3.63   Given the following SPICE input file,

```
PROBLEM 3.63
V1 1 0 DC 20
R1 1 2 60
R2 2 0 30
R3 2 3 20
I1 0 3 DC 1
R4 3 4 10
V2 0 4 DC 15
.END
```

Realizamos la simulacion en lt spice: