Producing Gerber Files in Altium

Each team will place two Zip files in their final upload folder.
Zip files should be named:

1. ENGR_100_950_Team#_Altium
Should be your project directory, containing your project file, schematic, pcb file, provided schematic and pcb libraries, and all other files native to said directory. Do not put the files I'm asking you to generate below in this zip file!

2. ENGR_100_950_Team#
Should contain only output Gerber files described below. Do not put any other project files in this zip file!

cam.drl -> Drill file
cam.rpt -> Drill report
pcb2.gbo -> Bottom overlay (Only if you are using silkscreen on the bottom layer)
pcb2.gbp -> Bottom paste (Only if you are using surface mounts on the bottom layer)
pcb2.gbl -> Bottom layer
pcb2.gbs -> Bottom solder mask
pcb2.gko -> Keep out
pcb2.gpb -> Bottom pad master
pcb2.gpt -> Top pad master
pcb2.gtl -> Top layer
pcb2.gto -> Top overlay
pcb2.gtp -> Top paste
pcb2.gts -> Top solder mask

How to get these files? Click on File -> Fabrication Outputs -> Gerber Files.

A window should pop up called Gerber Setup.
Click Format 2:3 (or 2:4)
Navigate to the Layers tab, Click on Plot Layers and click Used On.
Navigate to the Drill Drawing tab, Click on Plot all used drill pairs for Drawing and Guide Plots.

Click OK.

A new window should open in your workspace called CAMtastic.
Navigate to the CAMtastic panel on the left side of the screen, in Generated where your Projects Tab is. If you can't see it, look at the options on the lower left side, labelled Files, Projects, Navigator, etc.

Check out the different layers by checking and unchecking the different files listed in the CAMtastic editor panel on the left side. Click on Click on File -> Export -> Gerber...
Click OK on the window that pops up.

You will see a list of file names. Uncheck the ones that are not listed at the top of this page. The only ones remaining checked should be those listed. .DRL and .RPT will not be listed. Additionally, .GBO and .GBP files may not be listed. That is fine.

Make sure to save the files to an easy to access output folder.

Now close CAMtastic. Again, from your PCB file, navigate to File -> Fabrication Outputs -> NC Drill Files

A pop up window will appear. Click Generate EIA Binary Drill File (.DRL) at the bottom of the page. Then click OK.

A window entitled Import Drill Data will appear. Click OK to close it. Another CAMtastic window will open up, with all the holes on your PCB. Click File -> Export -> Drill…

Check the box at the bottom: Write Report file (*.RPT)
Click Save.

In the next window, choose the folder you'd like to save to. Click OK.

Zip em, send em. I'll let you know if I run into problems.