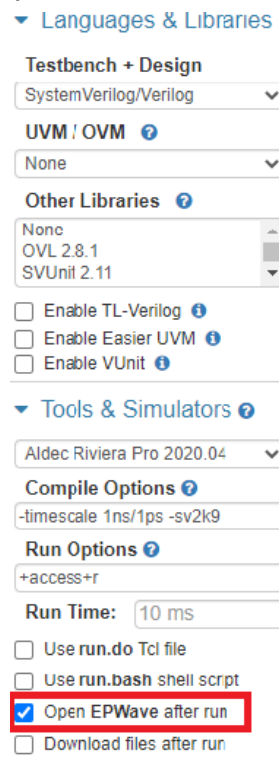


How to Visualize Waveform in EdaPlayground

This document includes necessary steps to generate waveform in EdaPlayground.

Step 1: Make sure that “Open EpWave after run” box is checked.



The screenshot shows the configuration panel for EdaPlayground. The 'Tools & Simulators' section is expanded, showing the following settings:

- Testbench + Design**
 - SystemVerilog/Verilog
- UVM / OVM**
 - None
- Other Libraries**
 - None
 - OVL 2.8.1
 - SVUnit 2.11
- Enable TL-Verilog** ☐
- Enable Easier UVM** ☐
- Enable VUnit** ☐
- Tools & Simulators**
 - Aldec Riviera Pro 2020.04
- Compile Options**
 - timescale 1ns/1ps -sv2k9
- Run Options**
 - +access+r
- Run Time:** 10 ms
- Use run.do Tcl file** ☐
- Use run.bash shell script** ☐
- Open EPWave after run** ☒
- Download files after run** ☐

Step 2: Put this snippet inside your code once at the beginning so that EDA Playground can generate waveform.

```
initial begin
    $dumpfile("dump.vcd");
    $dumpvars(1);
end
```

Step 3: Do avoid “forever begin end” constructs in the testbench as infinite loops prevent the EDA playground from generating a waveform. Consider using long *for loops* instead as in the snippet below.

```
initial begin
    for(int i = 0; i < 100; i++)
        begin
            #10; clk = ~clk;
        end
end
```

Step 4: When the testbench completes execution, the waveform will show up. Now, click the GetSignals button and append the signals inside the testbench module (ignore other modules that show up in the Scope region) and then close this popup.



Step 5: Finally, you will be able to view the waveform (some part of the image is blurred intentionally).



Note: To revert to EPWave opening in a new browser window, set that option on your user page.