The LTspice Transient Analysis Command

The phrase "transient analysis" means we're solving for functions of time, i.e., for the "time-domain" circuit response to some input signal. Selected voltages, currents, or other signals within the circuit can be displayed in the LTspice Waveform Viewer as the simulation progresses, much like an oscilloscope on a laboratory workbench. More simply, the `.TRAN` command directs LTspice to compute what happens within the circuit when it is powered up and run. Signals may be applied as independent sources or described tabularly by data stored in a file.

The command syntax can be as brief as:

```
.tran <Tstop> [options]
```

or, if more control is desired:

```
.tran <Tstep> <Tstop> [Tstart [ΔTmax]] [options]
```

The second form is the traditional SPICE `.tran` command. The <> brackets indicate "required" specifications, while the [] brackets indicate optional ones. In any case the brackets themselves are ***not*** included on the command line. T_{step} is the plotting (or tabular) increment for the waveforms and is also used as an initial step-size guess. LTspice uses a technique called waveform compression (https://ieeexplore.ieee.org/document/9180295), so the T_{step} parameter is of little value and can usually be omitted or set to zero. T_{stop} is the duration of the simulation, in seconds. Transient analyses always start at $t = 0$. However, if T_{start} is specified, the calculated waveform data between $t = 0$ and $t = T_{start}$ is not saved. This is a means of managing the size of waveform files by allowing startup transients to be ignored. The final parameter, ΔT_{max}, is the maximum time step LTspice is allowed to take while solving the circuit equations. If either T_{start} or ΔT_{max} is specified, T_{step} must be also. Otherwise, everything but T_{stop} can be omitted and left to their SPICE default values.

As shown above, several *options* can be included with the `.tran` command:

- **`UIC:`** Use Initial Conditions. Skip the DC operating solution and use user-specified initial conditions. Normally, a DC operating point analysis (in other words, a default `.op` calculation) is performed before starting the transient analysis, even though it is not expressly stated in the netlist. The UIC directive suppresses this initialization. Even so, the initial conditions of some circuit elements can be can be specified on a per-instance basis.
- **`steady:`** Stop the simulation when steady state has been reached.
- **`nodiscard:`** Don't delete the part of the transient simulation before steady state is reached.
- **`startup:`** Solve for the initial operating point with independent voltage and current sources turned off (but using any constraints specified by a `.ic` directive). Then start the transient analysis and linearly ramp these sources on during the first 20 µs of the simulation.
- **`step:`** Compute the step response of the circuit.