CFD Online
www.cfd-online.com

- [Home](#)
- [News](#)
- [Forums](#)
- [Wiki](#)
- [Links](#)
- [Jobs](#)
- [Books](#)
- [Events](#)
- [Tools](#)
- [Feeds](#)
- [About](#)
- [Search](#)

[Home](#) > [Forums](#) > [Software User Forums](#) > [OpenFOAM](#) > **[OpenFOAM Running, Solving & CFD](#)**

PIMPLE residual control not workin in OF. 6.0

User Name [User Name] ☑ Remember Me

Password [   ] [Log in]

REGISTER   BLOGS ▽   COMMUNITY ▽   NEW POSTS ▽   UPDATED THREADS ▽   SEARCH ▽

😀**9** Likes

[Post Reply]

LINKBACK ▽   THREAD TOOLS ▽   SEARCH THIS THREAD ▽   DISPLAY MODES ▽

📄 March 14, 2019, 15:04	📄 **PIMPLE residual control not workin in OF. 6.0** #**1**

Magistrane
Member

Ilan
Join Date: Dec 2018
Posts: 37
Rep Power: 3
🟩

Dear foamers,

I have a problem when I try to switch my case from OpenFoam 2.3 to OF6.0 in the fvSolutiondict :
My first code was :

Code:

```
PIMPLE
{
    momentumPredictor        yes;
    correctPhi               yes;
    nOuterCorrectors         1000;
    nCorrectors              1;
    nNonOrthogonalCorrectors 1;
    pRefCell                 0;
    pRefValue                0;
    moveMeshOuterCorrectors  yes;
    consistent               yes;

    residualControl
    {
        "(U|p)"
        {
            relTol          0;
            tolerance       1e-6;
        }
    }
}
```

It worked on OF 2.3 exactly as expected : pimple's iterations until reaching residuals convergence.

On 6.0, with this code, I receive an error message :

Code:
```
--> FOAM FATAL ERROR:
Solution convergence criteria specified in PIMPLE.residualControl must be given as single val
```

A read a forum saying that I have to change my code to :

Code:
```
PIMPLE
{
    momentumPredictor         yes;
    correctPhi                yes;
    nOuterCorrectors          1000;
    nCorrectors               1;
    nNonOrthogonalCorrectors  1;
    pRefCell                  0;
    pRefValue                 0;
    moveMeshOuterCorrectors   yes;
    consistent                yes;

residualControl
{
    U   1e-06;
    p   1e-06;
}
```

This is absolutely not working : pimple finish the 1000 iterations before going to the next time step not matter if the residuals are under 1e-06 or not.

Do you have any explanation for me ? Any alternative syntaxe ?

Thanks,
Best regards.

> amuzeshi likes this.

Quote

| March 15, 2019, 06:50 | | #2 |

Magistrane
Member

Ilan
Join Date: Dec 2018
Posts: 37
Rep Power: 3

Up,
residualControl variable name changed.
Remplaced by :

> Quote:
> outerCorrectorResidualControl
> {
> "(U|k|epsilon)" { relTol 0; tolerance 0.0001; } }

> amuzeshi, Gerry Kan, mendokokubun and 3 others like this.

2020/12/1 PIMPLE residual control not workin in OF. 6.0 -- CFD Online Discussion Forums

---

August 22, 2019, 04:30		#**3**

Gerry Kan
Senior Member

Gerry Kan
Join Date: May 2016
Posts: 224
Rep Power: 7

Folks:

Here is a (rhetorical) question ... if the old "residualControl" is now reassigned to "outerCorrectorResidualControl", what does the new "residualControl" monitor?

Thanks, Gerry.

> amuzeshi and SHUBHAM9595 like this.

December 11, 2019, 15:25		#**4**

sjohn2
Member

Stanley John
Join Date: Sep 2018
Posts: 66
Rep Power: 4

Same question here the outerCorrectorResidualControl does not do anything?

May 18, 2020, 09:25		#**5**

sqek
New Member

Join Date: May 2017
Posts: 2
Rep Power: 0

Quote:

> Originally Posted by **sjohn2**
> *Same question here the outerCorrectorResidualControl does not do anything?*

The absolute and relative tolerances both have to be met, so setting relTol to 0 means it will never count as converged
If you only want the absolute tolerance to be checked set relTol to 1
(So opposite to the solvers section of fvSchemes, where any convergence criteria being met will make it stop, as far as I'm aware)

Post Reply

« Previous Thread | Next Thread »

Posting Rules

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 ⊗

Thread	Thread Starter	Forum	Replies	Last Post
Segmentation fault when using reactingFOAM for Fluids	Tommy Floessner	OpenFOAM Running, Solving & CFD	4	April 22, 2018 13:30
chtMultiRegionSimpleFoam turbulent case	Aditya Patil	OpenFOAM Running, Solving & CFD	6	April 24, 2017 23:13
simpleFoam error - "Floating point exception"	mbcx4jc2	OpenFOAM Running, Solving & CFD	12	August 4, 2015 03:20
pimpleFoam: turbulence->correct(); is not executed when using residualControl	hfs	OpenFOAM Running, Solving & CFD	3	October 29, 2013 09:35
calculation stops after few time steps	sivakumar	OpenFOAM Running, Solving & CFD	7	March 17, 2013 07:37

All times are GMT -4. The time now is 11:10.

Contact Us - **CFD Online** - **Privacy Statement** - **Top**

twitter🐦 facebook

© CFD Online