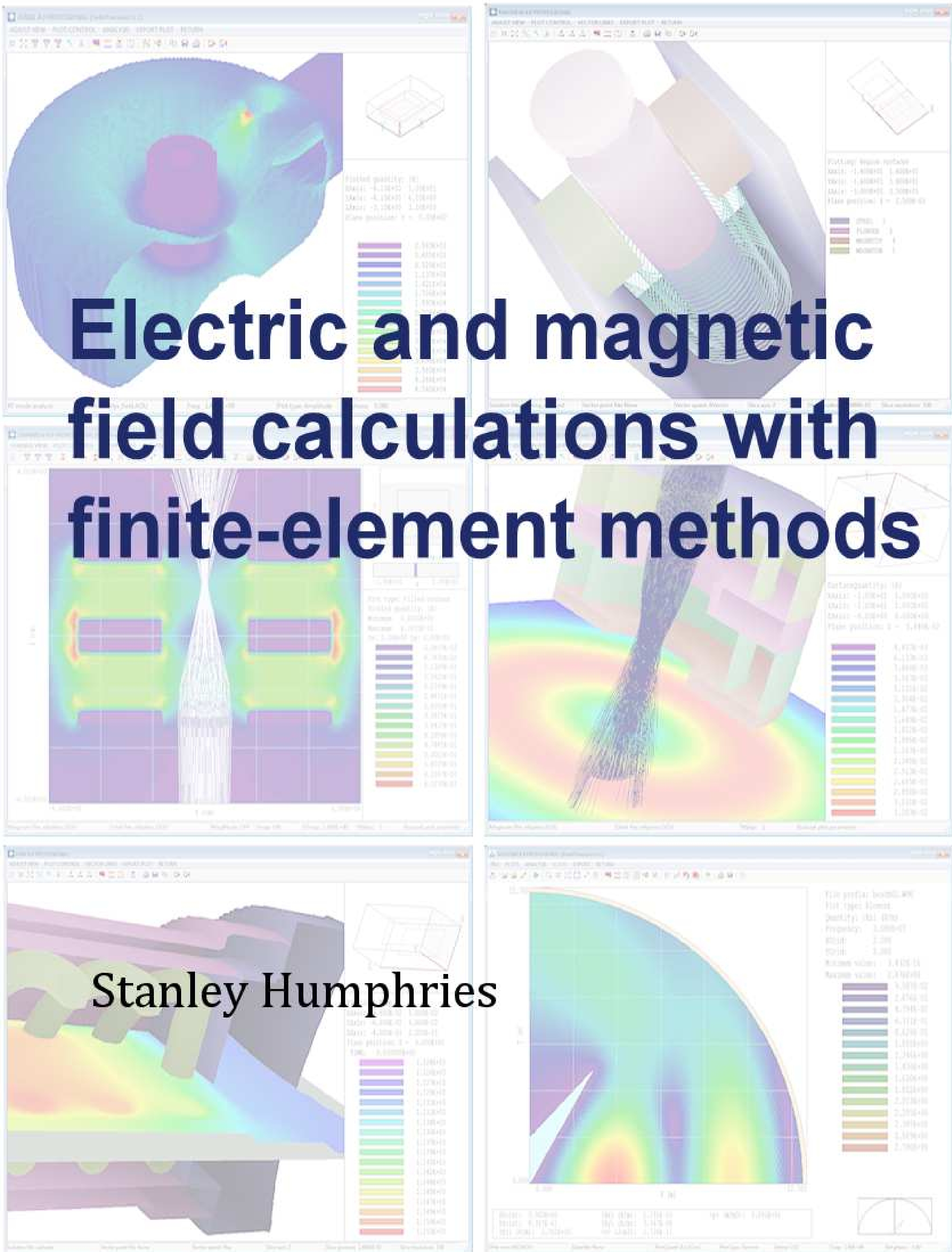


Electric and magnetic field calculations with finite-element methods

Stanley Humphries

Field Precision LLC

Electric and magnetic field calculations with finite-element methods

Stanley Humphries
President, Field Precision LLC
Professor Emeritus, University of New Mexico

Published by
Field Precision LLC
PO Box 13595, Albuquerque, NM 87192 U.S.A.
Telephone: +1-505-220-3975
E mail: techinfo@fieldp.com   Internet: http://www.fieldp.com

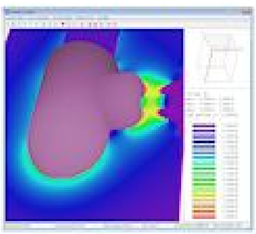
Copyright © 2015 by Field Precision LLC
All rights reserved. This electronic book may be distributed freely if (and only if) it is distributed in its entirety as the file `humphriessfem.pdf`. Sections of the file, text excerpts and figures may not be reproduced, distributed or posted for download on Internet sites without written permission of the publisher.

http://www.fieldp.com

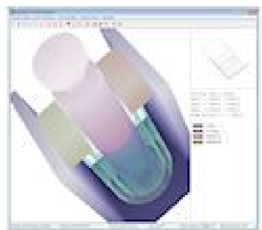
Where to find... | Application areas | Software products | Company information | Educational resources

Field Precision LLC

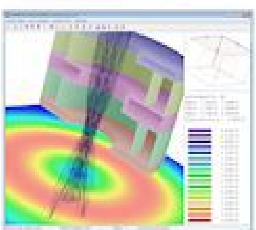
Field Precision creates advanced finite-element simulation software for a broad spectrum of applications: high voltage engineering, magnet design, charged-particle devices, microwave technology, X-ray imaging, and thermal analysis. Unitized 2D and 3D packages are complete and self-contained. Affordable **Basic** packages run on any Windows computer. **Professional** packages with parallel-processing support and unlimited memory access run under 64-bit XP, Vista, 7 and 8. Click on your application area for information on available packages.



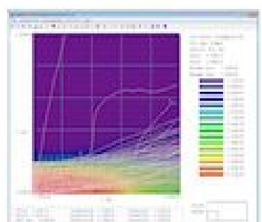
Electrostatics and high voltage engineering
Calculations of field stress on electrodes and dielectrics, electric field lines, capacitance and electrostatic forces.



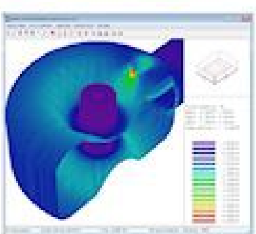
Electromagnets and permanent magnets
Magnetic field distributions and forces. Calculations may include drive coils of any shape, iron with saturation effects and multiple permanent magnets.



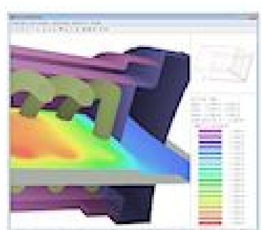
Electron/ion guns and accelerators
Integrated design tools for electron/ion injectors, accelerators and transport systems. Advanced features include beam-generated fields and secondary emission.



X-ray imaging and source design
Monte Carlo simulations of electron, photon and positron interactions in matter from 0.25 keV to 1 GeV. End-to-end design of X-ray sources.



RF and microwave technology
Resonator design, electromagnetic scattering, pulsed-power technology and microwave devices. Time-domain and frequency-domain solutions for 3D structures.



Thermal transport in solids and biological media
Dynamic and steady-state solutions of the thermal transport and bioheat equations. Features include radiation boundaries and perfusion. Option for power input from microwave and X-ray calculations.

Contents

1 Introduction

Field Precision finite-element programs covers a broad spectrum of physics and engineering applications, including charged particle accelerators and X-ray imaging. The core underlying most of our software packages is the calculation of electric and magnetic fields over three-dimensional volumes. To use our electric and magnetic fields software effectively, researchers should have a background in electromagnetism and should be able to make informed decisions about solution strategies. First-time users of finite-element software may feel intimidated by these requirements. My motivation in writing this book is to share my experience in field calculations. I hope to build users knowledge and experience in steps so they can apply finite-element programs confidently. In the end, readers will be able to solve real-world problems with the following programs:

- **EStat** (2D electrostatics)
- **HiPhi** (3D electrostatics)
- **PerMag** (2D magnetostatics)
- **Magnum** (3D magnetostatics)

To begin, its important to recognize the difference between 2D and 3D programs. All finite-element programs solve fields in three-dimensions, but often systems have geometric symmetries that can be utilized to reduce the amount of work. The term 2D applies to the following cases:

- Cylindrical systems with variations in r and z but no variation in θ (azimuth).
- Planar systems with variations in x and y and a long length in z.

Which brings us to the first directive of finite-element calculations: never use a 3D code for a calculation that could be handled by a 2D code. The 3D calculation would increase the complexity and run time with no payback in accuracy.

We need to clarify the meaning of *static* in electrostatics and magnetostatics. The implication is that the fields are constant or vary slowly in time. The criterion of a *slow* variation is that the systems do not emit electromagnetic radiation. Examples of electrostatic applications are power lines, insulator design, paint coating, ink-jet printing and biological sorting. Magnetostatic applications include MRI magnets, particle separation and permanent magnet devices. A following coarse will cover simulations of electromagnetic radiation (*e.g.*, microwave devices).

Secondly, its important to have a clear understanding of the purpose of computer calculations of electric and magnetic fields. Numerical methods should be used when it is not possible to generate accurate results with analytic methods. Numerical solutions are necessary in the following circumstances:

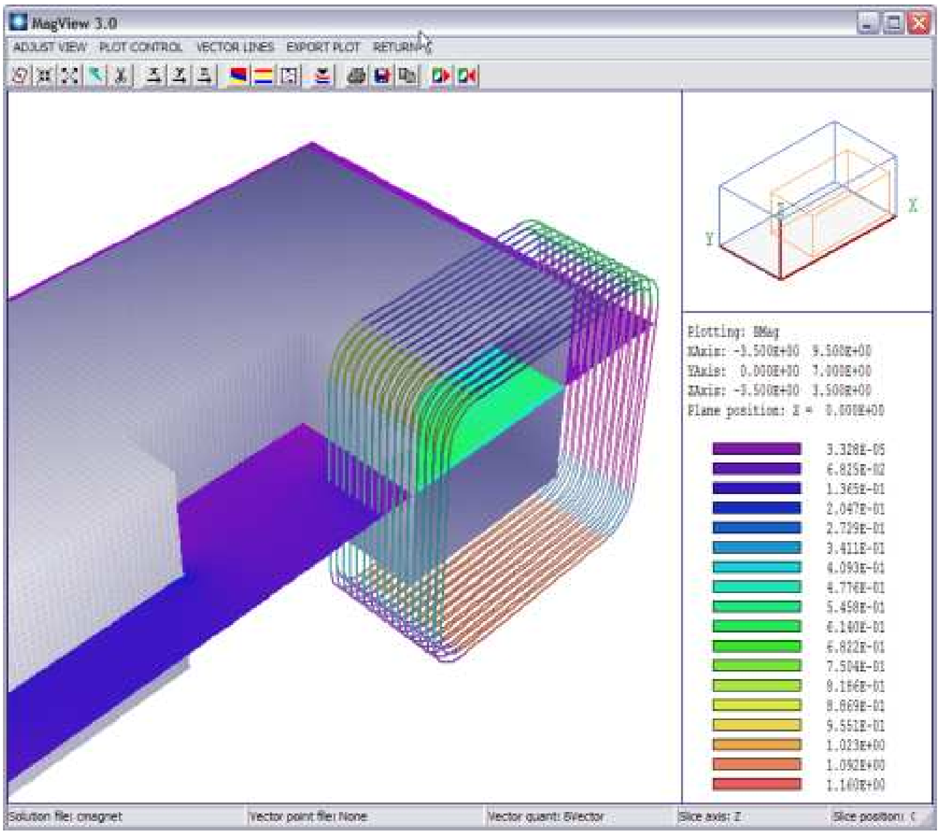


Figure 1: Screenshot of the **MagView** postprocessor for 3D magnetostatics.

- The system has a complex asymmetric geometry.
- The solution volume contains many objects with different material properties.
- Materials have complex properties (*e.g.*, saturation of iron in magnetic circuits)

In an ideal case, a user makes analytic estimates of field values and then applies numerical methods to improve the accuracy. The initial analysis gives an understanding of the physics involved and the anticipated scales of quantities – essential information for effective solution setups. The worst case is when a user treats a program as an omniscient black box. No matter what software manufacturers may claim, using a field program without understanding fields is at best a gamble. Sometimes you may get lucky, but most of the time considerable effort is wasted generating meaningless results.

In summary, I would like to help you become an informed software user. I suggest you start by downloading a free textbook that will help you brush up on electric and magnetic field theory. The book also gives a detailed description of the FEM techniques I will discuss:

S. Humphries, **Finite-element Methods for Electromagnetics** (CRC Press, Boca Raton, 1997) (available for free download at `http://www.fieldp.com/femethods.html`).

The following chapter describes how to download and to set up field-solution software packages.

2 Installing 2D electric-field software

In this chapter, we'll discuss how to install and to test trial or purchased software. As a specific example, consider a trial of the **Electrostatics Toolkit** for two-dimensional electric fields. To request a trial, contact us a `techinfo@fieldp.com`. You will receive an E mail message that includes information like the following:

```
Name: Ernest Lawrence
Organization: LBL
Software: Electrostatics Toolkit
Date: August 20, 2014
Registration code: LAWRENCEER

Thanks for requesting a trial of Field Precision software. To download the
installer, please use this link:

Package: Electrostatics Toolkit Basic
Link: www.dsite.us/download/bin16/ElectrostaticsToolkitSetupBasic.exe
User: bin16
Password: BxHv7821%
```

Click the link to open it in your browser and copy-and-paste the *User* and *Password* information to start the download process. Save the file `ElectrostaticsToolkitSetupBasic.exe` to a convenient location on your hard drive or a USB drive. If you have purchased the software, be sure to keep a copy of the file in case you need to move the software or to install it on a second computer.

When you run the installer, it sets up a directory containing programs, instruction manuals and examples. A file manager is useful to check out the new materials. Because number-crunching finite-element programs produce a lot of data, a good file manager is a critical tool for your future work. Figure 2 shows a screenshot of **FP File Organizer**, a free utility included with our software.

If you check the root directory of the hard drive, youll see that the installer has created the directory `c:\fieldp_basic` (or `c:\fieldp_pro` if you purchased the professional version). Figure 2 shows the directory contents (left-hand side). The file `readme_basic.html` is a useful summary of instructions. The `tricomp` sub-directory (right-hand side) contains the programs, documents and examples of the 2D package:

- `dielectric_constants.html`. Relative dielectric constants for a variety of materials, useful for setting up electrostatic solutions.
- `estat.exe`. The main solution program that combines information on the computational mesh and material properties to find electrostatic potential values at nodes. The program also creates graphs and plots of the solution (`i.e.`, post-processing).
- `estat.pdf`. The **EStat** instruction manual.

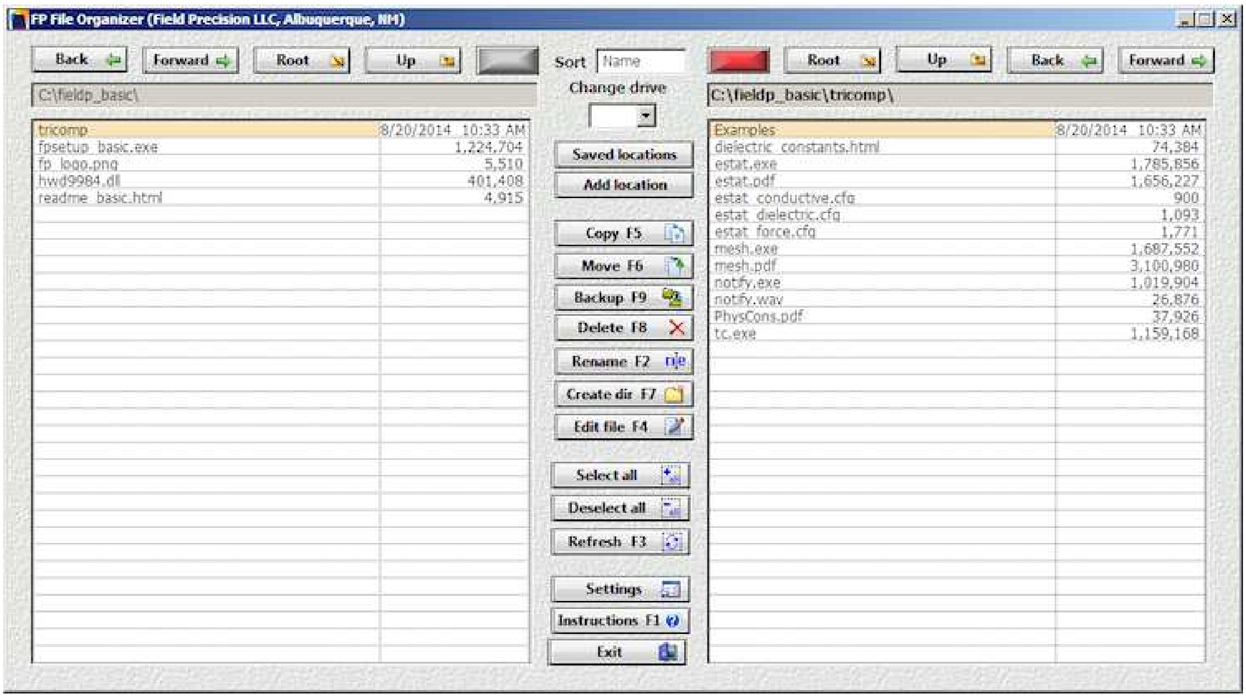


Figure 2: **FP File Organizer**, `fp_basic` directory.

- `estat_conductive.cfg`, `estat_dielectric.cfg` and `estat_force.cfg`. Configuration files for the **EStat** post-processor for different types of electrostatic solutions.
- `mesh.exe`. The automatic mesh generator to create 2D conformal, triangular meshes.
- `mesh.pdf`. The instruction manual for **Mesh**.
- `notify.exe` and `notify.wav`. Utility programs to signal the completion of an automatic batch run.
- `PhysCons.pdf`. A reference sheet of physical constants.
- `tc.exe`. An automatic controller for programs and resources of the 2D packages that we will discuss in detail later.

The `examples` sub-directory contains directories of prepared examples for both the **Mesh** and **EStat** programs (Figure 3). These examples can help you get off to a quick start. Well talk about running them later. For now, well concentrate on getting all components running.

The *Basic* versions of our programs use Internet license management. The installer creates a **TriComp** icon on your desktop (Fig. 4). Click on it to run `tc.exe`, the **TriComp** program launcher. Well discuss the functions of the buttons latter. For now, click the *Activation* button to launch `FPSetup_Basic.exe` (Fig. 5, left-hand side). Click the *License* button, read the license and then close the text window. Click the *Setup* button to open the activation dialog (Fig. 5, right-hand side). Enter the registration code that we sent and pick any user name. Check that you accept the terms of the license and click the *Process* button to receive a unique *Machine number* for your computer. This number is copied to the clipboard.

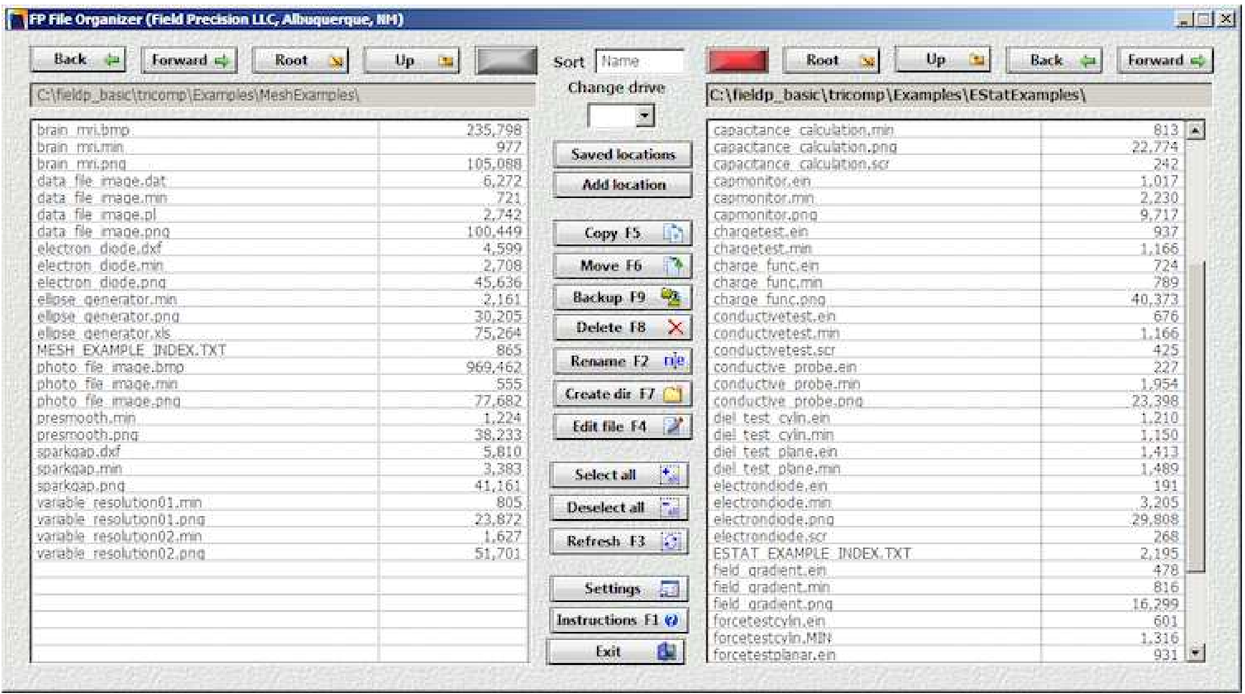


Figure 3: **FP File Organizer** display of example directories

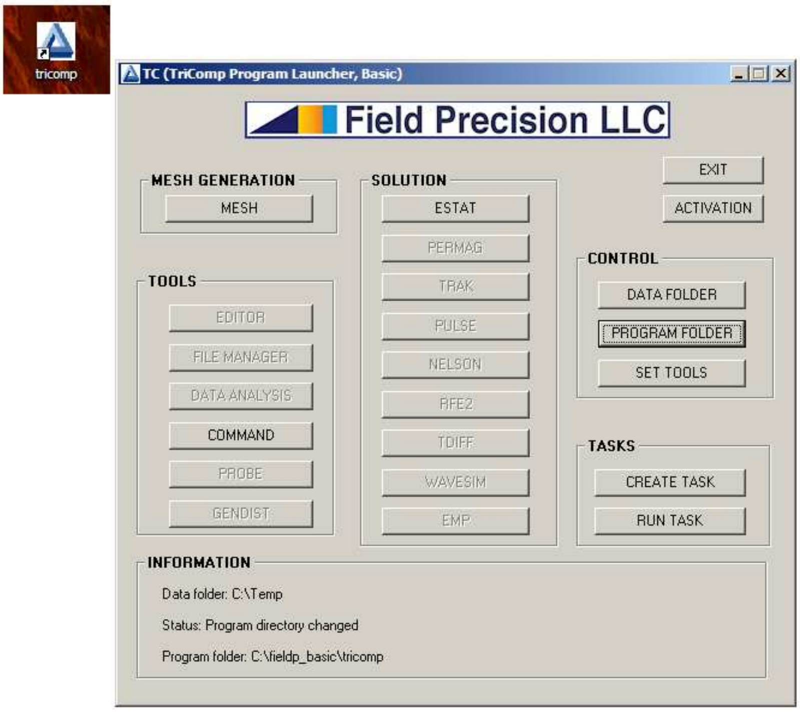


Figure 4: **TriComp** icon and **TC** program launcher.

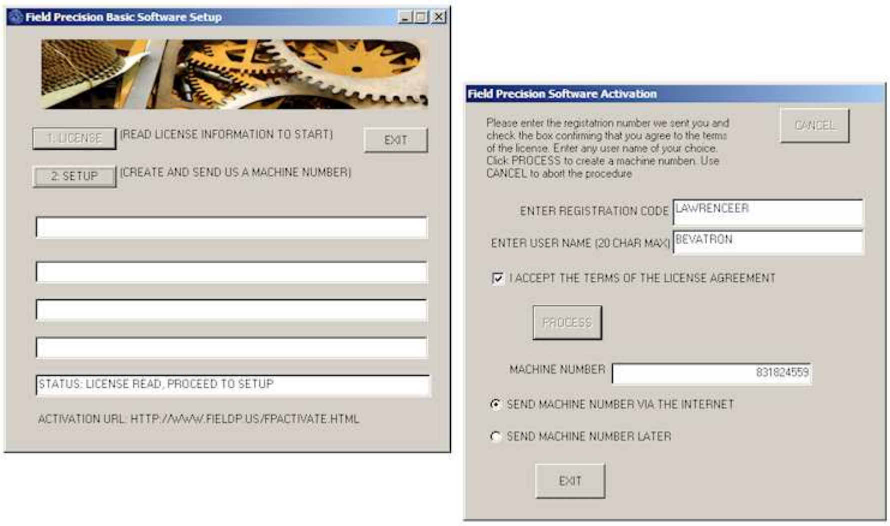


Figure 5: **FPSetup_Basic** program and *Setup* dialog

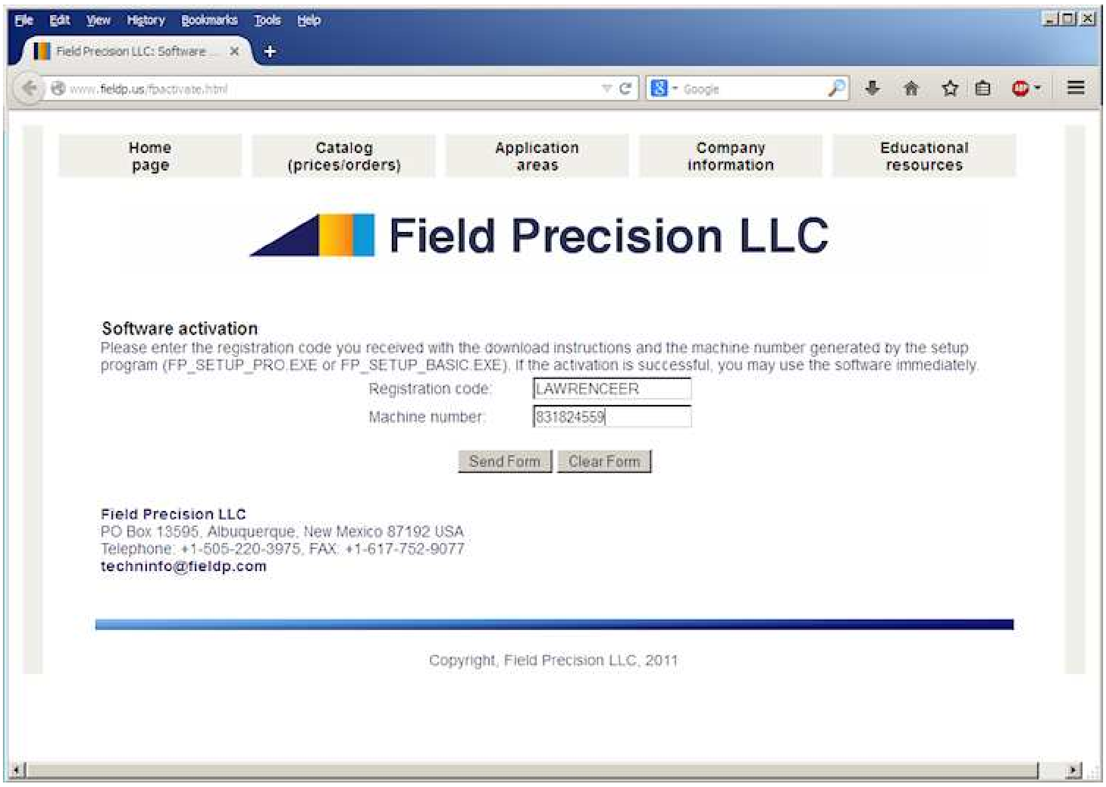


Figure 6: Internet activation page.

Check the *Send machine number via Internet* option and click *Exit*. Your default browser opens on our activation page (Fig. 6). Paste in the machine number, type your registration code and press *Send form*. The software is now ready to use. To check the setup, return to the program launcher (Fig. 4) and click the *Mesh* button. The program should open with no error message. There are two problems you may encounter.

Problem: the **Mesh** button is not active in `tc.exe`.
Solution: Click the *Program folder* button and navigate to `c:\fieldp_basic\tricomp`.

Problem: **Mesh** reports an Internet activation problem
Solution: Either your computer is not connected to the Internet or you are connected through a proxy server. If your computer is on a company network with a proxy server, it is necessary to set an environmental variable to use the software. See this blog post for instructions: `http://fieldp.com/myblog/2011/identifying-a-proxy-server/`.

In the next chapter, well solve and analyze a real-world problem using one of the prepared **EStat** examples.

3 First 2D electrostatic solution

In this chapter, well run through the steps of a solution and analysis of a 2D electrostatic problem without going into detail. The goal is a quick demo of the capabilities of **EStat**. Subsequent articles will cover details of program techniques.

A notable feature of our programs is dual input - there are two options for supplying geometric and material data for solutions:

- **Interactive**: the standard method for modern finite-element programs where you fill in items in dialogs. This option is useful for new users and for a quick setup of a new system.
- **Text**: the classic method using input scripts. This option allows experienced users to make changes to setups easily and facilitates automatic program operation under the control of external programs or batch files. Scripts also provide a permanent archive of setups.

In this demo calculation, we will check out prepared input scripts before running them using the built-in text editors of **Mesh** and **EStat**. For serious work, its essential to have a good text editor. This blog article describes how to obtain the **ConText** editor and how to add syntax color coding for our programs:

`http://fieldp.com/myblog/2014/using-the-context-text-editor-update/`

Before starting, we'll need to make some provisions for data organization. A little effort in the beginning circumvents headaches later. Run **FP File Organizer** or your own file manager. Navigate to a location where you would like to create a general directory (folder) for your finite-element calculations and create the directory `Simulations` (I will use `C:\Simulations`). Create the sub-directories `Electrostatics` and `Electrostatics\Practice`. In the right-hand window, navigate to `C:\fieldp_basic\tricomp\Examples\EStatExamples`. We will copy an example for our work. Highlight the files with name `ElectronDiode` and copy them to the `Practice` directory. Figure 7 shows the resulting setup in **FP File Organizer**.

Click on the desktop shortcut to run the **TriComp** program launcher `tc.exe` (Fig. 8). The **Mesh** and **EStat** buttons should be active as shown. If not, click the *Program folder* button and navigate to `C:\fieldp_basic\tricomp`. Click the *Data folder* button and navigate to `c:\Simulations\Electrostatic\Practice`. Subsequently, all input/output operations will target this folder.

There are three steps in a finite-element solution:

- Define the geometry of the solution space and divide objects into small volumes (*elements*). The process is called *mesh generation*.
- Create and solve a large set of linear equations to approximate the governing partial differential equation (such as the Poisson equation for electrostatics). The goal is to determine the electrostatic potential on points of the mesh (*nodes*).
- Analyze the results - use the potential values to find physical quantities of interest (*e.g.*, electric field, field energy, capacitance,....).

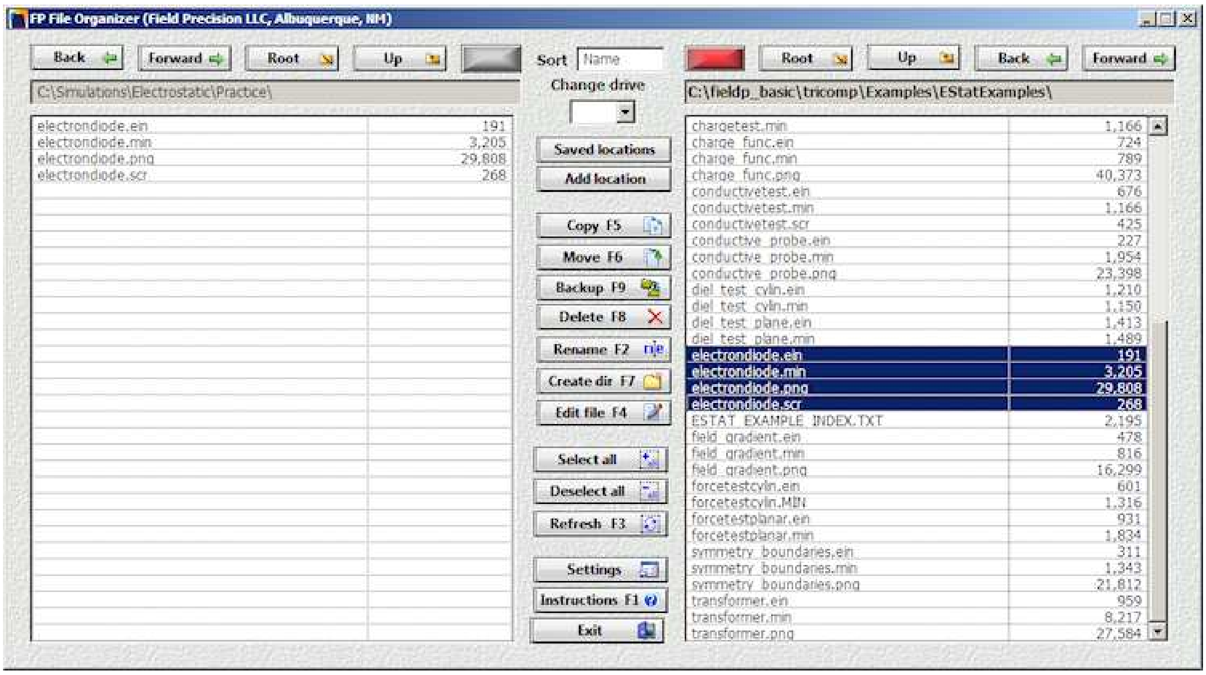


Figure 7: Set up a data directory and copy the examples.

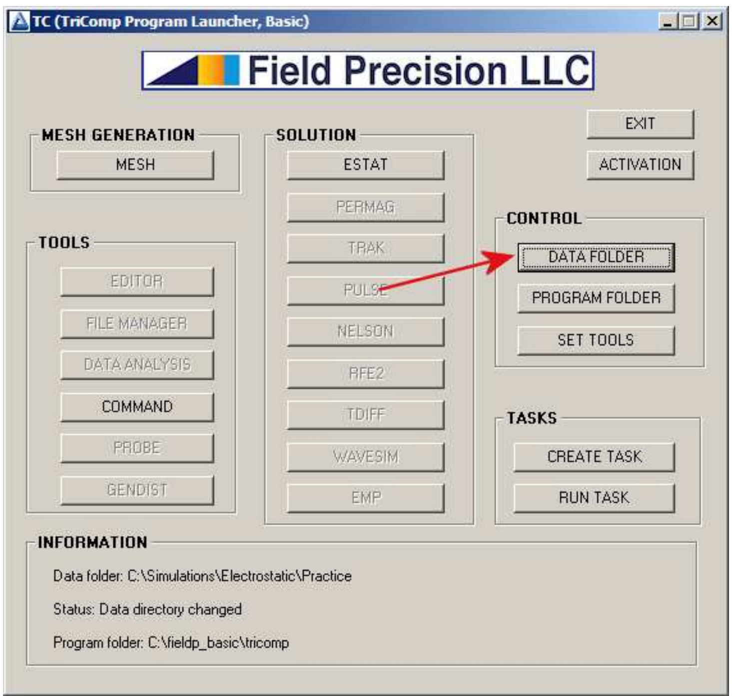


Figure 8: The **TriComp** program launcher.

The first function is performed by the **Mesh** program (`mesh.exe`) and the second and third functions are performed by the **EStat** program (`estat.exe`). Meshing is performed by a separate program because the same mesh may be used for different types of solutions. Output from **Mesh** is compatible with **PerMag** (magnetic fields), **TDiff** (thermal transport) and other solution programs of the **TriComp** series. In other words, what we learn about **Mesh** for electrostatics is also useful for the magnetic solutions we will discuss later.

Click the **Mesh** button to open the program. Initially, the screen is blank. Choose *File/Edit file* from the menu at the top. The selection dialog shows the four files in the data folder. Pick `ElectronDiode.MIN`, where `MIN` designates **M**esh **IN**put. The internal editor shows the file contents (Fig. 9), a set of organized numbers. For now, note that there are *Region* sections that represent the different physical objects in the solution space. Each region section contains a set of line or arc vectors that gives the region shape. The numbers are the coordinates of the vectors. There are two ways to create or to modify the content of `MIN` files:

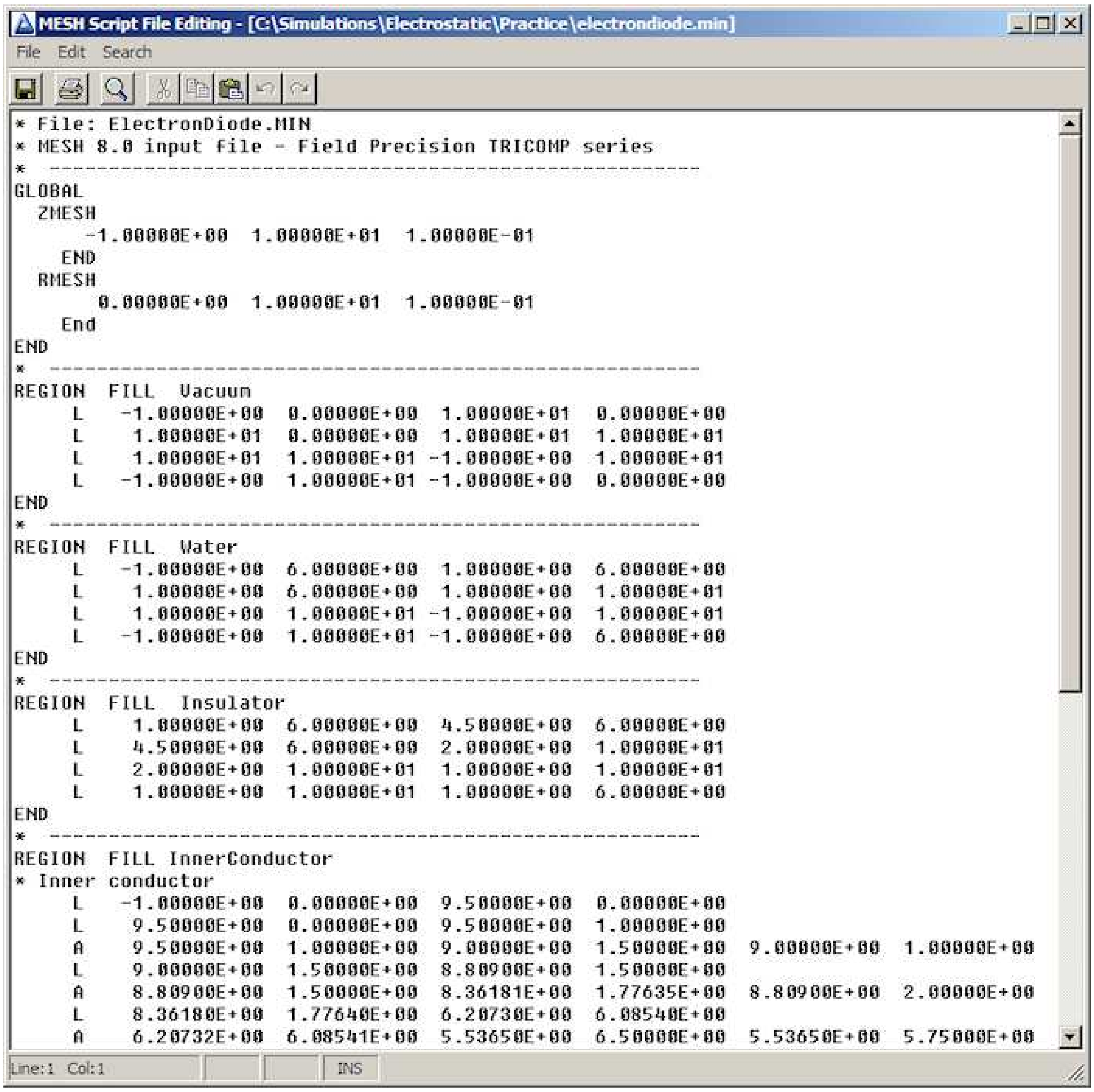
- Use the **Mesh** *Drawing Editor*, an interactive 2D CAD program.
- Use a text editor to change numbers directly.

We will discuss both options in following articles. For now, exit the editor with no changes.

Choose *File/Load script(MIN)* in the menu or use the *Open-file* tool on the left hand side of the toolbar to load the contents of the file `ElectronDiode.MIN`. Then pick *Process* or click the tool with the green square. In response, **Mesh** analyzes the desired element sizes and the region vectors to create a set of small elements. To view the result, choose *Plot/Repair* from the menu or click the *Plot/repair* tool to show the display of Fig. 10. The solution volume has been divided into the regions listed in the script to represent physical objects (electrodes and dielectrics). The cross section has been divided into triangular areas. Note that the calculation represents a cylindrical system, a figure of revolution about the bottom boundary ($r = 0.0$). When first viewing a z-r plot, many users ask wheres the bottom? The answer is that there is no bottom. Negative values of the radius r are undefined. On the other hand, a plot in a Cartesian slice plane like $y = 0.0$ would have both upper and lower components. In cylindrical solutions, elements are tori with triangular cross-sections that extend over the full range of azimuth ($\theta = 0^o$ to 360^o).

Lets take a closer look at the solution. Choose *View/Zoom window* and specify the corners of a box by moving and left-clicking the mouse. Figure 11 shows a magnified view for a better look at the element cross-sections. The inset at lower-right shows the view limits. Note that the triangles were flexed for a good match to region boundaries. The fitting allows high-accuracy calculations of surface fields. A mesh with element shapes customized to the geometry is called a *conformal* mesh. To conclude work with **Mesh**, return to the main menu and choose *File/Save mesh (MOU)*. Refresh the display in **FP File Organizer** (press F3). Two new files have been added to the folder:

- `ElectronDiode.MLS`: a text listing file of diagnostic information that may be useful if there is a problem.
- `ElectronDiode.MOU`: the main output file specifying element identity (region association) and coordinates of nodes (boundaries between elements). This file may be ported to any of the **TriComp** solution programs. The file is in text format, so you can inspect it with an editor.




```
* File: ElectronDiode.MIN
* MESH 8.0 input file - Field Precision TRICOMP series
* ----------------------------------------------------------
GLOBAL
  ZMESH
      -1.00000E+00  1.00000E+01  1.00000E-01
    END
  RMESH
       0.00000E+00  1.00000E+01  1.00000E-01
    End
END
* ----------------------------------------------------------
REGION  FILL  Vacuum
     L    -1.00000E+00  0.00000E+00  1.00000E+01  0.00000E+00
     L     1.00000E+01  0.00000E+00  1.00000E+01  1.00000E+01
     L     1.00000E+01  1.00000E+01 -1.00000E+00  1.00000E+01
     L    -1.00000E+00  1.00000E+01 -1.00000E+00  0.00000E+00
END
* ----------------------------------------------------------
REGION  FILL  Water
     L    -1.00000E+00  6.00000E+00  1.00000E+00  6.00000E+00
     L     1.00000E+00  6.00000E+00  1.00000E+00  1.00000E+01
     L     1.00000E+00  1.00000E+01 -1.00000E+00  1.00000E+01
     L    -1.00000E+00  1.00000E+01 -1.00000E+00  6.00000E+00
END
* ----------------------------------------------------------
REGION  FILL  Insulator
     L     1.00000E+00  6.00000E+00  4.50000E+00  6.00000E+00
     L     4.50000E+00  6.00000E+00  2.00000E+00  1.00000E+01
     L     2.00000E+00  1.00000E+01  1.00000E+00  1.00000E+01
     L     1.00000E+00  1.00000E+01  1.00000E+00  6.00000E+00
END
* ----------------------------------------------------------
REGION  FILL InnerConductor
* Inner conductor
     L    -1.00000E+00  0.00000E+00  9.50000E+00  0.00000E+00
     L     9.50000E+00  0.00000E+00  9.50000E+00  1.00000E+00
     A     9.50000E+00  1.00000E+00  9.00000E+00  1.50000E+00  9.00000E+00  1.00000E+00
     L     9.00000E+00  1.50000E+00  8.80900E+00  1.50000E+00
     A     8.80900E+00  1.50000E+00  8.36181E+00  1.77635E+00  8.80900E+00  2.00000E+00
     L     8.36180E+00  1.77640E+00  6.20730E+00  6.08540E+00
     A     6.20732E+00  6.08541E+00  5.53650E+00  6.50000E+00  5.53650E+00  5.75000E+00
```

Figure 9: Internal **Mesh** editor display.

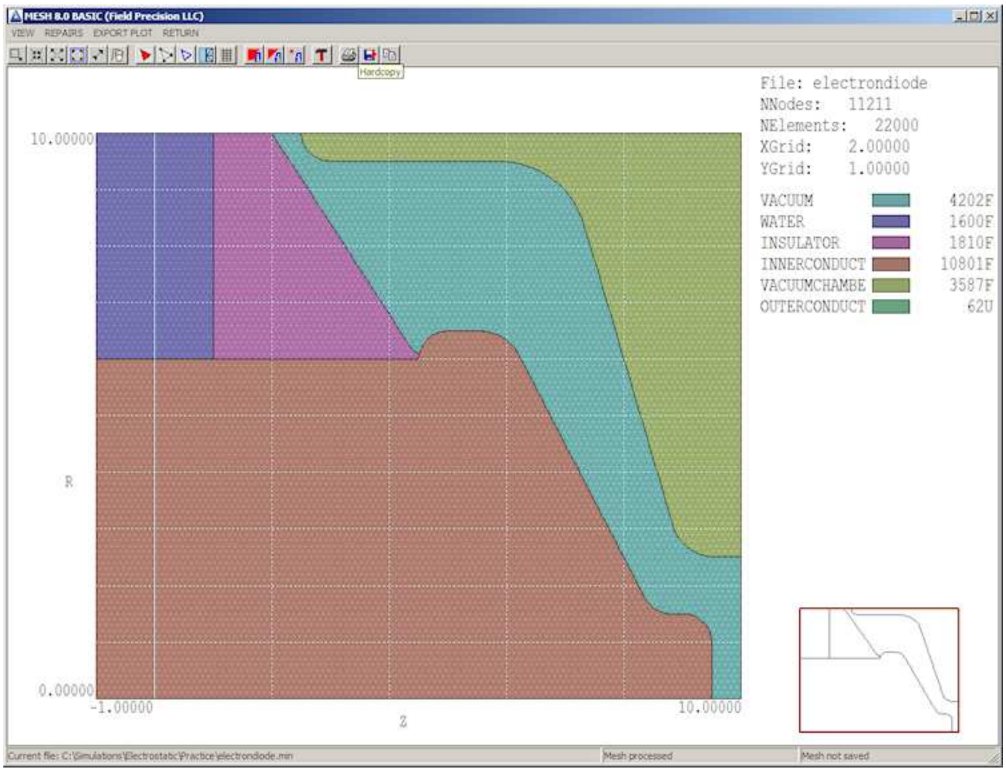


Figure 10: View of the full mesh.

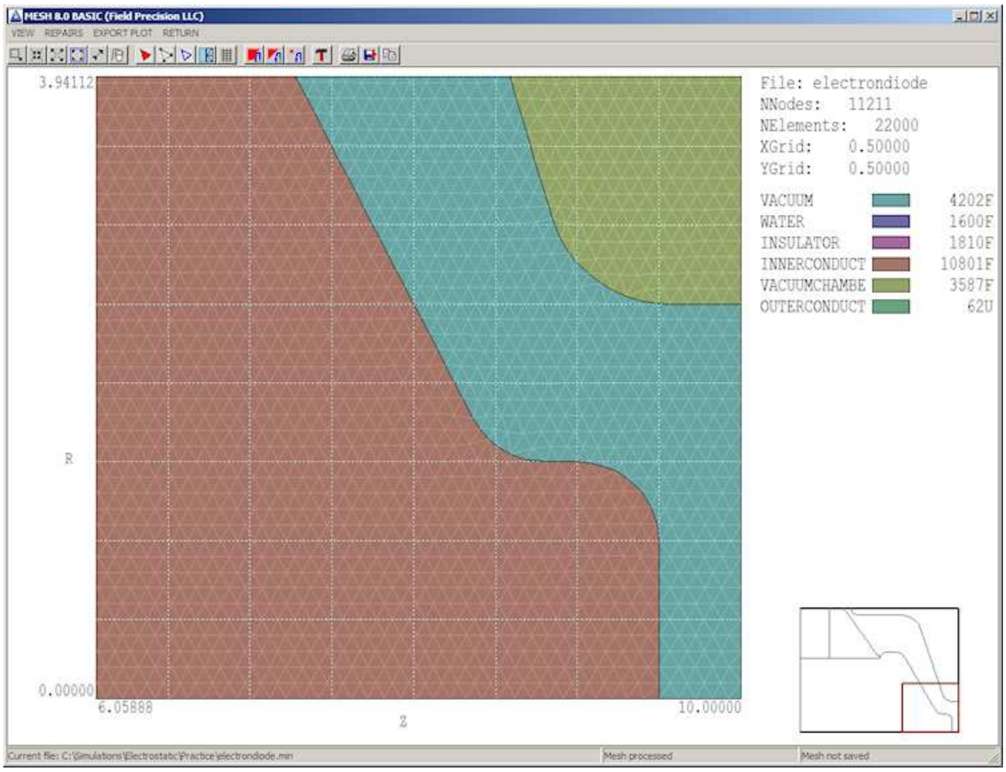


Figure 11: Detailed view of a mesh section.

Next, run **EStat** from the program launcher. Choose *File/Edit script (EIN)* and pick the file `ElectronDiode.EIN`. The editor shows the content

```
* File ElectronDiode.EIN
DUnit = 39.37
Geometry = Cylin
Epsi(1) = 1.0
Epsi(2) = 81.0
Epsi(3) = 2.7
Potential(4) = -500.0E3
Potential(5) = 0.0E3
Potential(6) = 0.0E3
ENDFILE
```

The two general commands at the top specify that the dimensions from the **Mesh** file should be interpreted in inches and that the system has cylindrical symmetry. The other commands set the physical properties of regions of the solution volume. The first three regions are dielectrics (vacuum, purified water and cast epoxy) and the other regions are fixed-potential electrodes. Again, there are two options for creating **EStat** input scripts - supplying values in an interactive dialog or direct input with a text editor.

The tools labeled *1, 2* and *3* invoke the three main functions of EStat:

- Create the input script.
- Perform the finite-element solution.
- Analyze the results.

Step *1* has already been performed, so click the tool marked *2* and pick `ElectronDiode.EIN`. **EStat** reads the geometry data in `ElectronDiode.MOU`, generates the set of finite-element coupled linear equations and solves them, all within a second. **FP File Organizer** shows that two new files have been created: `ElectronDiode.ELS` (a diagnostic text listing file) and `ElectronDiode.EOU` (the main solution file containing potential values at nodes). Click the *3* tool in **EStat** and pick `ElectronDiode.EOU`. The program shifts to the *Analysis* menu and displays the default equipotential-line plot of Fig. 12. This type of plot is useful for experienced users because it shows complete information. Section 7.5 of the companion text **Finite-element Methods for Electromagnetics** describes how to interpret equipotential plots.

There are many capabilities of the *Analysis* menu that you can explore. Lets check out two of them. A plot of the magnitude of the electric field $|\mathbf{E}|$ is useful to pinpoint areas where breakdowns may occur in high-voltage systems. To create the plot, press *Plots/Plot settings/Plot type* in the menu and choose *Element*. Then press *Plots/Plot settings/Plot quantity* and pick $|\mathbf{E}|$. The resulting plot (Fig. 13) shows that the peak field in the electron emission region is about 442 kV/cm and that the maximum field on the insulator vacuum surface is about 40 kV/cm. A second activity is to find the capacitance of the system downstream from the insulator. We can determine this quantity from the volume integral of electrostatic field energy density in the vacuum region. Press *Analysis/Volume integral*. **EStat** offers to open a data record file with the default name `ElectronDiode.DAT`. Global information will appear on the screen, but we need to check the file for a detailed breakdown by regions.

Thank You for previewing this eBook

You can read the full version of this eBook in different formats:

- HTML (Free /Available to everyone)
- PDF / TXT (Available to V.I.P. members. Free Standard members can access up to 5 PDF/TXT eBooks per month each month)
- Epub & Mobipocket (Exclusive to V.I.P. members)

To download this full book, simply select the format you desire below