
Subject: Re: Flow3 procedure and WHERE
Posted by [David Fanning](#) on Mon, 24 May 2004 01:56:20 GMT
[View Forum Message](#) <> [Reply to Message](#)

Adhara writes:

- > I updated the program and now I don't have that weird error message.
- > Sometimes it helps just logging off and starting again!

Yes, it sometimes helps. This is why this is the **first** thing a technical support engineer will ask you to do when you call in. It solves about half the problems before he even has to do any thinking. :-)

But that said, every time I resort to exiting IDL I consider it a mortal sin and I head straight for confession. :-(

- > I just sent another message including my updated program because I
- > noticed that I am not getting the Sx,Sy,Sz vectors with the values
- > that I should. I printed the Sx values and they are just a list of
- > numbers from 0 to n, where n coincides with the number of points where
- > the velocity is different from zero!
- > Do you know how can I define them so that they will only include the
- > coordinates at which Vx, Vy and Vz are different from zero?

I have to admit, I am totally confused by your program and your approach. I would be extremely surprised if it works. Your three "V" arrays are three dimensional, are they not? And many of the values are 0. Fair enough. But you are finding the locations where Vx is not equal to zero, then the locations where Vy is not equal to zero, and so forth. While many of these locations will be the same in all three arrays, presumably, many others will not. Using your approach you will very likely end up with three "V" arrays of unequal size. FLOW3 is not going to like that. :-)

What you want, I think, is where Vx and Vy and Vz are all three not equal to zero, but then you can still potentially throw away real data, because surely 0 is a possible vector magnitude. So, I think the approach is suspect, just from a scientific point of view.

I've never used FLOW3, but it seems to be a three-dimensional VELOVECT. Flow lines will be drawn in the 3D space according to the "forces" a point

source will feel as it moves though the volume.
So probably your "S" vectors are irrelevant to solving this problem. Eliminating them and allowing the program to choose random starting coordinates sounds like a good idea to me. At least I can't see any obvious benefit from starting the flow lines on top of the forces the point is suppose to be feeling, which I take is the gist of your current approach.

I take it the original problem was that the force field lines were tiny compared to the volume space. I believe you concluded this was because the force vectors were quite small (10^{-3} or something, if I recall). Have you tried just scaling everything by some factor? Have you tried setting the LEN keyword to FLOW3?

Or maybe you just need to re-think the entire problem. Have you tried making stacks of 2D contour plots to see if you can see anything that way? I don't know which approach might work best. But I think this one is not heading for a good result. :-)

Cheers,

David

--

David Fanning, Ph.D.
Fanning Software Consulting, Inc.
Coyote's Guide to IDL Programming: <http://www.dfanning.com/>
