



ICAP/4 Version 8.x.11 Build 4545

2500 Via Cabrillo Marina, Suite 208
San Pedro, CA 90731 USA
Tele: (310) 547-0197
Fax: (310) 547-1096
sales@intusoft.com

**Compatible with all 32bit and 64bit Windows
including Windows XP/Vista/7/8 and Win 10!**



**Check www.intusoft.com
for software updates!**

Web Download Option:

If you are currently on maintenance, you can also obtain this update via the web. Simply go to <http://www.intusoft.com/support.htm> and enter your serial number to obtain this update. Your serial number is on the "Installation Letter" that came with your initial purchase. You can also find your serial number from the Help pulldown, "About SpiceNet," then the "User Tab" dialog within the Schematic program. Should you have any questions about the status of your maintenance program, please contact Intusoft at (310) 547-0197.

New Features and Enhancements

**Note: Product features below are not necessarily available on all ICAP/4 software packages.
For a more detailed list go to: <http://www.intusoft.com/products/icapscomparison.html>**

CMSDK: Starting with Build 4545 all ICAP/4 simulation packages(excluding RX) will include our Code Model Software Development Kit (CMSDK). The CMSDK package has been introduced to allow designers to develop and debug HDL models for use with IsSpice4. It allows the designer to add new analog, digital, or mixed analog/digital models to IsSpice4 very easily. Adding a model to SPICE in the traditional manner can take several months. The Intusoft CMSDK cuts this time to days or even hours. CMSDK modeling is also called "Code Modeling" because models are developed using C subroutines. The use of a standard programming language like C greatly lessens the learning curve since users do not have to learn a proprietary language or use unfamiliar tools.

IsSpice4 users have been using many internally developed code models as part the simulation process by accessing the code model DLLs located in the /spice8/is folder. Starting with Build 4545 they get the ability to develop their own DLLs. No extra cost is added with this addition and now users can compile their code models with publicly available compilers. Please see the CMSDK getting started note for details.

New Simulation Options: These options could be set in the following tab.

Simulation Setup->Simulation Options->More->Screen/File

1) SaveOP: this options allows a user to save the bias (DC operating point) of a simulation and use it in subsequent simulations. It's particularly useful for circuits that take a long time to achieve DC convergence.

Usage :

.option saveop = filename

to save the operating point. For example:

.option saveop = dcop.txt

Afterwards use:

.include filename

in the "Simulation Setup"->"User Statement:" pane to load the saved operating point.

For example:

.include dcop.txt

Note that you should remove the saveop statement after the initial run from your setup to save time.

2) SrcSteps: This option controls the number of source steps to try for OP, default is 10.

3) GminSteps: This option controls the number of Gmin steps to try for OP, default is 10.

4) DcConv: Use this option with hard to converge circuits to get an operating point. Note that simulation accuracy might be severely affected by using this option. It could be used to get an initial value to be used in subsequent .nodeset statements(using SaveOP option) to achieve accurate DC convergence in subsequent simulations by removing the DcConv option.

5) FastTran: use this option to get significant runtime improvement with long transient simulations without much loss in accuracy.

Example: C:\spice8\circuits\powerDemo\FlybkTemplate\FlybkTemplateDemo.dwg

Use TranSubckt and tranAvg 1m settings to compare simulation run times with and without

.option FastTran

New SPICE models Added: Including Apex Opamp models and Cree LED models.

Keep your Maintenance Current to Receive Tech Support and the Next ICAP/4 Release.