun the simulation from that point.

For example, when you simulate a design and later you modify your design but you don't want to run the simulation from the start possibly because it takes a long time. In this case you can define and save CheckPoints then restart the simulation for running only the portion that is expected to have the outputs of interest.

## 2 Demo Project

The example circuit we would like to build up is a simple full wave rectifier. A demo circuit in the project full\_wave\_rectifier.opj is created as follows:



The amplitude of the voltage source V1 is 325V. According to the following function

$$V_{eff} = \frac{1}{\sqrt{2}} V_{pk} \quad (1)$$

where:

$V_{eff}$  is the effective voltage and

$V_{pk}$  is the peak voltage

We can get the effective voltage of V1 is about 230V.

### Note

PSpice doesn't allow loops of only inductors and/or voltage sources, therefore we need a resistor R1 to break the loop in the primary side.

The centre tapped transformer can be built with three inductors and an ideal linear coupling for which the value of "COUPLING" is set to "1".

In an ideal transformer, the voltage in the secondary side  $V_S$  is proportional to the voltage in the primary side  $V_P$ . The transfer factor  $\gamma$  can be described using function 2:

$$\gamma = \frac{V_P}{V_S} = \frac{N_P}{N_S} \quad (2)$$

where  $N_P$  is the number of turns in the primary winding and  $N_S$  is the number of turns in the secondary winding.

Further there is

$$\frac{N_P}{N_S} = \sqrt{\frac{L_P}{L_S}} \quad (3)$$

Then we get

$$\frac{V_P}{V_S} = \sqrt{\frac{L_P}{L_S}} \quad (4)$$

In the demo circuit  $L_1=4\text{mH}$ ,  $L_2=1\text{mH}$  and  $L_3=1\text{mH}$ , hence the transfer factor

$$\gamma = \frac{V_P}{V_{S1}} = \sqrt{\frac{4\text{mH}}{1\text{mH}}} = 2 \quad (5)$$

### 3 Using CheckPoint Restart

As an example, here we can change the value of the load resistor  $R_{\text{Load}}$  to observe how the load current changes.

To use CheckPoints you need to:

- Specify the CheckPoints
- Run the initial analysis
- Specify the restart point
- Restart the analysis

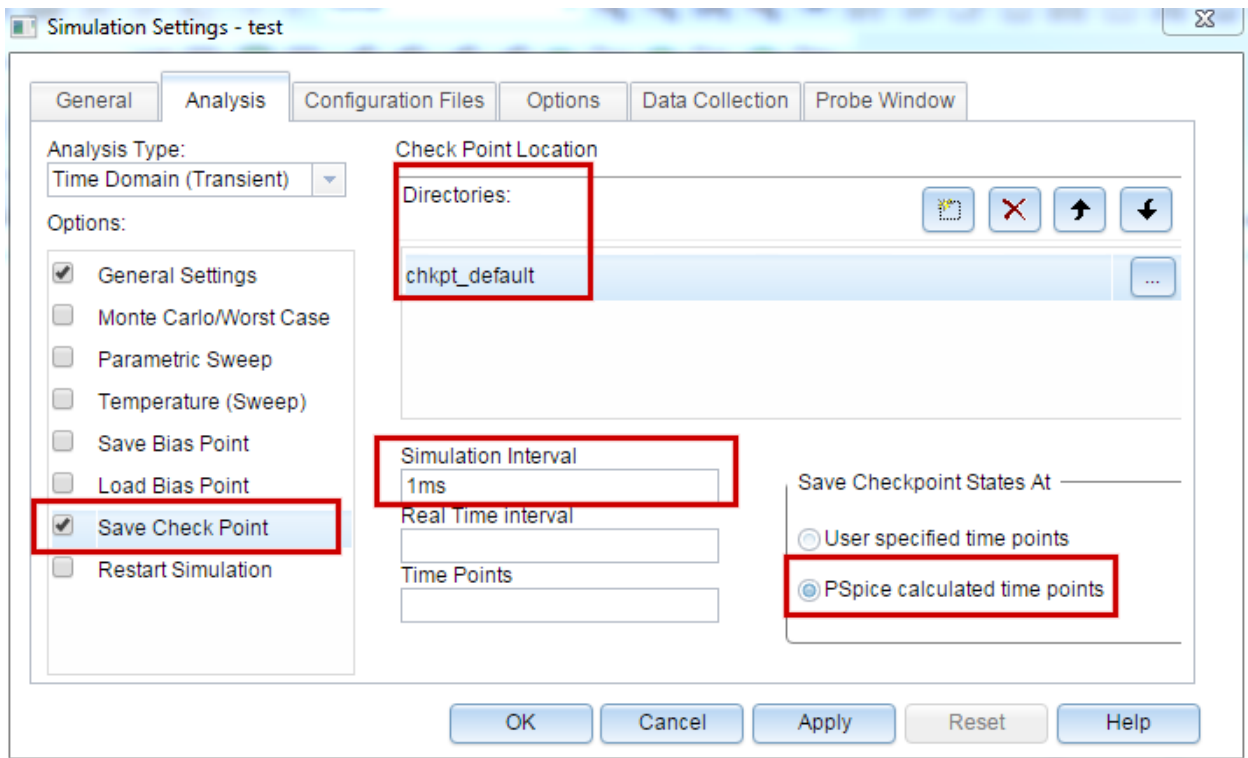
#### 3.1 Specify the CheckPoints

Normally you can define CheckPoints either from the **Simulation Settings** dialog box or by editing the circuit file (.cir). As an example, here we will just define CheckPoints from Simulation Profile.

Assuming you have opened the PSpice project full\_wave\_rectifier.opj. From the menu bar click **PSpice > Edit Simulation Profile >** the **Simulation Settings** dialog box comes up.

Click the **Analysis** tab > leave the **General Settings** unchanged and select **Save Check Points** in the **Options** list.

You need to define the following:



### 3.1.1 Check Point Location

Specify the location where the CheckPoint data will be stored. If you don't specify another location the default directory named `chkpt_default` will be used and the CheckPoint data will be saved locally in the Simulation Settings Folder.

### 3.1.2 Check Point Time

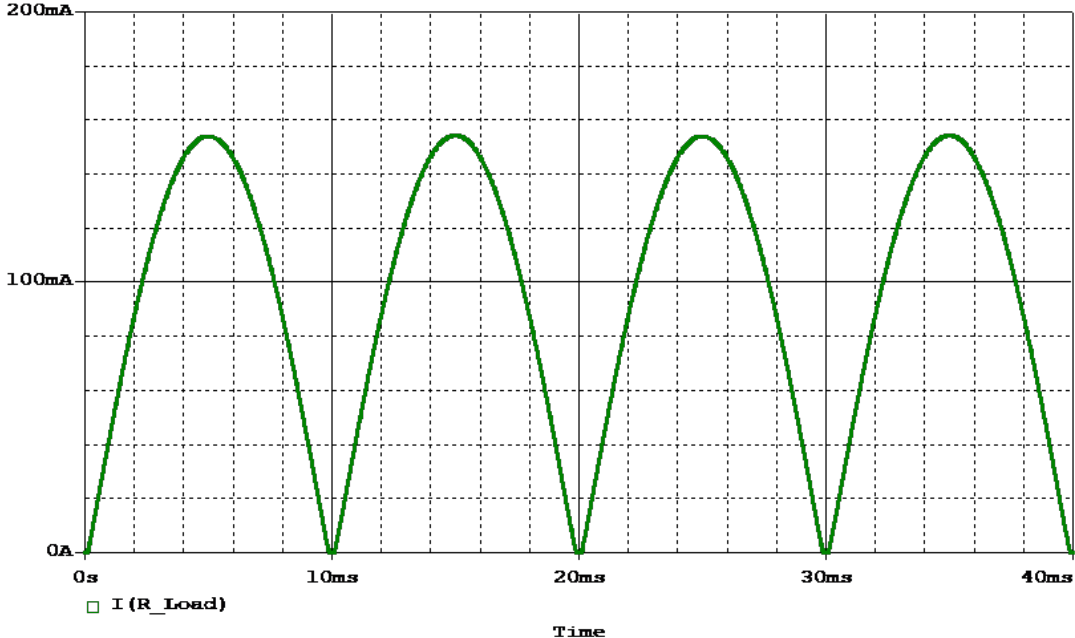
Using one of the three possibilities **Simulation Interval/Real Time/Time Points** to specify when CheckPoints should be created. As an example, we set 1ms for the **Simulation Interval**. This means the CheckPoints will be created every 1ms. For more information about the options **Real Time** and **Time Points** please refer PSpice User's Guide [1].

### 3.1.3 Save CheckPoint States at

Here we can e.g. select the **PSpice calculated time point** for saving Checkpoint States. Click **OK** in the **Simulation Settings** dialog box to accept the settings.

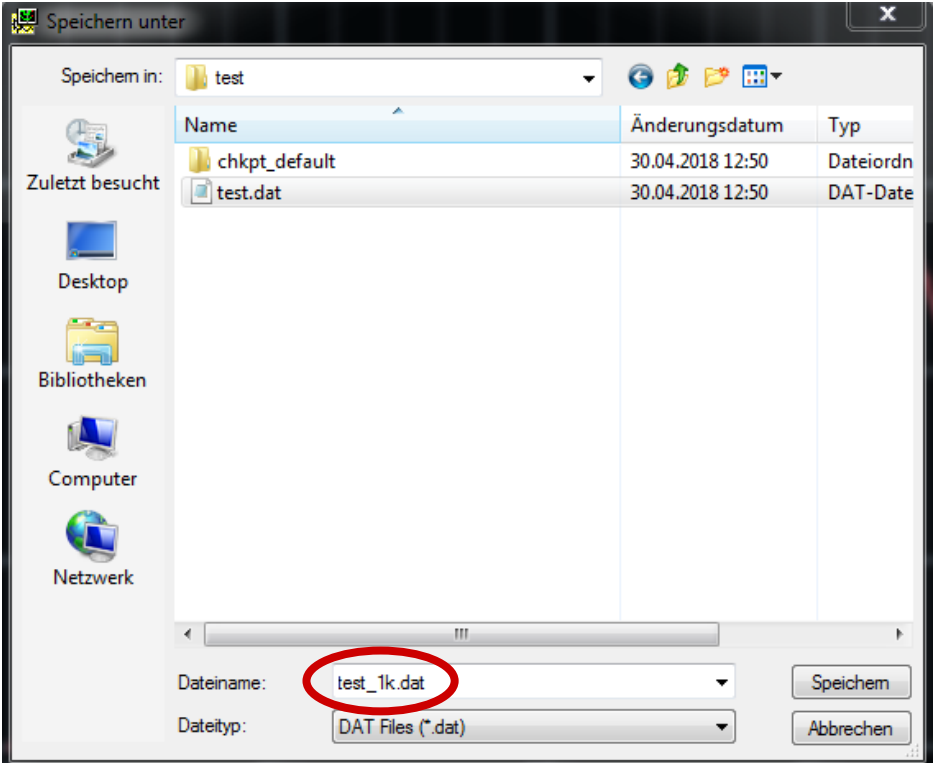
### 3.2 Run the Initial Analysis

After you finish setting the simulation profile, run the simulation for e.g. 40ms. The current  $I(R\_Load)$  which flows through the load resistor is displayed as follows:



In the above diagram you will see that both the positive half wave and the negative half wave are transmitted.

We can save this probe data file for  $R\_Load=1k$  Ohm. In probe window click **File > Save as...**



In text box **Dateiname** enter a new name e.g. test\_1k.dat, then save it.

## 3.3 Specify the Restart Point

You might want to restart a simulation from a saved CheckPoint after changing the design. You can change the following in the schematic before restarting a simulation:

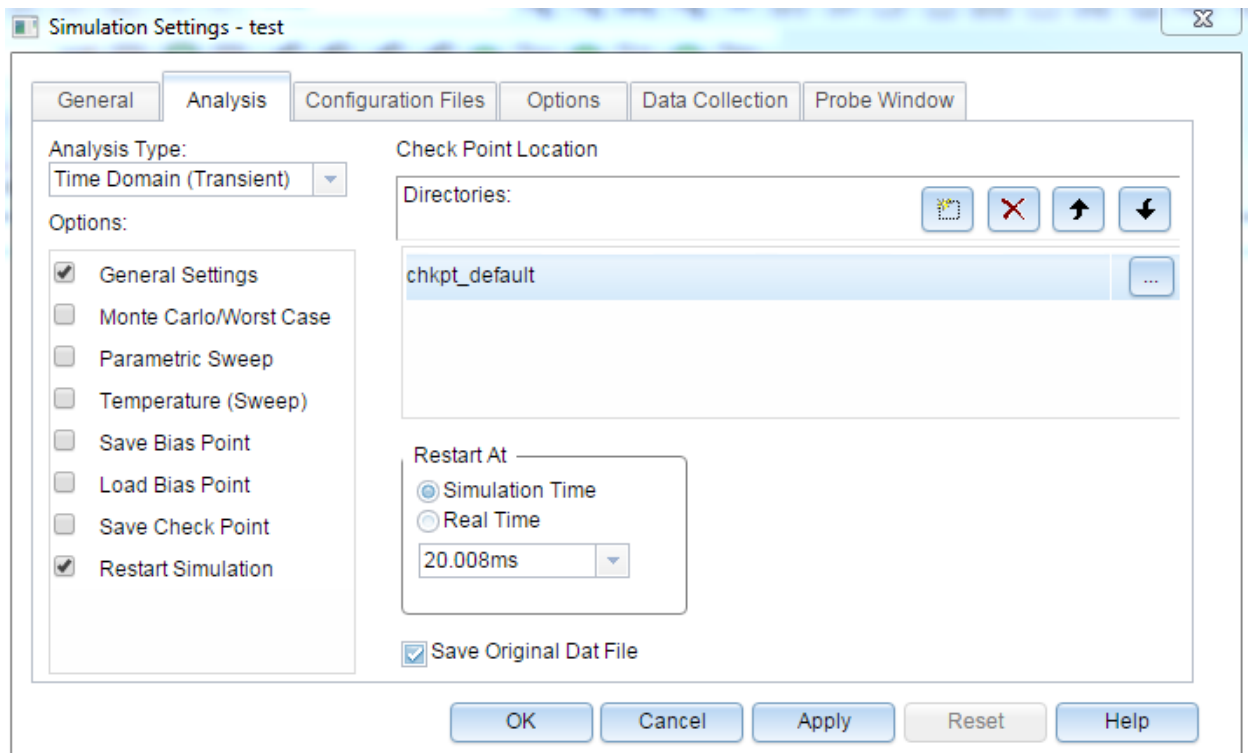
- Component values
- Parameter values
- Simulation options
- CheckPoint restart options
- Data save options

But you can not do the following:

- Add or remove components before restarting simulation
- Change device name or order in the circuit file
- Change the initial condition of device such as capacitor
- Change multi-analysis options, such as temperature, parameter, or MC sweep
- Topology changes for digital and mixed signal circuits

For example, we change the value of R\_Load from 1k Ohm to 500 Ohm in schematic page.

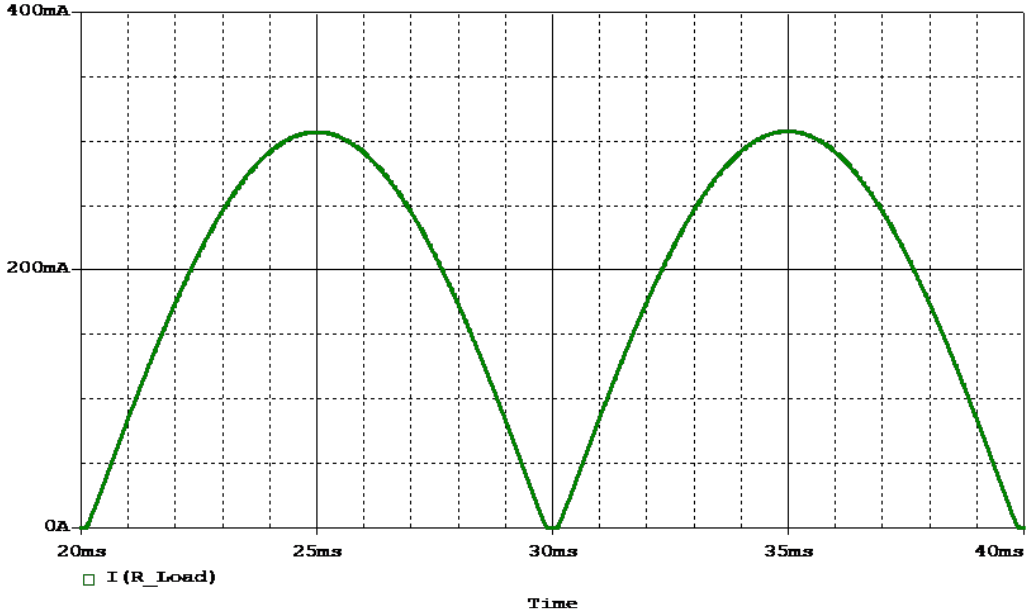
In the **Simulation Settings** dialog box uncheck **Save Check Points** and check **Restart Simulation**. We use the same directory in which the CheckPoints are saved. Select e.g. the **Simulation Time** for **Restart At** and restart the simulation at 20.008ms.



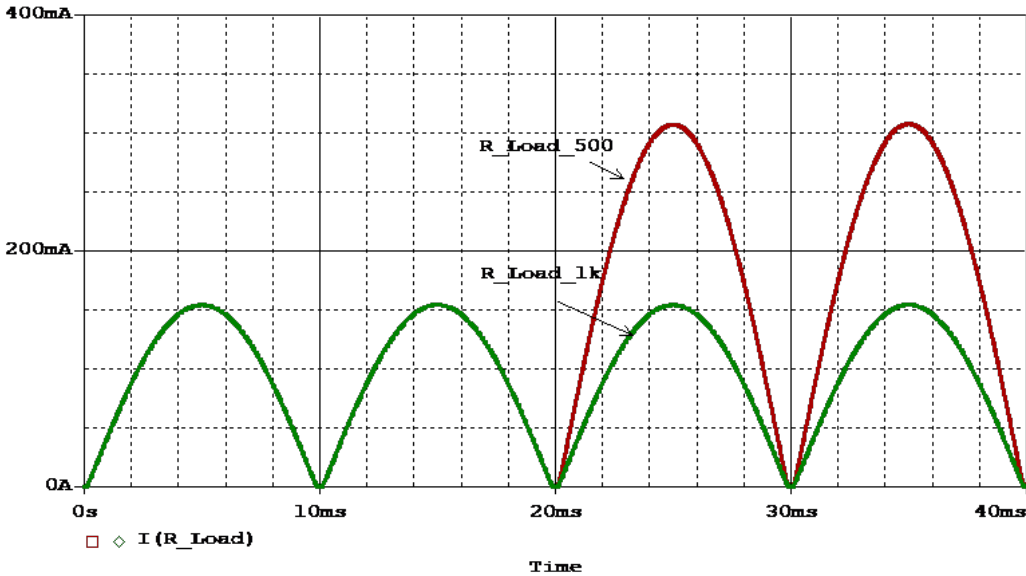
Click **OK** to save this simulation profile.

### 3.4 Restart the Analysis

Run the simulation. The new result is as follows:



If you want to see the difference before and after changing the load resistor, you can append the waveform. In the probe window, click **File > Append Waveform (.DAT)...** > select the test\_1k.dat file and open it.



In this probe window you can see that the load current  $I(R\_Load)$  for the modified design has in fact doubled after 20ms.

## 4 Bibliography

- [1] PSpice User's Guide, Cadence
- [2] OrCAD Capture User's Guide, Cadence
- [3] Leistungselektronische Schaltungen, D. Schröder, Springer