

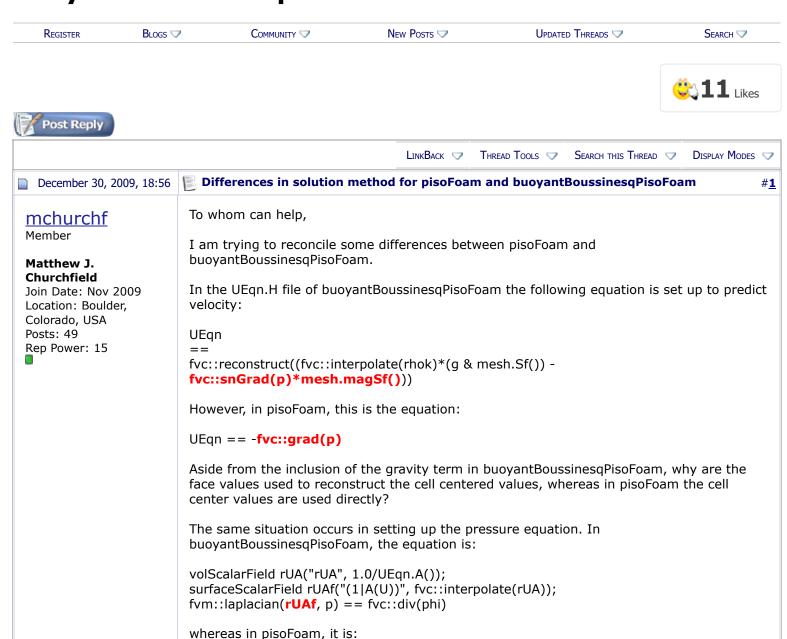


- Home
- News
- Forums
- Wiki
- Links
- Jobs
- Books
- Events
- Tools
- Feeds
- About
- Search

Home > Forums > Software User Forums > OpenFOAM

Differences in solution method for pisoFoam and buoyantBoussinesqPisoFoam





volScalarField rUA = 1.0/UEqn.A(); fvm::laplacian(rUA, p) == fvc::div(phi)

buoyantBoussinesqPisoFoam is solving for pressure on the faces, and pisoFoam is solving for cell centered pressure. Does this make a difference? Why are the two codes different in this manner?

Thank you

rajibroy likes this.





January 10, 2010, 18:27



#2

alberto

Senior Member

Alberto Passalacqua

Join Date: Mar 2009 Location: Ames, Iowa, United States Posts: 1,912 Rep Power: 32 Hi,

in pisoFoam you have the standard implementation, in buoyantBoussinesqPisoFoam the solution algorithm is modified as follows:

- you reconstruct the gravity and the pressure gradient contributions from the corresponding contribution to the flux
- you solve a "pseudo-staggered" version of the pressure equation
- you correct the flux
- you obtain the velocity correction reconstructing from the flux again (remember the flux is always continuous)

This technique tries to mimic a staggered grid arrangement. It is applied to the gravity term too, since it is included in the pressure equation.

Best, Alberto

kaifu, MPJ, mechy and 6 others like this.

Alberto Passalacqua

GeekoCFD - A free distribution based on openSUSE 64 bit with CFD tools, including OpenFOAM. Available as in both physical and virtual formats (current status: http://albertopassalacqua.com/?p=1541)

<u>OpenQBMM</u> - An open-source implementation of quadrature-based moment methods.

To obtain more accurate answers, please specify the version of OpenFOAM you are using. \bigcirc

Last edited by alberto; January 10, 2010 at 18:28. Reason: removed quote



#<u>3</u>





samiam1000

Senior Member

Samuele Z

Join Date: Oct 2009 Location: Mozzate - Co -

Italy Posts: 519 Quote:

Originally Posted by **alberto D**

in pisoFoam you have the standard implementation

Rep Power: 15

Dear Alberto,

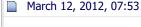
I am trying to understand the different solvers. So, for simpleFoam I've found this wiki... As far as the pisoFoam is concerned, you wrote that it present the standard implementation.

What does this mean? Is there a reference page?

Thanks a lot, Samuele



#4



alberto

0

Senior Member

Alberto Passalacqua

Join Date: Mar 2009 Location: Ames, Iowa, United States Posts: 1,912

Rep Power: 32

Quote:

Originally Posted by **samiam1000 Dear Alberto**,

I am trying to understand the different solvers. So, for simpleFoam I've found this wiki..

As far as the pisoFoam is concerned, you wrote that it present the standard implementation.

What does this mean? Is there a reference page?

Thanks a lot, Samuele

Hi Samuele,

unfortunately I don't think there is a reference page. In pisoFoam you have the standard PISO algorithm you find in books, without body force term and without any particular treatment, except the Rhie-Chow interpolation. You can take a look at the icoFoam page on the wiki and you'll see many similarities.

My statement has to be read in the context of the comparison between the two solvers in the topic, where buoyantBoussinesqPisoFoam uses flux reconstruction to improve the solution procedure when body force terms are included.

Best,

Alberto Passalacqua

<u>GeekoCFD</u> - A free distribution based on openSUSE 64 bit with CFD tools, including OpenFOAM. Available as in both physical and virtual formats (current status: http://albertopassalacqua.com/?p=1541)

OpenQBMM - An open-source implementation of quadrature-based moment methods.

To obtain more accurate answers, please specify the version of OpenFOAM you are using. \bigoplus





March 12, 2012, 07:57

Senior Member

Samuele Z

Join Date: Oct 2009 Location: Mozzate - Co -

Italy Posts: 519 Rep Power: 15 That's great, thanks.

And what about the pimpleFoam solver? Do you know which solver is embedded in such a solver?

Thanks a lot, Samuele







#6

alberto

Senior Member

Alberto Passalacqua

Join Date: Mar 2009 Location: Ames, Iowa, **United States**

Posts: 1,912 Rep Power: 32

The "pimple" solvers use a "combination of PISO and SIMPLE", which is not that far from the flavors you find in other codes with different names (unsteady SIMPLE, iterative PISO, depending on the creativity of the authors :-)).

In short, it is an iterative solution method with sub-iterations over the set of equations to improve the robustness of the algorithm using under-relaxation, and to speed-up transient simulations or perform pseudo-transient simulations.

Best,

samiam1000 likes this.

Alberto Passalacqua

GeekoCFD - A free distribution based on openSUSE 64 bit with CFD tools, including OpenFOAM. Available as in both physical and virtual formats (current status: http://albertopassalacqua.com/?p=1541)

OpenOBMM - An open-source implementation of quadrature-based moment methods.

To obtain more accurate answers, please specify the version of OpenFOAM you are using. ₩





September 11, 2013, 00:24



<u>sharonyue</u>

Senior Member

Dongyue Li

Join Date: Jun 2012 Location: Beijing, China Posts: 776 Rep Power: 13

Quote:

Originally Posted by alberto 2 Hi,

in pisoFoam you have the standard implementation, in buoyantBoussinesqPisoFoam the solution algorithm is modified as follows:

- you reconstruct the gravity and the pressure gradient contributions from the corresponding contribution to the flux
- you solve a "pseudo-staggered" version of the pressure equation
- you correct the flux
- you obtain the velocity correction reconstructing from the flux again (remember the flux is always continuous)

This technique tries to mimic a staggered grid arrangement. It is applied to the gravity term too, since it is included in the pressure equation.

Best, Alberto

Dear Alberto,

Can I say that I have to reconstruct it when there is a body force? Is it a must?

for example, in interFoam's UEqn:

Code:

Can I code it like:

Code:







« Previous Thread | Next Thread »

Posting Rules



You **may not** post new threads You **may not** post replies You **may not** post attachments You **may not** edit your posts

BB code is On
Smilies are On
[IMG] code is On
HTML code is Off
Trackbacks are Off
Pingbacks are On
Refbacks are On

Forum Rules



Similar Threads				8
FVM,FDM AND FEM	d	Main CFD Forum	4	May 30, 2003 04:19
Trouble w. tri. FV method and lam. backstep flow	Steve Reuss	Main CFD Forum	12	December 26, 2001 12:47

All times are GMT -4. The time now is 23:49.

<u>Contact Us</u> - <u>CFD Online</u> - <u>Privacy Statement</u> - <u>Top</u>

