Step by step setup for the FEMM DC simulation:

1. Create a project: File > New > Current Flow Problem. Then set up with „Problem" on the menu bar. Here we set the plane copper thickness (depth), length unit (mm), frequency (1Hz), minimum mesh angle (5deg) and other parameters.
2. File > Import DXF. Select DXF file and wait. Before DXF import we should simplify the geometry in a DXF editor like the freeware A9CAD. Even then the import process might take minutes or an hour.
3. Simplify the geometry by deleting unnecessary points and lines. Reduce arcs to polygon lines, by selecting all of them, then press space, then set the „Max Segment" filed to 45 degrees. Make sure that all poly-lines and plane edges are closed and lines and arcs separate the different areas.
4. Set up materials, copper and air using the Properties > Materials > Add. Sigma is 59600000 for copper, and 0 for the air, assuming the units were millimeters.
5. Set up the boundary conditions, source and load at Properties > Conductors. For the source (DC/DC converter) we create one conductor with fixed voltage option and VCC voltage value. For the load (BGA chip) we use the „Total Current" option with the load current with negative sign.
6. Assign the materials by „Assign block Labels" button. Place one in the copper area, and one outside of the power plane shape („Air". Also select „Default" option to cover all the plane voids with default Air material). Right click select, then space-key to select one of the pre-defined materials.
7. Assign the conductors. Select multiple edges (in line or arc mode) using right click, then press space for the properties dialog. Select multiple via (all power pin vias) edges under the BGA chip for the load conductor, and multiple vias at the DC/DC converter for the source conductor. Which via is a power via – we have to check them one by one in the PCB layout viewer.
8. Mesh > Create Mesh. If it fails, then back to geometry simplification. The mesh should not be larger that 100k nodes. We can also try to reduce the min angle.
9. Analysis > Analyze. If this is not complete in an hour, then again back to geometry simplification. If successful then we can open the result widow with Analysis > View Results.
10. Measure voltage drop: In line edit mode click on one point of the DC/DC converter, and then one one point of the BGA power vias. Then from the menu click: Integrate > Voltage Drop. This gives a complex number, but normally with a 99.99% real part, so we can go with the real part only, close enough. To get the DCR, we can calculate: DCR = V_Source / I_Load. If we use multiple layers to deliver the power, then we can simulate on all layers separately, calculate DCR separately, then calculate the effective total DCR as parallel circuit effective resistance 1/Re = 1/R1 + 1/R2 +... Then the total voltage drop in the system will be V_drop = Re * I_Load.
11. It is useful to see which parts of the plane carry more current than others, so we can optimize the plane shape to reduce DCR by eliminating hot-spots. For this select View > Density Plot > „|J|" in the menu. If the colors are not telling the story very well, then we can change the red and blue thresholds by experimenting with the numbers in the View > Density Plot > „Upper Bound", or „Lower Bound" fields.

Step by step setup for the Sonnet Lite AC simulation:

1. Create a new design: Project > New Geometry.
2. Set up the design: In separate windows accessed from the Circuit menu we set up the stackup, materials (dielectric layers and metal types) and other simulation parameters. We also define a "circuit box" that is needed for simulation (at Circuit > Box menu), and should be sufficiently large, for example 50 thicker than the PCB, and have cell size around 1mm, and 100x100 cells total size. We should use the bottom plane of the box as ideal ground plane, and define one metal layer (called "0") for the power plane. We select FR4 material in between the power plane and the box bottom side, and air between the power plane and the box top side. If we have multiple power/GND plane pairs then we have to generate separate S-parameter models for each.
3. Manually draw a polygon on the power plane layer (called "0") by Tools > Add Metalization > Draw Polygon from the menu. Make sure that the larger dimensions are accurate; measure them in the PCB layout design file.

4. Draw two small polygons inside the plane, one for the voltage regulator and one for the load.
5. Place simulation ports on the edges of the small polygons by Tools > Add Port from the menu. Set their type to “Auto-ground” by double-clicking on them.
6. Simulation setup: Analysis > Setup, and Analysis > Output File > S,Y,Z Parameters.
7. Run simulation from Project > Analyze. It saves it to a Touchstone file. We can display the result with Project > View Response > New Graph.
8. Another useful feature is the Analysis > Estimate Box Resonances. Also try the “View Currents” feature that displays current distribution on selectable frequencies.