I have two files **Comsol\_innerboundary\_problem.mph** and **Comsol\_outerboundary\_problem.mph**. They both have identical equations and constants. They differ in which boundary the boundary condition is applied. In both the problems there are two spheres. One little one of R=1 inside of a bigger sphere of R=2.

In the **Comsol\_innerboundary\_problem.mph** the inner sphere is excited with a magnetic potential

 

The resulting velocity profile 'v' from x=-1 to +1 at y=z=0 is as follows



It is of the order 10-13 .

Using the same boundary condition but this time applying it to the outer sphere and maintaining a **continuity** boundary condition on the inner sphere gives the following profile



The order of the flow is 10-3

I am aware that the radius is larger which lowers the field internal to the sphere, and Im also aware that demagnetizing effects occur. However I dont think that this is enough of a reason to give such a big discrepancy in the order of the flow.

I have localized the problem to the profile of the magnetization vector. For instance, comparing the magnetization vector in the x direction for the outer sphere excitation gives this profile at t=10s at y=z=0



The difference between the values at x=0 and x=-1 is of the order 10-4 which is about the same as the velocity. I suspect that this is where the problem lies. This difference creates a change in the expression for FMx and that drives the flow. Below is FMx as a function of time



 Looking at the M\_x vector for the case where the inner sphere is excited at the same time and position gives



and the corresponding FMx as a function of time is



How do I resolve this discrepancy? I have increased the boundary layer mesh and the general meshing but havent seen any improvement. Can you give me an insight into why this is happening?