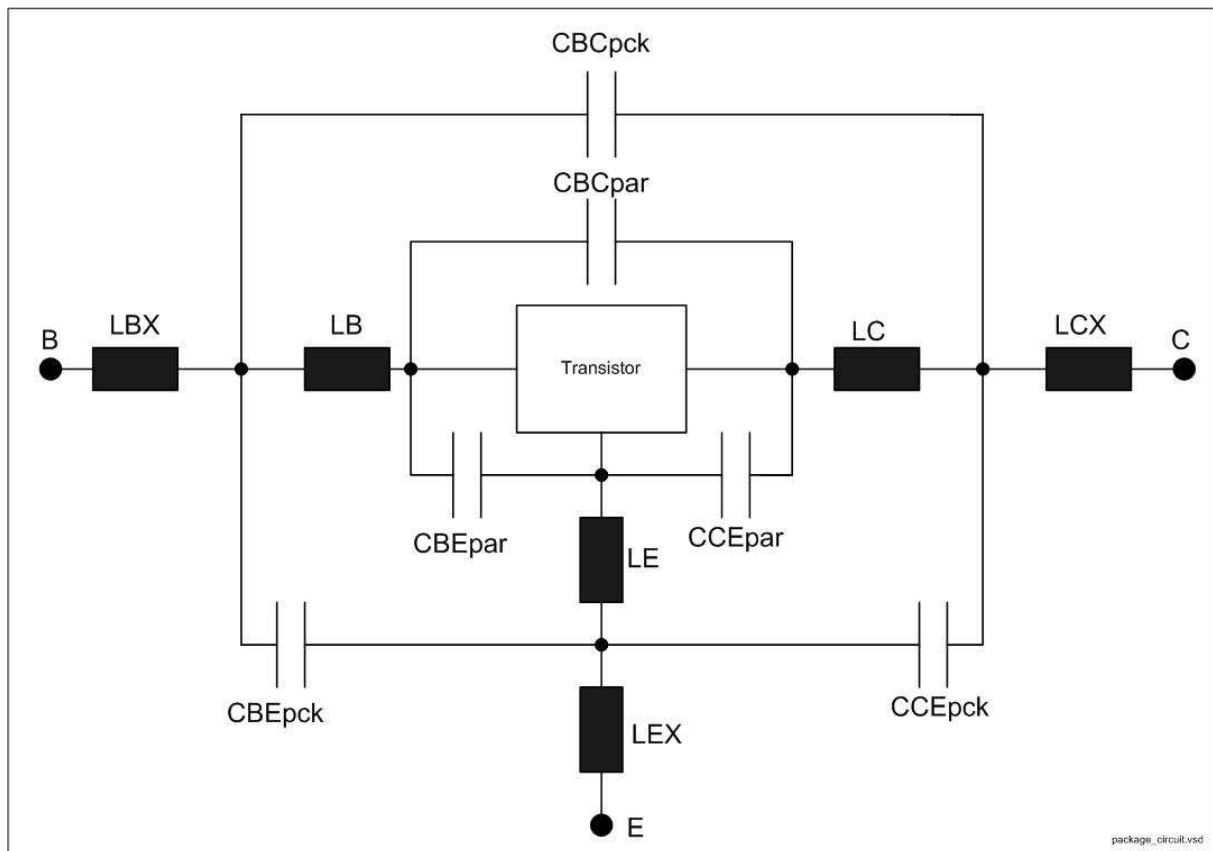


```

*****
* Infineon Technologies AG
* GUMMEL-POON MODEL IN SPICE 2G6 SYNTAX
* VALID UP TO 10 GHZ
* >>> BFP740ESD <<<
* (C) 2010 Infineon Technologies AG
* Version 1.0 June 2010
*****
* - Please use the global SPICE parameter TEMP to set the junction
* temperature of this device in your circuit to get correct DC
* simulation results.
* - TEMP is calculated by  $TEMP = TA + P \cdot (R_{thJS} + R_{thSA})$ . The junction
* temperature TEMP is the sum of the ambient temperature TA and
* the increment of temperature caused by the dissipated power
*  $P = V_{CE} \cdot I_C$  ( $I_C$  collector current,  $V_{CE}$  collector-emitter voltage).
* -  $R_{thJS}$  is the thermal resistance between the junction and the
* soldering point.  $R_{thJS}$  for this device is 325 K/W.  $R_{thSA}$  is the
* thermal resistance of the PCB, from the soldering point to the
* ambient. For determination of  $R_{thSA}$  please refer to Infineon's
* Application Note "Thermal Resistance Calculation" AN077.
* - The model has been verified in the junction temperature range
* -25°C to +125°C.
* -  $T_{NOM} = 25^\circ\text{C}$  is the nominal ambient temperature.
* Please do not change this value.
*****

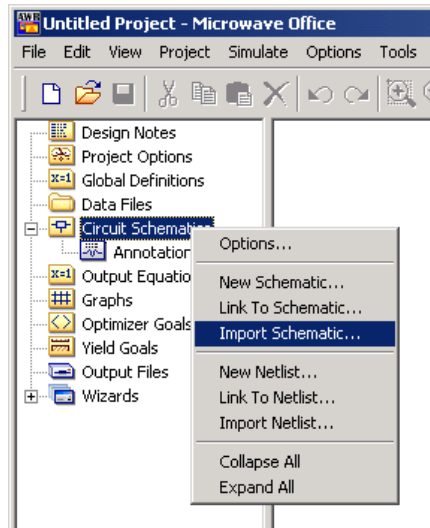
```

Only for completeness reasons below you can find the package model for SOT343. The package parasitics are already included in the model, no matter what format you use (SPICE syntax or ADS- or MWO-format). The models are ready to use!



Import Schematic in MWO (Microwave Office from AWR)

Right mouse click on Circuit Schematic, then select Import Schematic and go to the folder where you have stored the MWO-Schematic (*.sch) of the device.



Import SPICE-Model in ADS (Advanced Design System from Agilent)

Got to File -> Copy Design... -> set the path of the folder where you have stored the ADS-File of the device and select the "*.dsn" file. Set also the path of your current project. Then click OK.

