I have taken the time to list some tips that may help you run the Feritas programme to be used to run simulations for your CAE report.

**Programmes to be downloaded**

Please note that the programmes listed here are specifically for windows and may have other alternatives on Macintosh.

* MinGW - To ensure proper functionality, download all packages during installation. Downloading MSYS (Available from the installation packages) is very important.
* FreeCAD - A free 3D modelling software where you can make an object and obtain coordinates
* EPSviewer, Infran, Adobe illustrator – Feritas saves its simulation diagrams in the EPS file format. Windows cannot natively view these files. You will have to download one of the above software to view these files. (I believe Macintosh can natively view these files)
* Download the Feritas zip folder from the server. Extract all the files from Zip folder to a location of your choosing

**Running Feritas**

Once all the above software is downloaded and installed, you need to use command prompt to access the directory where you stored the Feritas folder. I shall list below some commands that should be useful when navigating the command prompt (These are specific to Windows).

dir - List all files in the current directory (similar to ls if you’re using linux)

cd \[new directory] - Change directory/ enter new folder

cd\.. - Return to previous directory

make - This command runs the make file which prepares all necessary files for the Feritas simulation

del [file] - Delete file

* You may have an error when trying to execute the ‘make’ command to run the makefile program located insider the Feritas folder. To use ‘make’, you need to go to the location where you saved MinGW. Inside the MinGW folder, there should be another folder named ‘msys’. Enter the folder and enter the ‘bin’ folder. Copy the directory address and paste it into the command prompt and add ‘\make’. This should fix your problem.
* If you have any other errors after running makefile, enter into the command prompt ‘del \*.o’ which deletes all files with the .o file extension. Run the makefile using ‘make’ command once more.
* Run the ‘Feritas.exe’ program located inside the app folder. You may get a message saying that ‘ the input file was not found’. These input files can be found in either ‘elast, heat, thermoelast’ folders. Copy one of these files into the same directory as Feritas.exe. Note: there are Feritas.exe programs inside the ‘elast, heat, thermoelast’ folders and may not function properly. It is recommended that you run the Feritas.exe file in the previous directory. After running Feritas.exe successfully, you should see files with the .eps file extension that have been saved in the same directory.

**Changing Parameters in Input file**

You have to change the parameters inside the input.txt file to run a unique simulation. However, editing these may be quite cumbersome. A simple solution is to design a Solid in the FreeCAD software. I shall list here the steps you should take to make a design.

* Select ‘Create New…’
* Just below the tool bar, you should see a tab labelled ‘start’. Change ‘Start’ to ‘Draft’.
* Select a shape. Before drawing the shape, put a tick on the ‘Filled (L)’ check box before drawing’.
* If you want to add a hole to your drawing, etc. select another shape and draw on top of your previous drawing.
* Change ‘Draft’ to ‘Part’.
* Select your main body first and then the body which shall make a hole next and then select either the merge, cut, etc. tools from the bottom toolbars (Shown as icons with a bunch of circles merging or cutting).
* Once you’ve created a shape which you are pleased with, change ‘part’ to ‘FEM’.
* Select you shape from the left hand menu and then go into the Mesh tab inside the top most toolbar. Select FEM mesh from shape by Netgen.
* In the left hand menu, deselect the ‘second order’ check box. Adjust the max size of nodes and click apply and then okay. Your shape should have a mesh grid applied to it.
* Select FEMMeshNetgen from the left hand menu and go to the file tab in the top most toolbar. Select export and save the file in a folder of your choosing. The file Save type should be ‘FEM mesh formats’. When writing the file name, write [filename]’.inp’ (Add the .inp file extension).
* Once the file is saved, rename the saved .inp file as [filename].csv. You will be asked to confirm this selection.
* In the CSV file, please clear the contents in the rightmost column for the Nodes section. These are the coordinates for the z axis and are unnecessary in our simulation as we shall only be running a 2D simulation.
* In the Face elements section, please renumber the \*elements column from 1 onwards (similar to how the nodes have been numbered).
* Copy the coordinates from the csv file to the input.txt file. I shall send you images of which data need to be replaced inside the input.txt file.

There should be further adjustment necessary, but I shall let you know as soon as I’ve figured those out.