Power Up Sequence:
If you read the User Manual it gives the correct way to power the devices up. To reiterate, first plug in two coincells into the coincell holders, then plug in two LiPo 7.4V batteries to the switch in the bottom of the case, one on pin 1 and one on pin 3. Connect the GND ends to the GND header, then enable the switch that connects to the positive terminal. The system will then power on. If you are adding an LCD, connect the LCD before power on as well as the GPS unit, in no particular order. Before power is provided, ensure the BeagleBone barrelcable is SECURELY inserted into it's male socket.
The unit will not enter full power mode unless >7.4V is applied to the positive LiPo battery terminals. The unit will not boot unless there are backup batteries in place in the coincell holders. Once the entire system is running off of the LiPo battery supplies the backup batteries can be removed and the system will not shut down, however the system will not be able to reboot if required.

Note on Eagle PCB and Fabhouse:
This is not the industry standard for PCB layout, however it is widely used and supported at Fabrication houses. The vendor we used to print the PCB's is community based- "Osh Park". You must register an account with them to order, but after that it is usually about a two week turnaround from order to reception of PCB's and you get three copies.
Eagle PCB is tedious unless you have a large library of components, which Todd Sukolsky does. To get that library, email him at sukolsky@bu.edu. Otherwise, follow the onscreen instructions on how to make new parts and add to a library.

Folder Explanation:
CAD-CAM Files) Contains two folders. The one with "Casing" has the Solidworks files that outline the casing and how to make it. Solidworks is a multi-thousand dollar program and BU has licensing to use it. It was used on University property with their software, so obtaining it is only possible online or through BU.
The second folder is the ReCyclePCB folder which contains all of the EAGLE PCB files used to design and layout the board. There is a zip folder that contains all gerber files to be sent to a fab house. EAGLE PCB can be obtained from the Eagle website. Google"Eagle PCB". If you do not know Eagle, there are great tutorials online giving simple demonstrations. We do not suggest trying to edit this board right away if you are unfamiliar with Eagle.

Schematics and PCB) Included are the final schematics for the PCB as well as top and bottom routing and top and bottom pad layout with silk. These files are also located in ../CAD-CAM/ReCyclePCB.

Technical Specs and Manuals) All necessary/relevant data sheets for components on the PCB are included in this folder. Important points are the BeagleBone SRM manual which has information on GPIO pinout and OMAP_MUX settings.

Vendor Info and BOM) In this file is a subdirectory with vendor information. Most hardware was

bought from digikey using the .digikey file in ReCyclePCB, however some were not and those are listed in text files. The BOM for the PCB is also included in this folder. It is a direct output from Eagle PCB.

Pictures) Pictures contains PDF’s with significant pictures of the hardware assembled and disassembled.