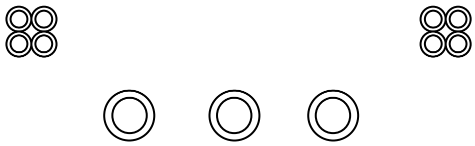
Hello again, Ivar. Thank you very much for your comments. I would first like to apologize for giving insufficient data in my previous messages to describe my problem and also for my English.

What I´m trying to do is to model a magnetic field mitigation system installed in the proximities of underground three-phase power cables in flat formation, which causes the magnetic field I want to mitigate. The mitigation system is done by installing low voltage cables that form a loop. In this loop, the magnetic field I want to mitigate induces currents which create an opposite magnetic field that cancels the field generated by the power cables. To get better mitigation efficiency, the loop impedance can be compensated including a capacitor. This capacitor uses to have so large capacitance values, but can be reduce by using a multi-turn loop, as shown in my sketch (see attached file). So my model has to be an ACDC 2D (non axi) quasi-static magnetic problem with perpendicular induction currents (time harmonic analysis). Of course, skin effects must be included. Since I want to use low voltage cables with cross sections about 120mm2, I don´t think that simulate them as a single turn by adapting area/volume and current is a good choice. Also, later I would like to add thermal coupling, so it would be needed to model the insulation and protection layers that form each cable.

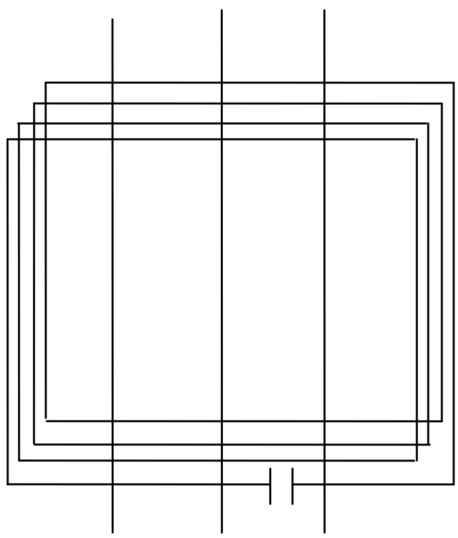
For example the model could like this sketch:



→1 loop with 4 turns (120mm2)

Phases: R S T (Power cables: 630mm2)

Front view



R S T

Plan view

My main goal is to “measure” the currents induced in the loop and the final magnetic field at a certain distance. So the main problem is how to model in comsol such configuration of the loop. In a previous message I commented that I modeled one single loop using “global equations” to force two cables to be parallel. But I don’t know how to do it with more turns and, of course, including the capacitance. I have solved this problem analytically, but there are some approximations that I want to verify with comsol. Moreover, it is difficult to include thermal analysis analytically in this arrangement, so approximations included in the IEC-287 standard must be used, but there are several issues in my model that are not considered in this standard, so finite elements are needed to verify this approximations.

I hope I have explained correctly my problem. Any suggestions would be welcome.

Thank you for your help.