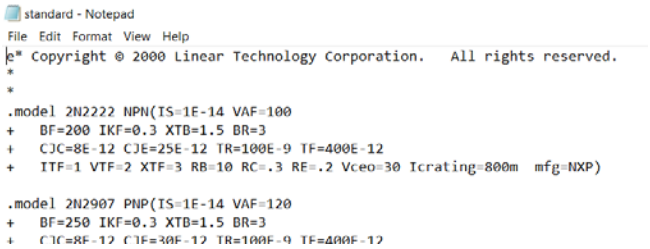
- Download and install LTspice from Linear technology website.
  - Place all three files from Learn in the same directory (any directory is fine).
- There is a bug in the LTspice software which is sometimes present (but not always). Open the standard.bjt file which is in the lib\cmp directory. An “e” may appear in the first line. You need to delete this before proceeding. Here is an example screen shot:



- Open run_TL494_test_CEH.asc. This will show an empty spot for the TL494 to be placed.
- Open TL494_CEH.sub in notepad. Enter the correct path names for the diodes, which depends on where LTspice installs the second folder. Save the file.
- Drag the TL494_CEH.sub file over to the run_TL494_test_CEH.asc circuit. A window will open showing all the code for the TL494 chip.
- On the blue line .subckt TL494_CEH DTC FB …, highlight TL494_CEH, right click on this and then “create symbol”, then click yes. This automatically creates a directory called “AutoGenerated” and the TL494_CEH symbol will be in there.
- To insert the TL494, click on component, navigate to AutoGenerated then click on TL494_CEH and press OK.
- You can now drag and drop the TL494 LTspice chip into the empty spot.
- To run it, click on the Control panel -> SPICE.
- Change Gmin, Abstol, Chgtol and Volttol to 1e-006, then click on the “Gear” Integration Method and click OK. Note that this step is very important otherwise there are errors in the ode45 solver and it runs very slowly.
- The simulation will now run and for the test circuit it’s very fast.
- To run the control circuit, just click on component and drop the TL494_CEH into the empty spot. This circuit takes about 20 seconds to run.