DC Simulations using SPICE :

DC Sweep (.dc)

- This performs a dc analysis while sweeping the dc value of an independent variable (voltage , current , temperature etc.)

- It is also useful to compute the DC Transfer Function of a circuit , or plotting the current voltage characteristics.

Syntex :   .dc <srcnam> <Vstart> <Vstop> <Vincr> + ...
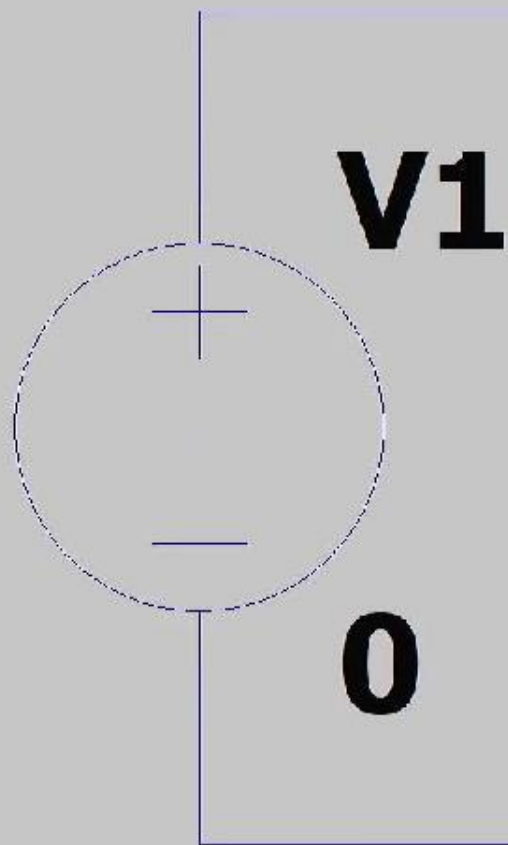
DC Simulations using SPICE :

DC Sweep (.dc)

- This performs a dc analysis while sweeping the dc value of an independent variable (voltage , current , temperature etc.)

- It is also useful to compute the DC Transfer Function of a circuit , or plotting the current voltage characteristics.

Syntax : .dc <srcnam> <Vstart> <Vstop> <Vincr> + ----- .

- DC sweep under various junction temperature

Syntax : .TEMP <T1> <T2> -----

         . STEP TEMP LIST <T1> <T2> -----

LTspice - [Draft1]

File   Edit   Hierarchy   View   Simulate   Tools   Window   Help

V1

D1

0

5817

Configure Analysis

Transient | AC Analysis | DC sweep | Noise | DC Transfer | DC op pnt | Transient Frequency Response

Compute the DC operating point of a circuit while stepping independent sources and treating capacitances as open circuits and inductances as short circuits.

1st Source | 2nd Source | 3rd Source

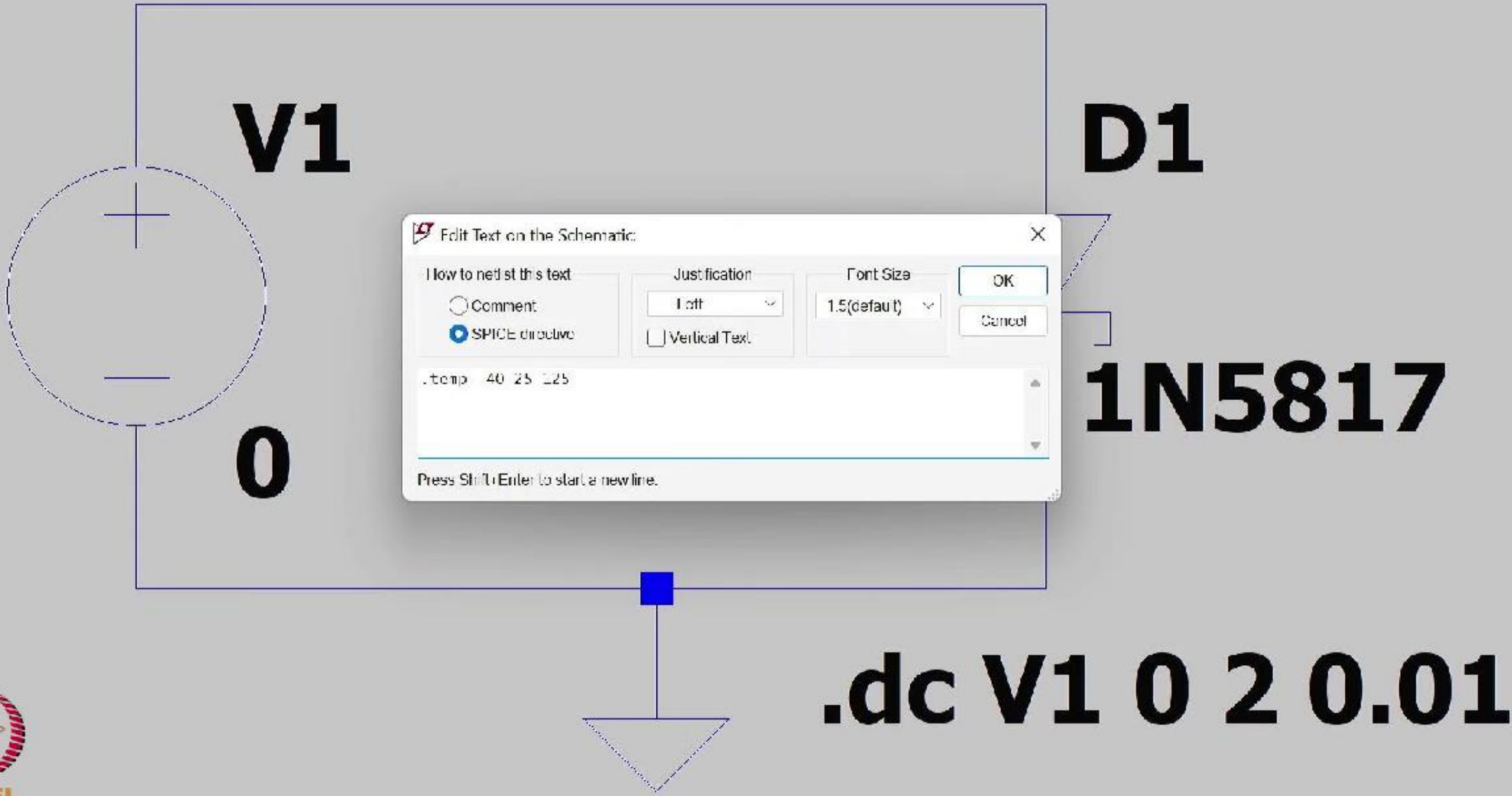Name of 1st source to sweep: V1

Type of sweep: Linear

Start value: 0
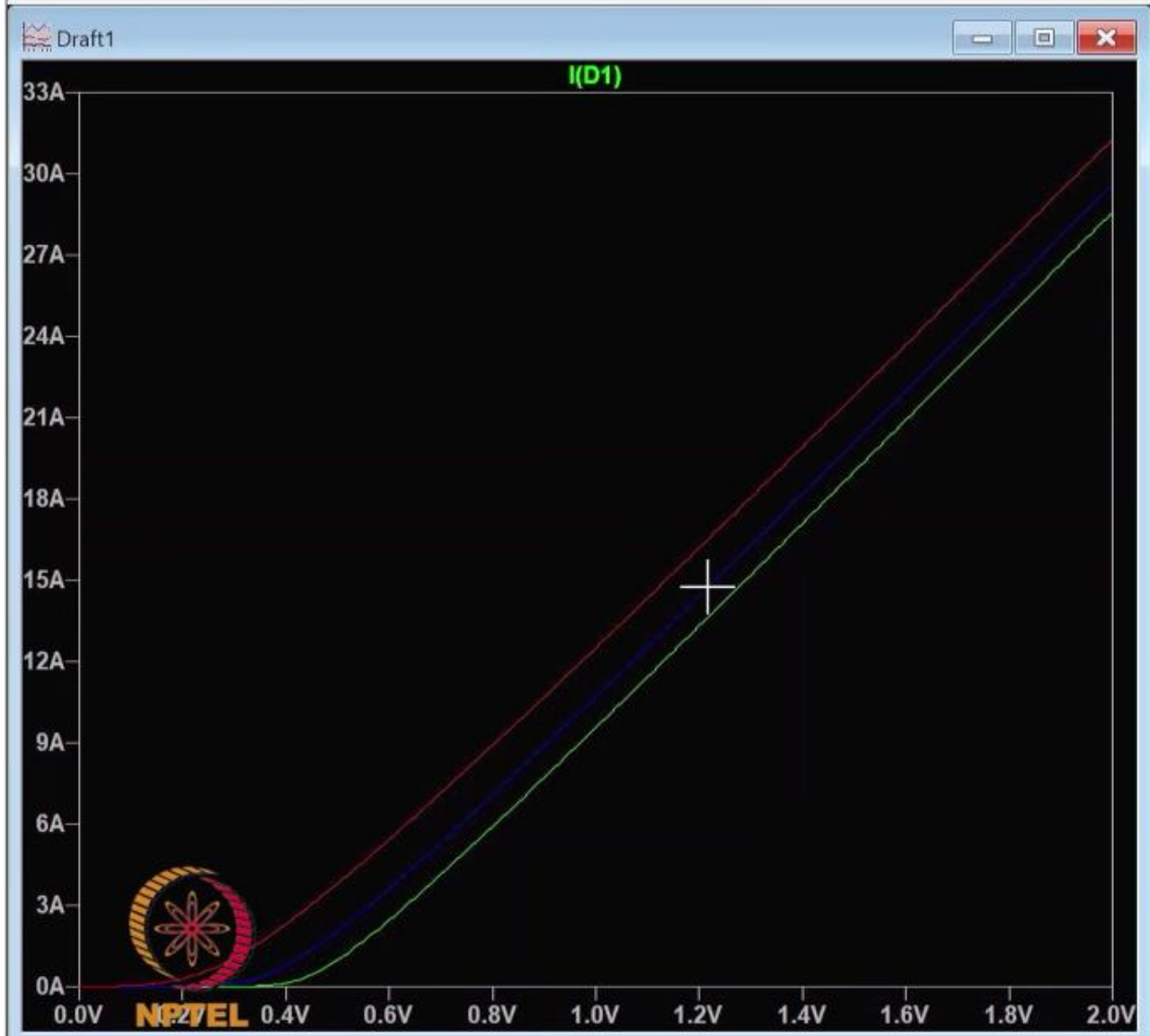
Stop value: 2

Increment: 0.01

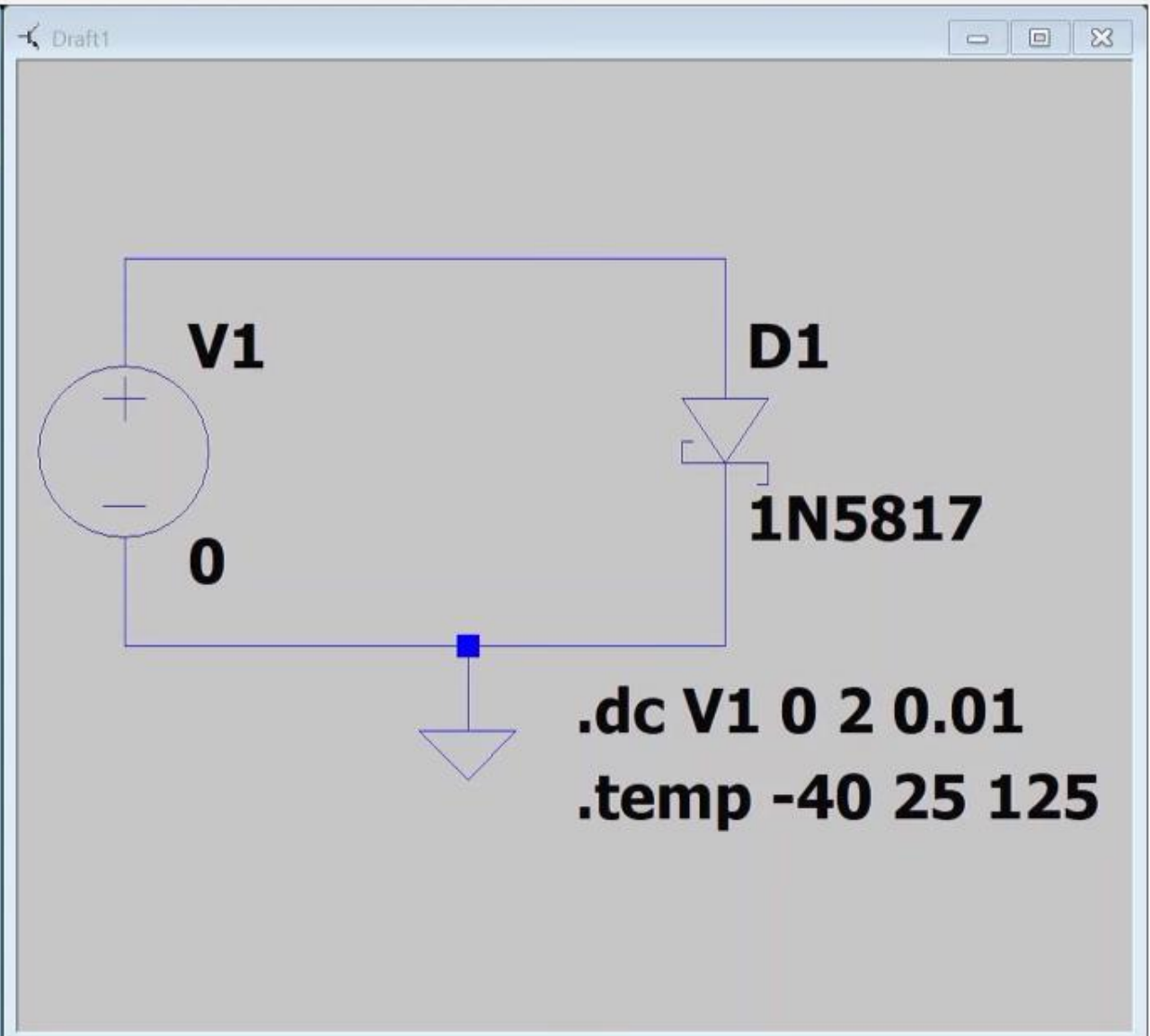Syntax: .dc [<oct,dec,lin>] <Source1> <Start> <Stop> [<Incr>] [<source2> ...]

.dc V1 0 2 0.01

OK   Cancel

NPTEL

Ready

V1

D1

+

1N5817

0

Edit Text on the Schematic   ✕

How to netlist this text

○ Comment

● SPICE directive

Justification

Left

☐ Vertical Text

Font Size

1.5(default)

OK

Cancel

.temp 40 25 125

Press Shift+Enter to start a new line.

.dc V1 0 2 0.01

NPTEL

DC Parameter Sweep :

Four types of DC sweep :

1) Linear   Equal intervals defined by increment.

2) Octave   Every 2 times   No. of octaves $= \log_2\left(\dfrac{\text{Stop value}}{\text{Start value}}\right)$

3) Decade   Every 10 times   No. of decades $= \log_{10}\left(\dfrac{\text{Stop value}}{\text{Start value}}\right)$

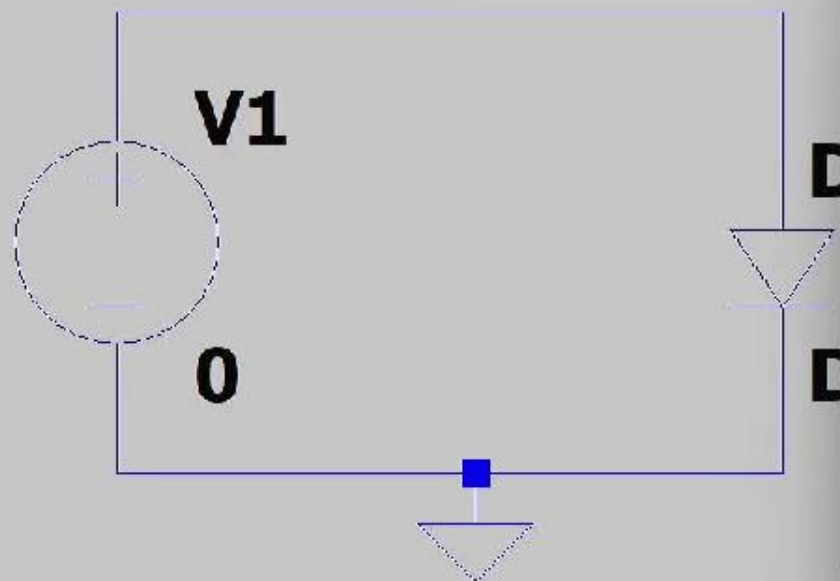4) List   Specified the list of individual values.

How to import third party models into LTSPICE ?

Two types of statement :

1)   .MODEL   ; intrinsic SPICE devices like diodes and transistors.

2)   .SUBCKT   ; collection of circuitry.

There are 4 methods to include

1) Method - 1 ;   Copy and Paste

2) Method - 2 :   Use .include   (or .inc)

3) Method - 3 :   Use .lib

4) Method - 4 :   Use URL   (link over internet)

V1

0

Component Attribute Editor

Open Symbol   C:\Users\Del\AppData\Local\LTspice\lib\sym\diode.asy

This is the second attribute to appear on the netlist line.

Attribute	Value	Vis.
Prefix	D	
InstName	D1	X
SpiceModel		
Value	DI_1N4001G	X
Value2		
SpiceLine		
SpiceLine2		

Cancel   OK

D

D

.MODEL DI_1N4001G D (IS=65.4p RS=42.2m BV=50.0 IBV=5.00u + CJO=14.8p M=0.333 N=1.36 TT=2.88u)

NPTEL