



- **Home**
- **News**
- **Forums**
- Wiki
- Links
- Jobs
- **Books**
- **Events**
- <u>Tools</u>
- <u>Feeds</u>
- **About**
- Search

<u>Home</u> > <u>Forums</u> > <u>Software User Forums</u> > <u>OpenFOAM</u> > <u>OpenFOAM Running, Solving & CFD</u>

## PIMPLE residual control not workin



[Sponsors]

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
                              ves:
    nOuterCorrectors
                              1000;
    nCorrectors
                              1:
    nNonOrthogonalCorrectors 1;
    pRefCell
                              0:
    pRefValue
                              0:
    moveMeshOuterCorrectors yes;
    consistent
                              yes;
residualControl
        1e-06;
    IJ
        1e-06;
    р
```

This is absolutly 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.

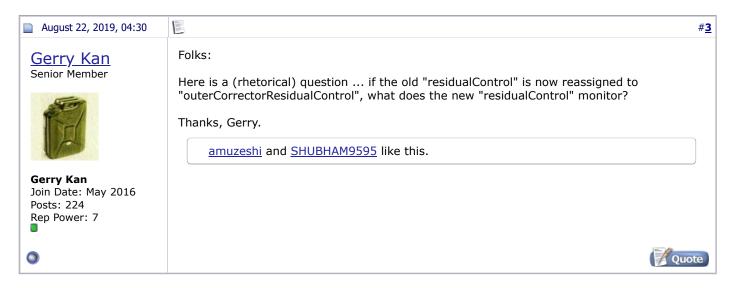
amuzeshi likes this.

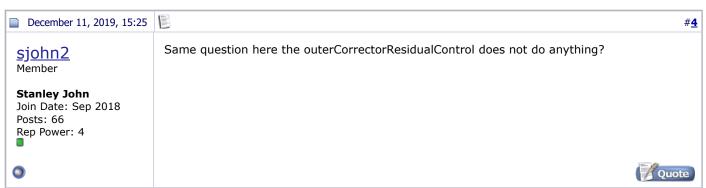
















« Previous Thread | Next Thread »





Similar Threads				8
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

© CFD Online \_