



# NEW USER'S TUTORIAL

Welcome to ICAP/4, Intusoft's suite of analog and mixed-signal simulation products. If you are using ICAP/4's demo CD, it premieres many of ICAP/4's capabilities, but you are limited in the component model count and a few capabilities. Within the demo CD, more than 1,450 models are provided from ICAP's collection of 23,500 parts, plus advanced analyses such as Monte Carlo statistical analysis, Magnetics Designer transformer and inductor design option, design optimizer, and component parametric sweeping.

Let's get started and test drive ICAP/4. This short guide will enable you to view an existing circuit drawing, enter a new drawing, set up a design simulation, view signal waveforms, modify your design until satisfied with its operation, and configure the windows environment to modify and save the way application windows come to view each design session. See if you agree that ICAP/4 is today's best and most powerful analog and mixed-signal offering. Intusoft guarantees this by providing a 30-day money back policy on all of its software, except for ICAP/4 Consumer.

- 1) Notes: The word "click" or "double click" implies pressing the left mouse button accordingly, unless stated as the right button. If you ever lose sight of the schematic, simulation, or waveform-display windows, simply left-click the mouse on their corresponding "icon" boxes at the bottom of your desktop to bring them to view. The boxes will be labeled "SpiceNet" for the schematic; "IsSpice4" for simulation windows including real-time marching waveforms, output file and error file; and "IntuScope," the full waveform-viewing tool. Resizing these windows so they come to view this way each time is covered in Step 12. In the extremely unlikely event that any of ICAP/4's programs go awry, invoke the "Icaps" icon at the bottom of the desktop toolbar and "Exit Icaps."
- 2) Select the "Start" box at the bottom left corner of your computer's desktop. Select "Programs," "ICAP\_4 Demo," and "Start ICAPS." This brings to view the ICAP/4 schematic and simulation environment, all in an integrated window shell labeled, "SpiceNet" in the banner. Next, select from the "File" pulldown, "Open." Browse to C:\spice8d\Circuits\PowerDemo\FlybkTemplate" and double-click on the FlybackTemplateDemo.DWG file. This brings the drawing to view. Next, go the middle of the top toolbar and you will see a small white window. Pulldown from its arrow and select "avg." This illustrates the top level of a Flyback SMPS power supply design. The schematic is comprised of several circuit layers, all visible in this drawing. You can scroll around the design by using the window's scroll bars, or by holding down the left mouse button on a blank area of the drawing and dragging the mouse in a desired direction. You can zoom in or out by first left-clicking on an area of interest on the drawing, then pressing the <F7> or <F8> keys. You can also left click the zoom icon on the right toolbar (magnifying glass), then hold down the left mouse button and drag around an area you want zoomed on the schematic, then release the mouse button. The <F6> key will fit the drawing within the schematic window. Finally, holding the Ctrl key while moving your mouse roller up or down (if equipped with a roller) will zoom in or out.

Note that the actual schematic window resides inside the "SpiceNet" outer shell window. For schematic resizing to work as intended above (i.e., F6, F7 and F8 keys), the inside schematic window should first be brought to full size within the outer SpiceNet shell window (if necessary) by pressing the inside schematic window's "maximize" icon. Let's next look at another schematic layer that is part of this design.

- 3) Hold down the left mouse button for a moment inside the op-amp located in the lower left area of the design. A gray shadow box appears around the op-amp. Briefly hold the left mouse button down on the “eye-looking” icon in the bottom toolbar of the screen (left). The corresponding layer of the design highlights in pink and its name (Subsckt) is indicated to the left of the eye. Using SpiceNet’s powerful layering system, a designer can begin creating a particular schematic layer, such as the one you highlighted, then add on other circuit portions in new layers. Wiring between layers is easy because visibility between all layers is maintained during the wiring process. Configurations (Options pulldown, “Configurations,” “Edit”) save designs as different revisions and enable instant reuse or modification to the circuitry.
- 4) Next let’s observe a sub drawing under a high-level component from the Avg design hierarchy. Right-click inside the “FLYBACK\_ID” component toward the top of the design, then select “Push into subdrawing.” The detailed design that resides under this component is now illustrated. Close this drawing by left-clicking the mouse on the “close” icon on the top right of the drawing’s banner.
- 5) Let’s pull down another circuit within the design hierarchy from the small white configuration window used previously. Select, “AvgSubckt.” Copy a component from within this design and connect the new part back in with wires. For example, select a resistor and hold down the left mouse button over it until it is highlighted. Release the mouse. Press the <CTRL C> keys to copy the resistor. Left click the mouse on the schematic where you desire the component to be placed, then press the <CTRL V> keys to paste it. As an alternative, you can hold the left mouse button down and drag a boundary box around the resistor, or any group of circuitry, and copy/paste using the previous steps. In this process, you may have unintentionally encompassed wires alongside the moved part(s). This is no problem because you can use them in connecting the parts into the schematic. Or, you can area-select a wire(s) and press the <Backspace> key to delete. Also, the <Ctrl Z> keys delete any steps you’ve taken, one step each time indefinitely. Finally, click on the second icon from the top of the far right toolbar. This brings up a wiring cursor. Simply hold down the left mouse button and drag the wire pencil from the desired beginning point (i.e., component leg) to an end destination such as another component or wire in the existing circuit. To exit this “wiring mode,” select the top icon on the right toolbar or simply press the <Esc> key.
- 6) If you double click on any component, its parametric information is revealed in a “Component Properties” dialog box. The selections within this box enable you to ascribe or change component values and parameters. Double click the resistor you placed (or any resistor) and observe the second entry in its Properties dialog box. There lies its current value. It should be highlighted, so to change it, press the <Backspace> key and type in a new value. Try changing other values that are permissible. To exit, press the “OK” button to save changes, or the “Cancel” button to ignore them.

Double click on voltage source (V2) toward the bottom mid-schematic. Double-click on “Pulse” next to “TRAN Generators” toward the middle of the Component Properties dialog box. Observe its specifications, then select some of the tabs on top to see different input waveforms for construction. Close all dialog boxes by pressing their “Cancel” button. Note that 30 types of signal generators are available from the “iGenerators” Part-Type field inside SpiceNet’s parts browser (just press the <x> key to bring it to view). Finally, right click on a component and note with many of the menu items listed, if selected, a “help” button is furnished inside their dialog box. Along these lines, left click on the context-sensitive help icon (black question mark) on SpiceNet’s top toolbar. Then, select any of the menu items from SpiceNet’s toolbars, and left-click on a desired function. Help for that selection will come to view for most items. Note selecting an entity directly on the schematic with the black question mark will invoke general SpiceNet help.

- 7) Let's place a new component onto the design. Select the "Parts" pulldown from the top menus. "Passive" and "Active" are chosen for generic parts, which contain no specific part number information. Notice that resistors and capacitors (common electronic components) lie in the "passive" menu. Real parts with a part number, or specialized components, are chosen from SpiceNet's "Parts Browser," which is positioned as the initial selection from this "Parts" pulldown. Select this and a dialog box will come to view showing all the types of part categories. Left-click any of the "Part Types" and observe corresponding selections in the "Sub Type" window. Then, specify any component from the "Part List" window. Finally, select the "Place" button. Alternatively, you can use the "Parts Browser" icon on the top toolbar, which looks like an open book, or simply push the <x> key on the keyboard. Re-invoke the parts browser and observe the "Find" box inside the dialog box. Left click on it and note that you can type in an actual part number (i.e., 2n2222), or a portion of a part number, and the browser will find all such component instances.
- 8) Close the SpiceNet schematic window (not the outer SpiceNet shell window) by pressing the "close" icon in its top right banner. Remember, you have placed new components, perhaps with dangling wires, so select "No" when queried to save changes. Reopen the FlybackTemplateDemo.DWG file. Select the "AvgSubckt" configuration as you did in Step 5. To its right lies another small white window. Pull down the "tranAvg 1m" as a simulation setup configuration.

To observe how a simulation is setup, under the "Actions" pulldown menu from the "SpiceNet" window, select "Simulation Setup" then "Edit." As an alternative, you could have selected the simulation-setup icon in the upper toolbar (shaped like a pencil on top of a waveform). Inside the dialog box, the "Transient" box is checked, so double click on it if you want to observe its simulation setup information. Click on other analysis boxes to see how they're set up, or would be setup. "Analysis" and "Convergence" wizards are selectable near the top of the dialog box. The "Analysis Wizard" is designed to help a user in setting up a simulation. The "Convergence Wizard" is used to guide a user automatically if changing SPICE convergence parameters becomes necessary on very complex design. When finished, press "Done" to close the main dialog box. Press the "Run Simulation" icon from the top toolbar, which looks like a person running. An "IsSpice4" window comes to view. A blue window inside the IsSpice4 shell provides a thumbnail sketch of real-time marching waveforms as the simulation progresses. Other windows provide a simulation output file and any error information. Note that a dialog box comes to view titled, "After Simulation Go To..." This enables either the schematic, IsSpice4 (simulator), or IntuScope window to be active (visible) following design simulations. Select the SpiceNet choice and have the "show this dialog after each simulation" box unchecked. You can always change your selection by recalling this dialog box from the IsSpice4 window's "Options" pulldown menu. Press OK. Now left click on the IsSpice4 window's banner. If a "Simulation Control" window appears, on its banner's top right corner click the "minimize" button. Again, click on the IsSpice4 window banner to make it active. Press the <CTRL T> keys to rescale the waveforms. To have these windows appear in the future as they are positioned now, or any way you reposition them, select from the IsSpice4 window's "Options" menu pulldown, "Save Preferences."

- 9) Let's insert a test point at a desired node by left-clicking on the main portion of the schematic (not inside a subcircuit). Select the "Parts" pulldown at the top of the window. Choose "Test Point" and "Voltage." This will insert the test point where you left-clicked on the schematic. Next, wire that test point to some portion of the circuit. Select the "Options" pulldown and "Rubberbanding." Carefully hold down the left mouse button exactly on the test point symbol (horizontal "V") and drag it a short distance away from its present location. Wire connectivity will be maintained. Try other components.

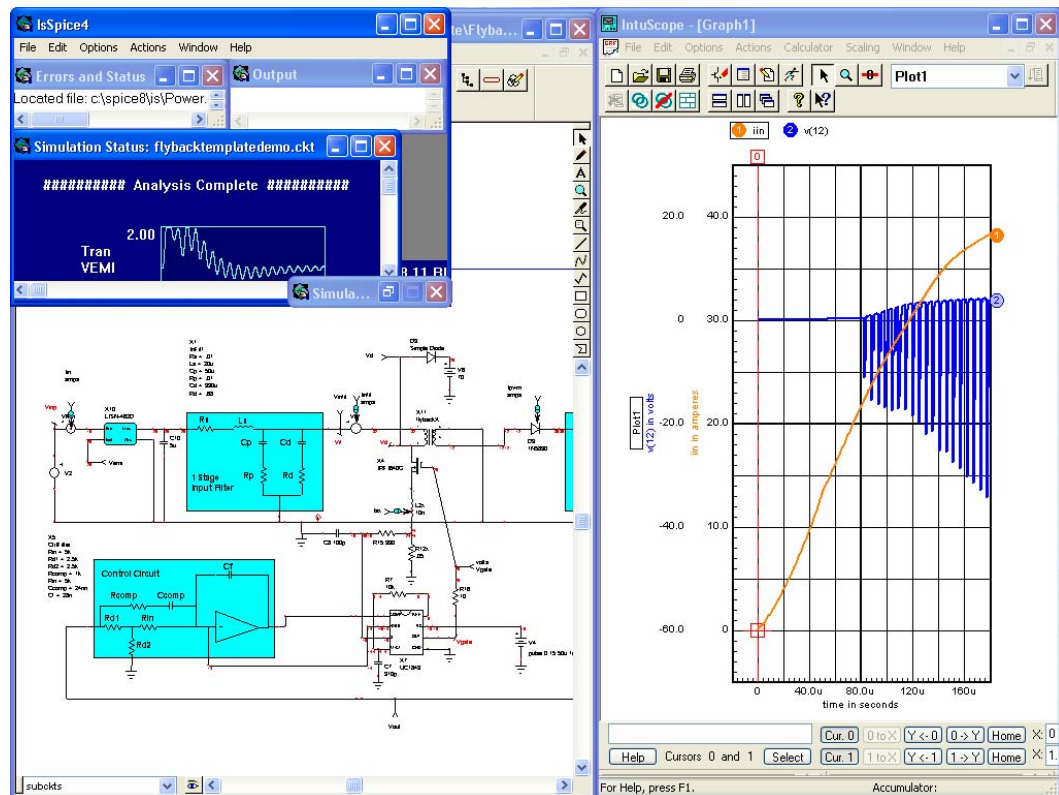
- 10) There are several powerful debug features available from the “IsSpice4” window, such as schematic-less sweeping of components. Those are covered in the Getting Started manual. Let’s now view waveforms using ICAP/4’s powerful waveform viewer and signal processor called, “IntuScope.” Note that simulation functions, such as “simulation setup,” “IntuScope,” etc. are selectable by icon or from the “Action” pulldown of the “IsSpice4,” “SpiceNet” and “IntuScope” windows. The "Scope" (IntuScope) icon looks like the crosshairs as found on an oscilloscope, and is located on the SpiceNet top toolbar. By the way, the three icons to the right of the "Scope" icon turn on or off, and refresh, the design’s DC operating point display on the schematic. Try these. If you press refresh after selecting an op pt. display, then click on the schematic afterwards, the operating point data will assume colors. Next, click on the Scope icon. IntuScope’s "Add Waveform" window comes to view. Inside this, click on any of the design node names or components shown in IntuScope’s "Y-Axis" window. Each time you select a node or component, click the "Add" box at the lower left corner of this dialog box. If you desire the schematic window, IsSpice4 windows, and/or IntuScope windows to be viewed simultaneously, this will be addressed in Step 12 below. Remember, their boxes are always accessible from the bottom desktop.
- 11) Cross probing to view waveforms directly from the schematic is simple. Optionally, first delete any waveforms you may have added in IntuScope’s viewing window by clicking on any of their names on top and pressing the <delete> key on your keyboard. Select the "SpiceNet" box at the bottom of your screen to bring the schematic to view. At the toolbar on the far right of the schematic page, select the icon that’s fifth from the top, called "Probe." When you want to deselect Probe, select the toolbar’s top icon, or press the <Esc> key, or right click the mouse. If you left-click the Probe cursor on any circuit wire or component, a corresponding waveform will automatically appear in IntuScope. Try a few. If you cross-probe while holding down the <Shift> key, the waveform will appear directly on the schematic. Probe a few entities in the design. Then (if necessary) select the "IntuScope" box at the bottom of your screen to bring it to view to observe the waveforms. If the waveform data is too packed to read clearly, select the zoom icon (magnifying glass) in the IntuScope window. Hold down the left mouse button and drag a rectangle around the desired waveform section. Repeat if necessary. Next, find the “Link All Y” and “Unlink All Y” IntuScope icons on its top toolbar (hold the mouse cursor over) and see what effect they have on the display. Try other icons.

There are numerous functions provided in IntuScope’s waveform calculator, all useful for instant signal processing. View the many selections from the "Calculator" pulldown menu by first left-clicking on a waveform name at the top of the window. Then try some of the functions from the calculator menu and see the results instantly display. If you hold down the left mouse button and drag the cursors (labeled “0” and “1”) horizontally across the waveforms, note the numerical information change toward the bottom of IntuScope. Finally, in the Add Waveform dialog box, you can select different types of simulation runs for waveform display from the “Type” box, if more than one analysis type was performed.

- 12) Finally, you may have had a preference on how various windows come to view each time during the above tasks. Advantageously, there’s a valuable way of customizing this, especially useful during repetitive simulation runs and waveform viewing.

First, maximize the inner schematic window. Then, resize the outer shell window by holding down the left mouse button while selecting the outer corner of the SpiceNet window. Drag the mouse to shrink the window to about two-thirds the screen size. Next, press the "fit to page" icon in the upper schematic toolbar (shape of a blank computer screen), or, simply press the <F6> key. This resizes the schematic to fit the window. Now, resize the IntuScope window in the same fashion. Next, hold down the left mouse button on

the SpiceNet window's top banner, and move the window to exactly where you want it on the screen. Repeat this for the IntuScope window. In both windows, select the "Options" pulldown and select "Save Preferences." Finally, bring the IsSpice4 window to view by selecting its box at the very bottom of the desktop. There are three to four separate windows here. First, shrink the size of the blue "simulation status" window so only the "Analysis Complete" message shows toward the top. Then, move this window toward the bottom of the main IsSpice4 window. Minimize the "Simulation Control" window (if it appears) by selecting the "minimize" icon in its banner. Reduce the "Errors and Status" window to a small size that's still readable. Reduce the "Output" window to roughly the same size as the "Errors and Status window." Finally, shrink the main IsSpice4 window to encompass all the other windows you modified. The figure below shows an example of this window environment. Try any re-sizing scheme you want.



### One possible window configuration preferences for debug cycles

From the "Options" pulldown menu in the IsSpice4 window, select "Save Preferences." From this point on, the schematic, IsSpice4, and IntuScope windows will all come to view in this fashion. You can resave preferences at any time to change the way these windows will come to view in the future.

A use for this "save preference" example would be for debugging a design. Example: Double click on a component to change its value or parametric data, re-simulate the design by selecting the simulation icon, then cross-probe a design node or component to bring waveforms to view in IntuScope. During this procedure, all windows will come to view as prescribed when you "saved preferences." Along these lines, in the IntuScope window, select the "Actions" pulldown and choose the "Update" menu. Select any of the last three functions. These enable new waveform data of previously viewed signals in your waveform plot to be automatically updated after a simulation run, simply by pressing the <Ctrl U> keys after a run is complete. Depending on which option you selected, previous waveform data will be overwritten with the new data, or appended to the same waveform plot, or inserted into a new plot.

- 13) Select the “AvgSubckt” configuration in the SpiceNet window. To its right, select the “tranAvg 1m” simulation setup. To automate the change of a component’s value or temperature (or design temperature) through a desired range, the "Alter" function located in the schematic’s right-hand toolbar can be used. Select the "Alter" icon. Note that once a simulation is initially run on the design, only then is the "Alter" function enabled. Place the magnifying-glass cursor that appears over C11 (find C11 by pressing <Ctrl F> keys), and left-click the mouse. In the “Alter” dialog box, select the “Linear” circle toward the top.
- Note: if you ever select the “Logarithmic” circle instead of the “Linear” circle, never ever type in a “0” for the data!** Hold down the left mouse button and highlight the existing value in the “Start” box. Press the <Backspace> key to delete it. Type in “1u.” Repeat these steps for the “Stop” box to the right and type “20u.” Type in “5u” in the “Step” box. Next, left-click “vout” in the “Y Axis” box toward the left of the dialog box. To its right, left-click the “Add to Plot List” box. Finally, click on the “Run” box at the bottom. A curve family is automatically produced in IntuScope for the four simulation runs representative of the sweep (this will take up to a couple of minutes). Use IntuScope’s Zoom function to see detailed waveform variations for the curve family of V\_2. Quickly try selecting any “link,” “tile,” and “cascade” icons in the IntuScope toolbar to see what effect they have in displaying the waveforms. Incidentally, ICAP/4 provides two other powerful methods of sweeping, one that produces marching waves while simply pressing a button to vary a component. Your Getting Started manual covers this feature.
- 14) Congratulations! You have now acquired enough basic skill to perform design and debug using ICAP/4 demo. Of course, the power and feature set of ICAP/4 offering at large extends far beyond the demo version capabilities. You can view a wealth of functions from the pulldown menus provided at the top of the schematic, simulation and IntuScope windows. To get your feet wet, left-click on the black question mark icon from SpiceNet’s toolbar. Then, left click on the “simulation control” icon (light blue square). Review the help information that comes to view, because the simulation-control dialog box will be a valuable feature as you progress into more advanced design-verification analysis. Your next step in learning more about using ICAP/4 to a fuller potential is to begin using the “Getting Started Guide,” which is accessed from SpiceNet’s help pulldown (select PDF Manuals). Also, there are movies on the design flow process under the Help pulldown; select "Tutorial Movies."
- 15) Please provide your feedback on this tutorial by emailing Intusoft at: [rmktg@intusoft.com](mailto:rmktg@intusoft.com)