

## Spice Opus can not handle Transient Response?

After having entered the following lines in the Spice Opus:

```
piceOpus (c) 1 -> c:\spice
c:\spice: no such command available in SpiceOpus (c)
SpiceOpus (c) 2 -> cd c:\spice
current directory: c:\spice
SpiceOpus (c) 3 -> source ttlinv.cir
Circuit: * this is a ttl inverter subcircuit
```

```
SpiceOpus (c) 4 -> tran 0.01us 20us
```

**Numerical oscillations (trapratio) detected at time=1.079141e-006, xmu temporary decreased (xmumult).**

```
SpiceOpus (c) 5 -> plot v(2)
```

It gives a warning and continues plotting the following, for a standard ttl inverter:

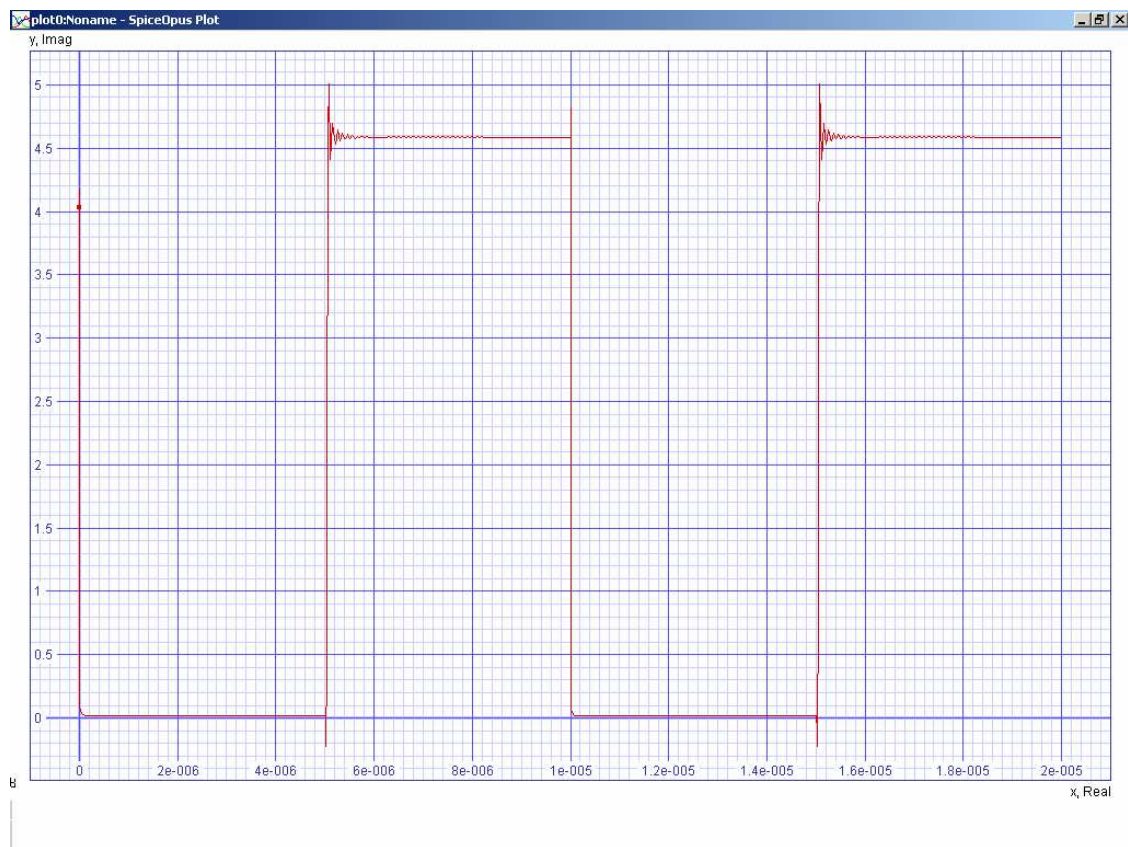


Figure 1 Spice Opus output

Then  $V_{OH}$  turns out to be 4.5. Yet the Voltage Transfer Characteristics (VTC) is already known as plotted in Figure 2.

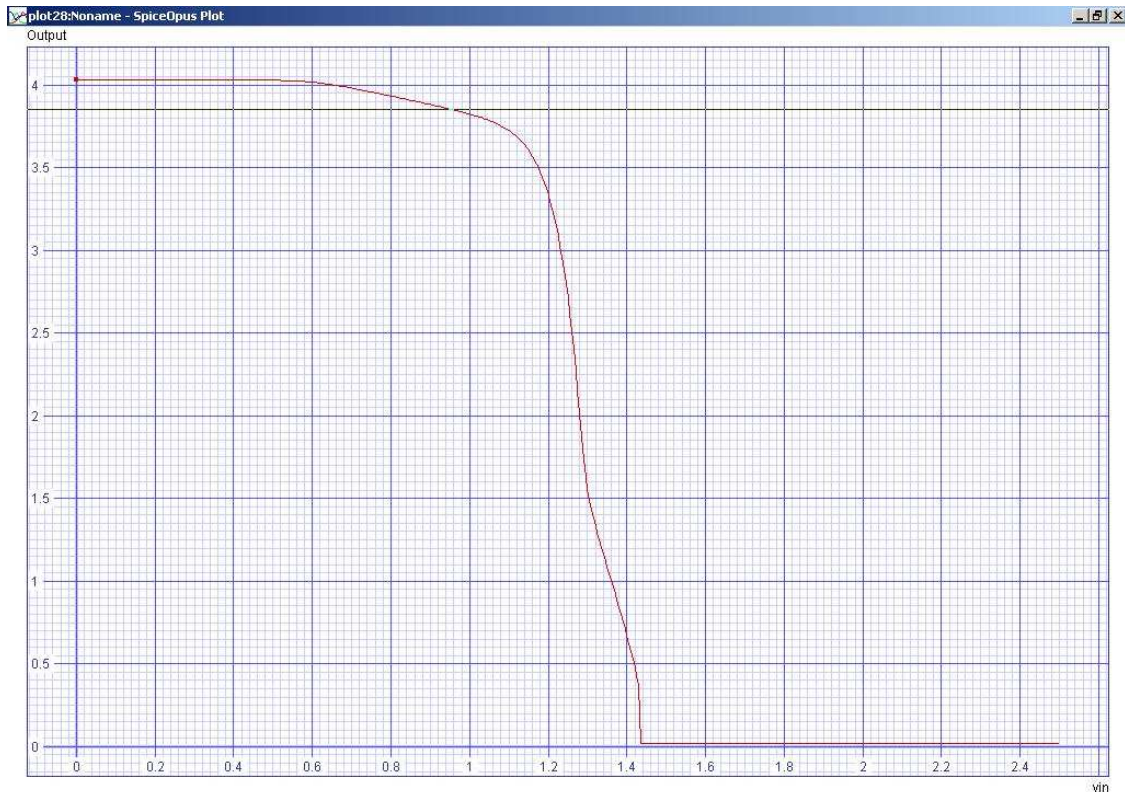


Figure 2 VTC of the TTL inverter

Contradictory thing is that  $V_{OH}$  is around 4 Volts. These two values can not be true at the same time. 5Spice simulations reveal similar outputs for the VTC. As it can be depicted from Figure 3,  $V_{OH}$  is again 4 Volts, which seems to be the meaningful answer. The same codes and subcircuits have been inserted into 5Spice.

*Actually it turns out that there is a stable region where both of the simulation programs give similar results, but outside that region, we are again not in the safe region. Depicted in Figure 4.*

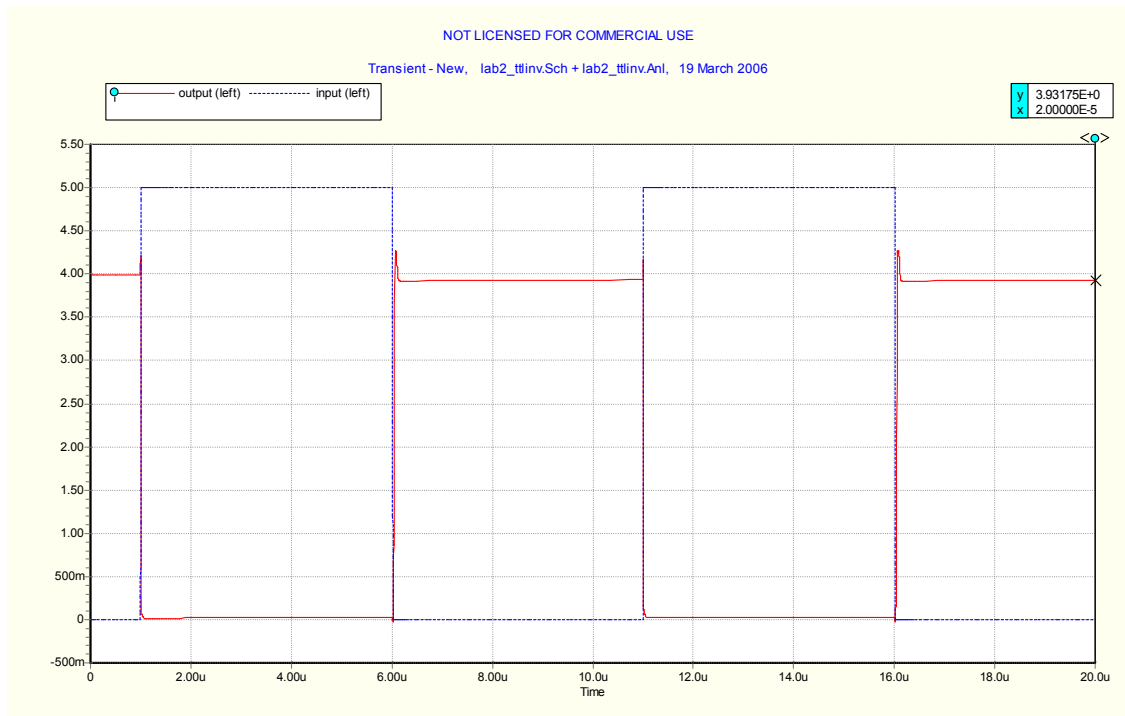


Figure 3 Transient Response according to 5Spice

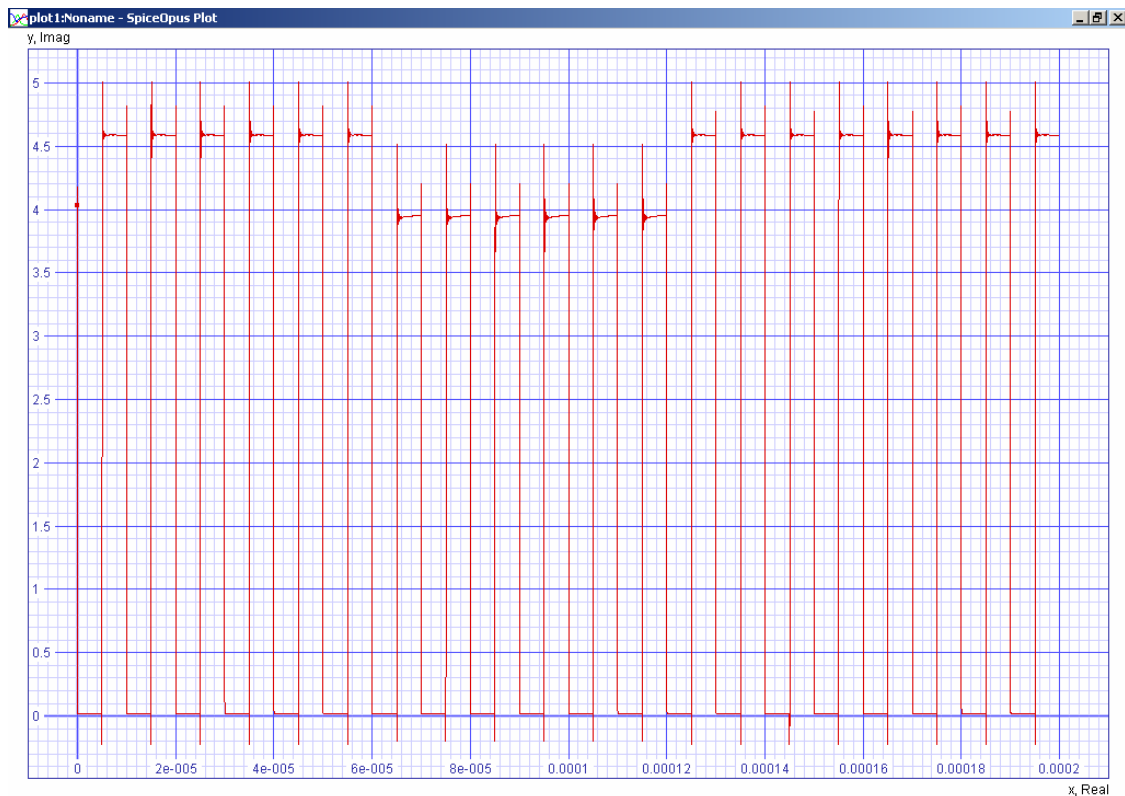


Figure 4 Stable Regions in Spice Opus