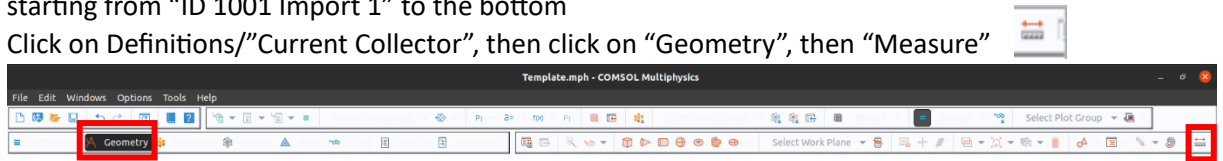


To launch a COMSOL simulation :

1. Open Template.mph (COMSOL 5.6)
2. Click on “Browse” in the Mesh/”Import” tab and select your mesh file (.bdf)
3. Don’t click on “import” but click on “All Domains” in the Definitions/”Selections” tab
4. Once the mesh has been imported, tick “All domains” in Definitions/Selections/”All Domains” tab
5. Click on Definitions/Selections/”CBD”, then click on “Add” (“+” sign) and add “ID 3 Import 1”
6. Click on Definitions/Selections/”Separator”, then click on “Add” (“+” sign) and add “ID 21 Import 1”
7. Click on Definitions/Selections/”NMC811”, then click on “Add” (“+” sign) and add all the items starting from “ID 1001 Import 1” to the bottom
8. Click on Definitions/”Current Collector”, then click on “Geometry”, then “Measure”



9. Click on Definitions/”NMC811”, then click on “Geometry”, then “Measure”
10. Click on Global Definitions/”Parameters”, then copy paste the surface (displayed in the “Messages” tab) of the current collector into the “Surface_CC_Solid” variable, and the volume of the NMC into the “VolumeNMC” variable
11. Press “compute”

Notes:

1. If you are using an electrode with a different thickness, you have to modify in Definitions/Selections/”Box Li Foil” the parameter “z minimum” accordingly. Here $z_{\text{minimum}} = 156 \mu\text{m}$ ($150 \mu\text{m}$ electrode + $6 \mu\text{m}$ separator).
2. If you want to simulate a discharge at a different C-rate you have to modify the “C_rate” parameter in Global Definitions/Parameters. For instance, $C_{\text{rate}} = 0.5 = C/2$. Also, in Study/Step2: Time-Dependent, in “Output times”, modify accordingly the timestep at which the solution will be saved. The format is the following (starting time, timestep, end time).
3. If you encounter a “self-intersecting face” error when importing a mesh, right-click on “Mesh 1”, select Intersections and Partitions/”Detect Faces”, add the ID of the faces self-intersecting based on the error message, and put $\text{Maximum_neighbor_angle} = 20^\circ$