

Hspice Stimulus Types

There are two types of stimulus that we widely use in this class

1. The pulse mode
2. The piecewise linear mode

The following figure, shows the graph generated in response to the *pwl* (piecewise linear) input stimulus applied between **In** and **Gnd**.

Vin1 In Gnd pwl(0ns 0v 35ns 5V 35.1ns 0V 55ns 0V 55.1ns 5v 89.9ns 5V 90ns 0V)

This means that

At 0ns we are at 0V (point A)

At 35 ns we are at 2.5V (point B)

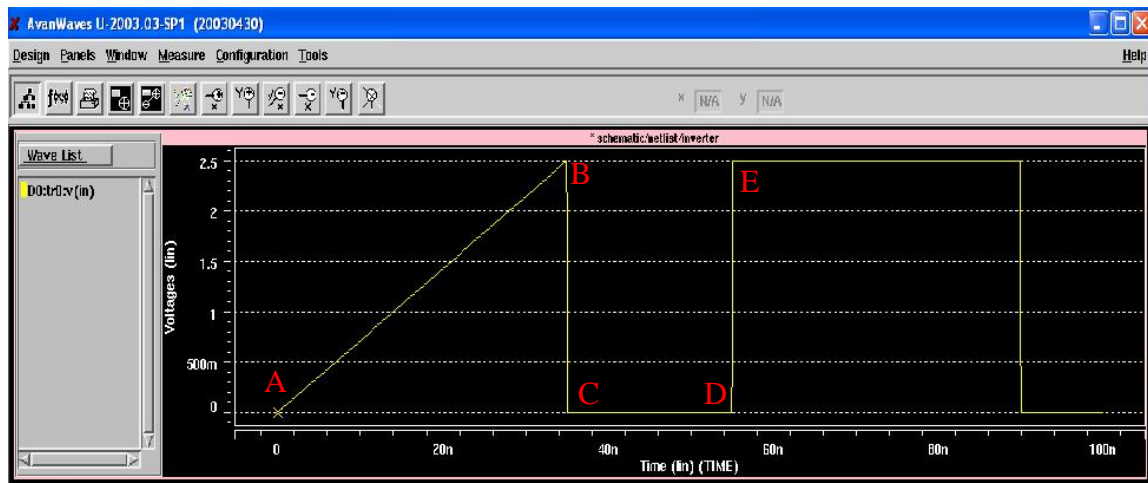
At 35.1ns we are at 0V (point C)

At 55ns we are at 0V (point D)

At 55.1ns we are at 2.5V (point E)

.....and so on

Now all these points are joined with straight lines to generate the following stimulus graph:

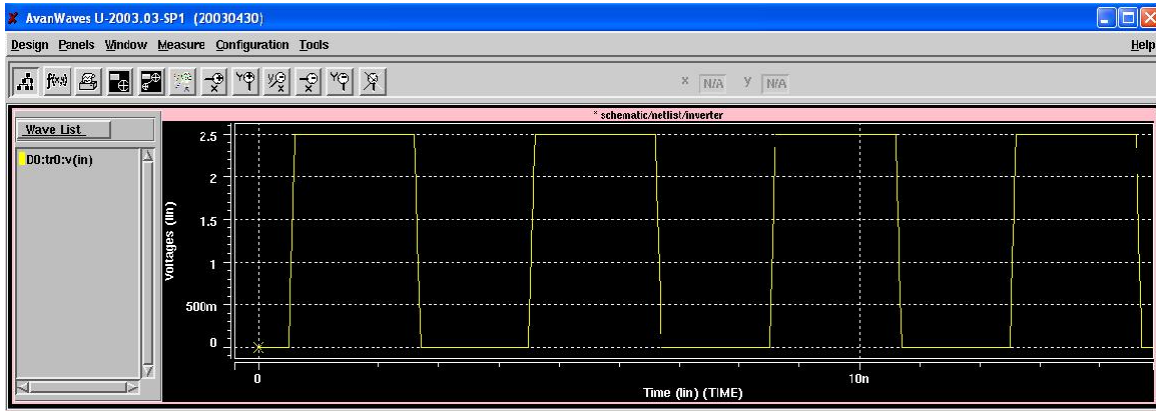


This stimulus is applied to the **In** input of the Inverter gate. The following figure below, shows the stimulus generated in response to the *pulse* input stimulus.

Vin1 In Gnd pulse(0v 2.2v 0.5n 0.1ns 0.1ns 2ns 4ns)

This means that you are creating a pulse waveform, Varying from 0V to 2.2V, with a delay of 0.5ns with a rise time of 0.1ns, with a fall time of 0.1ns, with a time period of 4ns and a high level time of 2ns. A more general form of the syntax is:

Vin1 node Gnd pulse (level 1, level 2, delay, rise time, fall time, time"level 2" is maintained, time period)



This stimulus is applied to the **In** input of the Inverter gate. Now you can find various delays (such as high-to-low and low-to-high propagation delays) by adding **PointToPoint measures** from **Measure** menu.

SWEEP

Sometimes you have to analyze the behavior of the circuit on some range of parameters. For example you may have to analyze the behavior of the inverter with an output load (**CAP**) range of 0-50fF. For this you have to use a sweep command. The syntax of the sweep command looks like

```
.TRAN 0.1n 20n SWEEP DATA=D
```

```
.DATA D
```

```
CAP
```

```
0fF
```

```
5fF
```

```
10fF
```

```
15fF
```

```
20fF
```

```
.
```

```
.
```

```
.
```

```
.
```

```
.ENDDATA
```

The above example asks spice to run a transient simulation using **.TRAN** command and sweep the simulation on the data given in D using **SWEEP** command.

The **.DATA** statement specifies the parameters for which values are to be changed (**CAP**) and gives the sets of values that are to be assigned during each simulation (**0fF, 5fF, 10fF, 15fF....**). The required simulations are done as an internal loop. This bypasses reading in the netlist and setting up the simulation, and saves computing time. Internal loop

simulation also allows simulation results to be plotted against each other and printed in a single output.

.MEASURE

.MEASURE command is used to print out user defined electrical specifications of a circuit like propagation delay, rise time, fall time etc. An example of measuring propagation delay of an inverter using .measure command is shown below

.MEASURE DELAY

```
+TRIG V(IN) VAL='SUPPLY/2' RISE=1  
+TARG V(OUT) VAL='SUPPLY/2' FALL=1
```

Where

.MEASURE

asks hspice to measure

DELAY

Name that is associated with the measured value in the output

TRIG

defines the starting point

TARG

end point

VAL

defined the starting value(ending value) from(upto) which the delay has to be measured. In our case it is 50% of the signal IN(OUT) rising(falling)

RISE(FALL)

The numbers indicate which occurrence of a CROSS, FALL, or RISE event causes measurement to be performed. For RISE=*r*, the WHEN condition is met and measurement is performed when the designated signal has risen *r* rise times. For FALL=*f*, measurement is performed when the designated signal has fallen *f* fall times.

The measured value will be saved in the file named as *file_name.mt0*