
KCIDE for Circuits

Engineers of the twenty-first century will need to be able to work in a “knowledge capturing integrated design environment” (known as KCIDE). Essentially, engineers will go to their computer where they will do their work on a platform, much like Windows, where their various software packages, laboratory work, and other support software packages (such as *Word*) will all come together and interact with each other to help them with their work. These platforms will capture the work being done by the engineer and make the data available to be used in any manner the engineer chooses (such as preliminary design reports, user manuals, papers, books, proposals, or requests for proposals).

A detailed presentation of all the elements associated with learning how to work in such an environment is beyond the scope of this book. However, a platform to begin the process of training engineers to work in this environment is included in this textbook. *KCIDE for Circuits* was designed to assist the circuits student to learn how to work in a simplified KCIDE environment designed especially for electrical circuits students. The software that is used in the platform includes *PSPice*, *MATLAB*, *Excel*, *Word*, and *PowerPoint*.

In this appendix, we will help you to understand the *KCIDE for Circuits* platform and how to use it. The software can be obtained, free of any charges, from the website <http://KCIDE.FennResearch.org>. More details and examples are also included at the website. In addition, we will also have support services available at the website.

1 How to Work with *KCIDE for Circuits*

The structure of the platform and how it is effectively used follows the problem-solving process used throughout the text. This is essentially a systems approach to problem solving that uses a structured process to capture your work and present it in two different formats. It will be helpful to work through an example to see how to use the platform.

Use the *KCIDE for Circuits* platform to solve Example 3.2.

Example 1

Solution:

Opening the software, we see the screen shown in Fig. 1, where we define a new project. Although we can name the project anyway we



Figure 1

Creating a new project in *KCIDE for Circuits*.

wish to, we name it *KCIDE Example 1 050626* (see Fig. 2). Note that the last six digits are: year/month/day. The reason for this is that if we create different files corresponding to different dates, the files will always appear in chronological order.

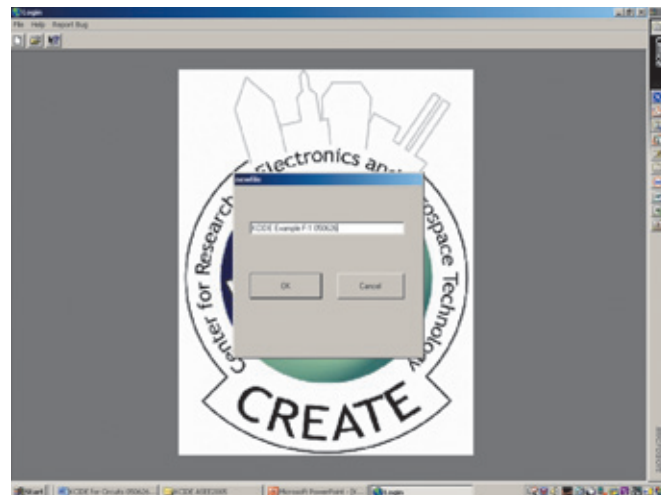


Figure 2

Naming the project.

We now enter the problem statement into the screen shown in Fig. 3. After we have entered the problem statement, we can click on the button to Open *PSpice*. The next screen, Fig. 4, shows what is seen when the Open *PSpice* button is clicked. To open the *PSpice* schematic capture, we need to click on the page 1 icon. In Schematic, we create the circuit representing our problem. This is shown in Fig. 5.

We now need to enter all we know about the problem by entering our problem analysis into the text box and then identifying the number

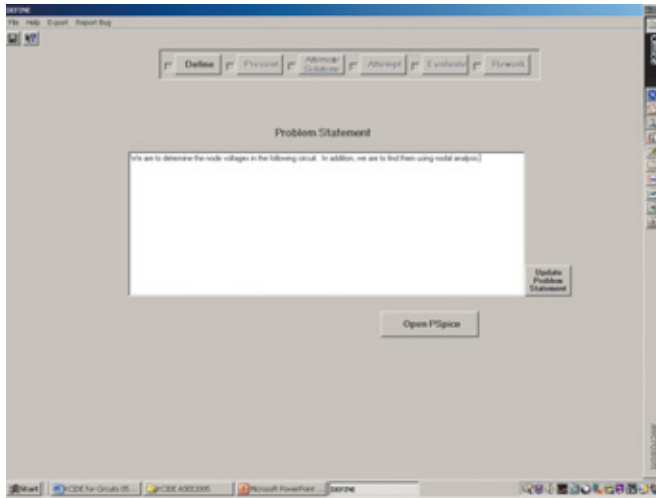


Figure 3
Entering the problem statement.

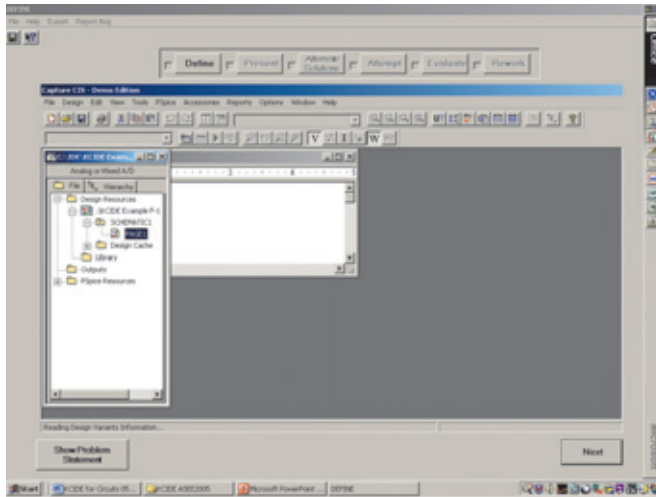


Figure 4
How to open the schematic capture of PSpice.

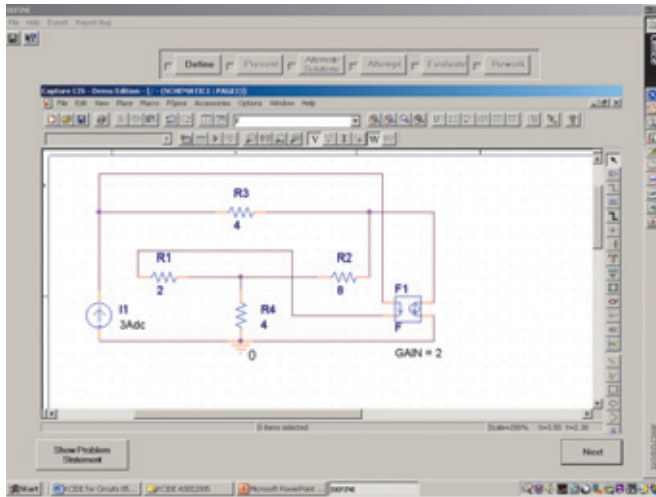


Figure 5
Circuit for Example 1.

of unknown nodes and loops for the circuit (see Fig. 6). We continue this process by going to the next screen and entering the requested information (see Fig. 7).

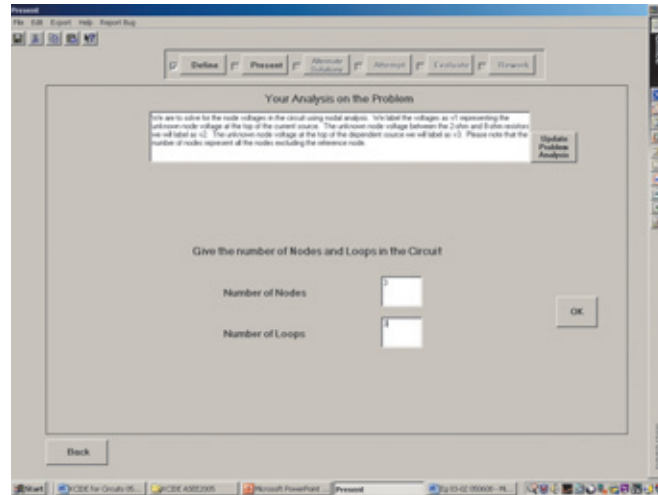


Figure 6

Presenting what we know about the problem, part 1.

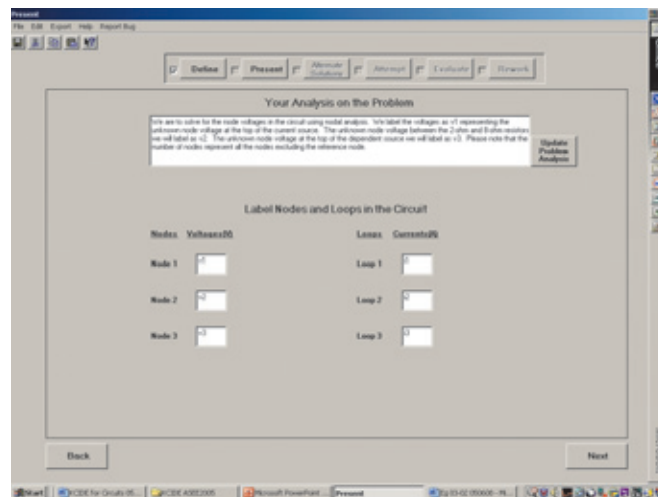


Figure 7

Identifying the unknown node voltages and unknown loop currents, part 2.

We now proceed to selecting the method of solution. We do this by entering the requested information into the screen shown in Fig. 8. Now we can develop the equations that will generate a solution for the problem. Since nodal analysis is required for the solution for the node voltages, all we need to do is to write the nodal equations. Once we have the appropriate equations, as shown in Fig. 9, we can select a solution technique. In this case we chose Excel to solve our simultaneous equations.

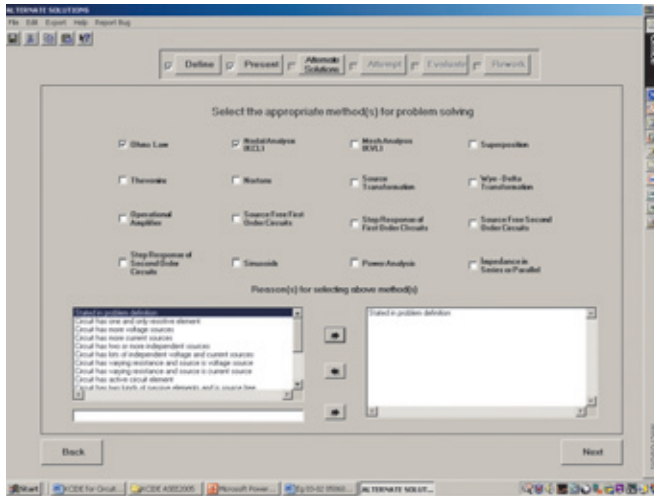


Figure 8
Selecting the method of solving the problem.

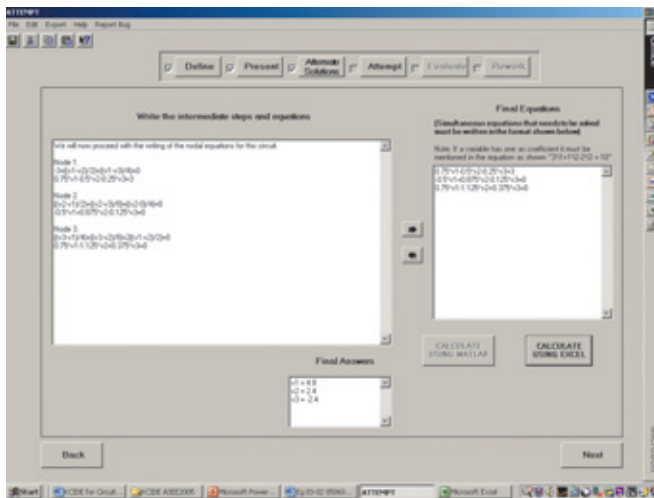


Figure 9
Solving for the unknown node voltages.

Now, when we go to the next screen, the Evaluate portion of the solution, we actually open *PSpice* again. We need to open page 1 to retrieve our original circuit (see Fig. 10). Once we have our original *PSpice* circuit, we need to prepare it for solving for our unknowns. The first step in this task is to go to the *PSpice* button and select New Simulation Profile (see Fig. 11).

We next need to assign a name to the new simulation profile (Fig. 12). Clicking on the Create button produces the screen shown in Fig. 13. For this problem, we select Bias Point for the Analysis type.

Clicking on OK returns the screen to the original condition. Now we go to the *PSpice* button and select Run from the dropdown menu,

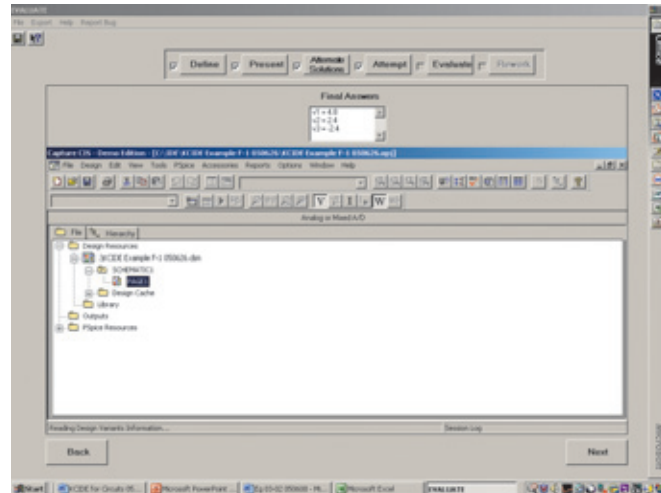


Figure 10
Opening *PSpice* again.

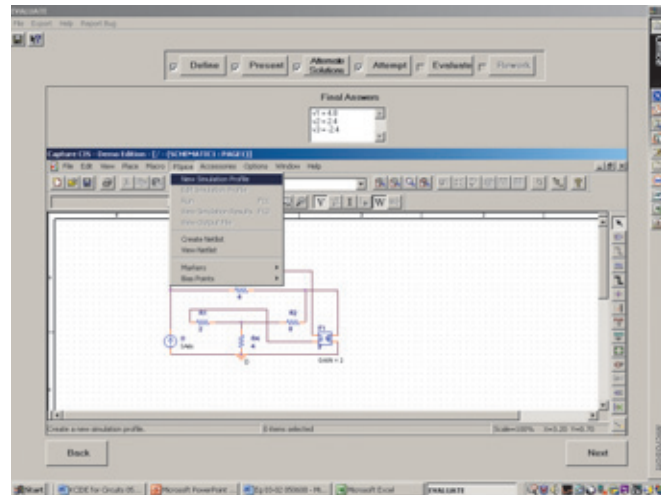


Figure 11
Setting up our circuit for solution by *PSpice*.

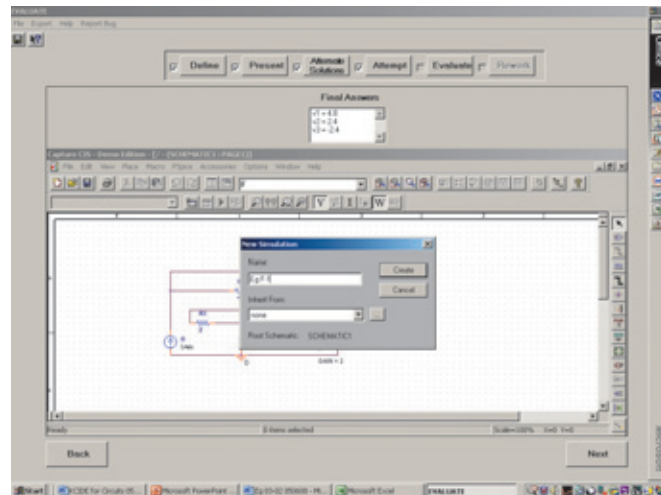


Figure 12
Setting up our circuit for solution by *PSpice*.

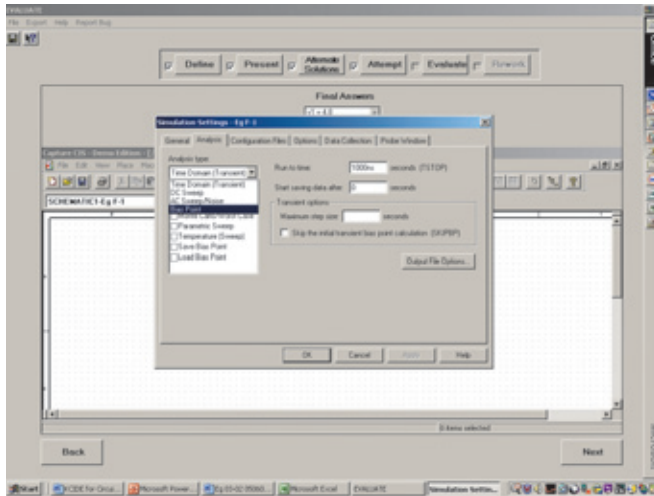


Figure 13
Setting up our circuit for solution by *PSpice*.

Fig. 14. Running *PSpice* produces the screen shown in Fig. 15. We can immediately see that the voltages agree with the solution we obtained by using nodal analysis. Clicking on Next leads to our being asked, as in Fig. 16, if we have any graphics to export. For this problem, we have no graphics.

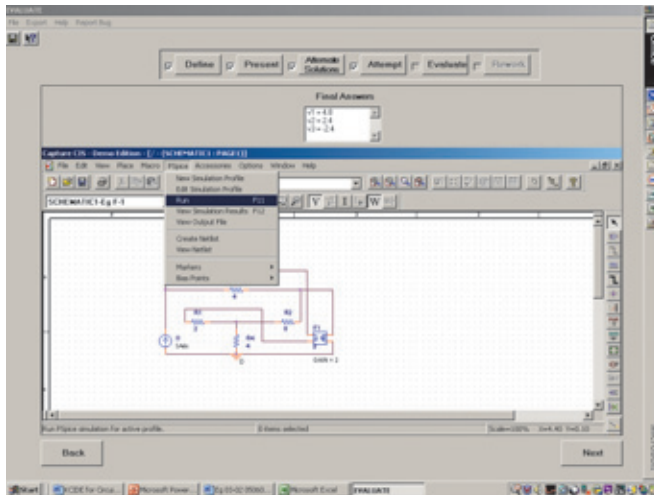


Figure 14
Setting up our circuit for solution by *PSpice*.

We are now approaching the end of the process. We are asked to comment about the solution, Fig. 17. And, we are asked if the answers agree with the *PSpice* solution. The answers do agree and we can proceed to determining what we want to export, Fig. 18. We can generate *Word* and/or *PowerPoint* files, Fig. 19. In this case, we select both but will only show the output of the *Word* file, Fig. 20. Note: This output was modified so that it could be presented on two pages.

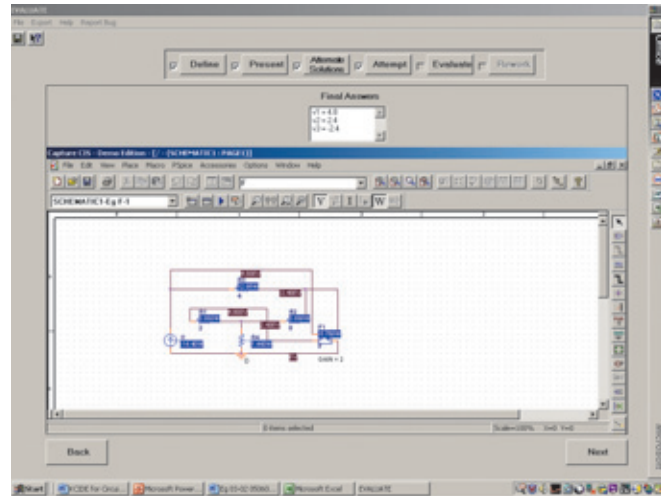


Figure 15
Problem solution using *PSpice*.

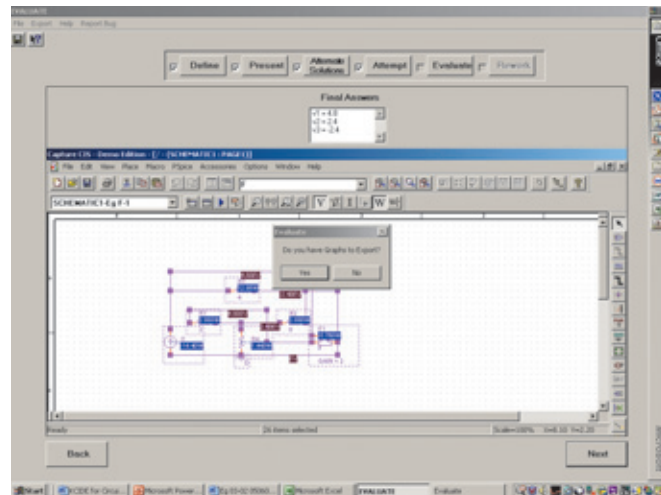


Figure 16
Screen for exporting graphs.

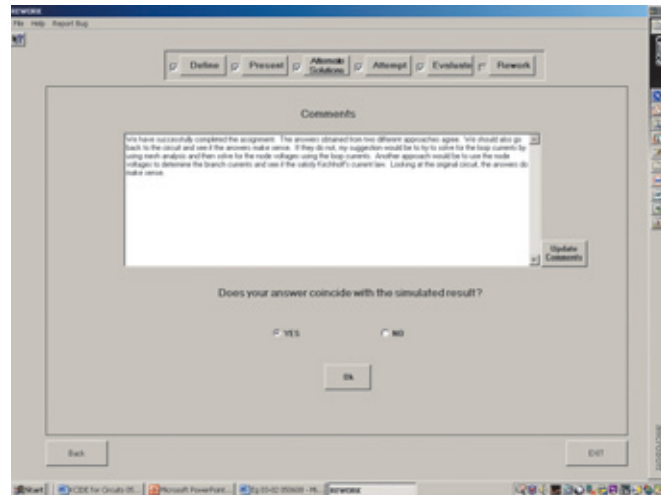


Figure 17
Determining if the problem has been solved correctly.

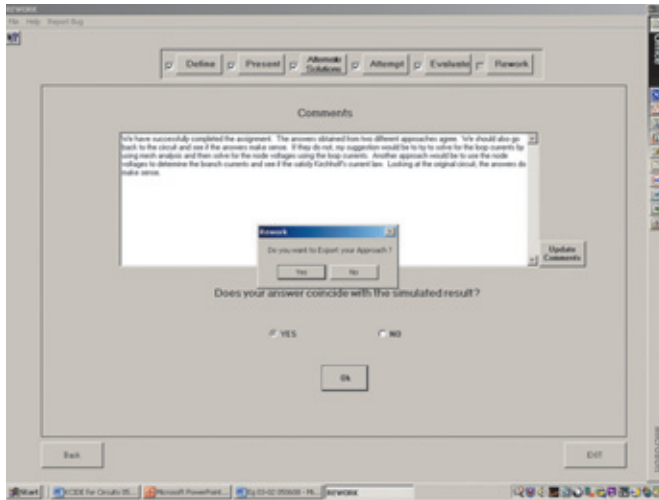


Figure 18
Determining if you want to generate *Word* and/or *PowerPoint* documents.



Figure 19
Generating *Word* and *PowerPoint* files for Example 1.

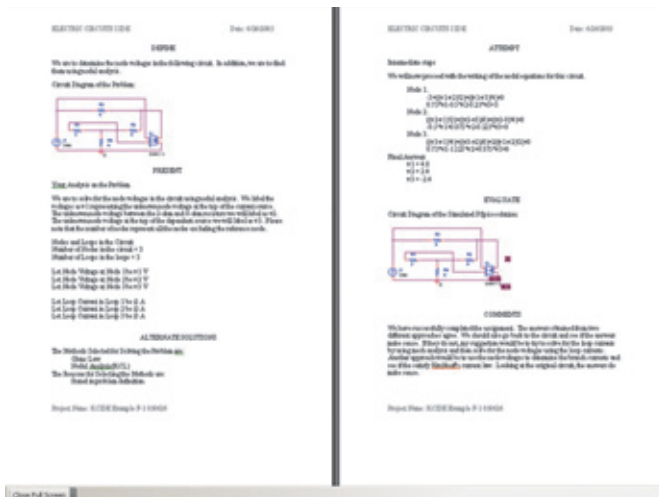


Figure 20
Word file output.

We have now completed a detailed example. We suggest that you first try to do this with the platform and look at your output in both *Word* and *PowerPoint*. To help you to continue to develop your facility with the platform, try working the following practice problem, using mesh analysis. For more examples, please go to the website.

Practice Problem 1

Use the *KCIDE for Circuits* platform to solve Practice Prob. 3.2.

Answer: $v_1 = 80 \text{ V}$, $v_2 = -64 \text{ V}$, and $v_3 = 156 \text{ V}$.
