



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

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

## initial residual and final residual

User Name  ☒ Remember Me  
 Password

[REGISTER](#)
[BLOGS](#)
[COMMUNITY](#)
[NEW POSTS](#)
[UPDATED THREADS](#)
[SEARCH](#)

**10** Likes

[Post Reply](#)

[LINKBACK](#)
[THREAD TOOLS](#)
[SEARCH THIS THREAD](#)
[DISPLAY MODES](#)

May 9, 2013, 05:16

**initial residual and final residual**

#1

[mscheng](#)

New Member

**Maosong Cheng**

Join Date: Aug 2012

Posts: 19

Rep Power: 10



Hi FOAMERS,  
 what is the definition of the initial residual and final residual, which can be used to decide convergence ?

[Quote](#)

May 15, 2013, 06:20



#2

[Anne Lincke](#)

Senior Member

**Anne Gerdes**

Join Date: Aug 2010

Location: Hamburg

Posts: 168

Rep Power: 12



The initial residual is evaluated based on the current values of the field before solving an equation for a particular field . The final residual is evaluated after the solution of the equation is performed.  
 The intial residual is more important to decide whether a computation converges, or not.

[Ahmed Khattab](#), [ashish.vinayak](#) and [TommyM](#) like this.

[Quote](#)

May 15, 2013, 09:23



#3

[timo\\_IHS](#)

Member

**Timo K.**

Join Date: Feb 2010  
 Location: University of  
 Stuttgart  
 Posts: 66  
 Rep Power: 12



Hi Anne,

can you give some rule of thumbs for values, which should be achieved for the initial/final residual and how many iterations should be used for pEqn per cycle e.g. for a transient simulation with a standard PCG solver.

Best Timo



May 15, 2013, 09:29



#4

[Anne Lincke](#)

Senior Member

**Anne Gerdes**

Join Date: Aug 2010  
 Location: Hamburg  
 Posts: 168  
 Rep Power: 12



Initial residual:  $e^{-4}$  or  $e^{-5}$  is a good result. Final residual has to be smaller ( $e^{-8}$ ).  
 I cannot answer your 2nd question.



August 17, 2016, 16:45



#5

[Manm](#)

New Member

**Manm**

Join Date: Apr 2015  
 Posts: 16  
 Rep Power: 7



Quote:

Originally Posted by **Anne Lincke**

*The initial residual is evaluated based on the current values of the field before solving an equation for a particular field . The final residual is evaluated after the solution of the equation is performed.  
 The intial residual is more important to decide whether a computation converges, or not.*

Hi, based on this definition, I was curious as to why the final residual in one time step NOT equal to initial residual in NEXT time step?

[Ahmed Khattab](#) and [ginop](#) like this.



August 17, 2016, 16:55



#6

[blais.bruno](#)

Member

**Bruno Blais**

Join Date: Sep 2013  
 Location: Canada  
 Posts: 64  
 Rep Power: 9



Assuming you are solving in steady state, it is because the equations are non-linear.

If you solve the momentum equation for U, at the next iteration, P and phi will have been changed. Therefore, the initial residual will be different from the final residual of the last iteration.

If you were to solve a linear equation, non-coupled with any other variables, in steady-state, then you would solve it in a single iteration... and therefore your

"next initial" residual would be your last final one...

Quote:

Originally Posted by **Manm** 

*Hi, based on this definition, I was curious as to why the final residual in one time step NOT equal to initial residual in NEXT time step?*

[Ahmed Khattab](#), [ashish.vinayak](#), [ginop](#) and [1 others](#) like this.



 August 17, 2016, 18:13



#2

[Manm](#)

New Member

**Manm**


Join Date: Apr 2015

Posts: 16

Rep Power: 7



Quote:

Originally Posted by **blais.bruno** 

*Assuming you are solving in steady state, it is because the equations are non-linear.*

*If you solve the momentum equation for U, at the next iteration, P and phi will have been changed. Therefore, the initial residual will be different from the final residual of the last iteration.*

*If you were to solve a linear equation, non-coupled with any other variables, in steady-state, then you would solve it in a single iteration... and therefore your "next initial" residual would be your last final one...*

Thank you. That helped to understand my question.

If I understand it right,

Say :  $U = f(p, \phi, t)$  where f is a complex function

If  $t_0 = 0$ :

During 1 st iteration, we start with a  $p_0$ ,  $\phi_0$  and dt time step and get  $U_1 = f(p_0, \phi_0, dt)$  as well as  $p_1$ ,  $\phi_1$  values. The corresponding final error/ residual is calculated from  $U_1$  and true U expected.


Before 2nd iteration starts, a (say)  $U_{12}$  is calculated based on new available values:

$U_{12} = f(p_1, \phi_1, dt)$  and this is used to calculate the initial error/ residual before the 2nd time step starts.

and then we go on to calculate  $U_2 = f(p_1, \phi_1, dt+dt)$  and corresponding  $p_2$ ,  $\phi_2$  and final error/ residual at the end of 2nd time step and so on....

[ginop](#) likes this.



 August 18, 2016, 09:31



#8

[blais.bruno](#)

Member

**Bruno Blais**

Join Date: Sep 2013

Location: Canada

Posts: 64

Yup, That is exactly so.

OpenFOAM is a segregated solver, that is U, V, W, phi and P are solved segregated. This is why such a thing happen.

There are block coupled solvers in foam-extend, but those are still linear solver and don't implement a non-linear newton method or something like that.

Rep Power: 9



Quote:

Originally Posted by **Manm** *Thank you. That helped to understand my question.**If I understand it right,**Say :  $U = f(p, \phi, t)$  where  $f$  is a complex function**If  $t_0 = 0$ :**During 1 st iteration, we start with a  $p_0$ ,  $\phi_0$  and  $dt$  time step and get  $U_1 = f(p_0, \phi_0, dt)$  as well as  $p_1$ ,  $\phi_1$  values. The corresponding final error/ residual is calculated from  $U_1$  and true  $U$  expected.**Before 2nd iteration starts, a (say)  $U_{12}$  is calculated based on new available values:* *$U_{12} = f(p_1, \phi_1, dt)$  and this is used to calculate the initial error/ residual before the 2nd time step starts.**and then we go on to calculate  $U_2 = f(p_1, \phi_1, dt+dt)$  and corresponding  $p_2$ ,  $\phi_2$  and final error/ residual at the end of 2nd time step and so on....*[« 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**  
[Refbbacks](#) are **On**

[Forum Rules](#)**Similar Threads**

Thread	Thread Starter	Forum	Replies	Last Post
<a href="#">High Courant Number @ icoFoam</a>	Artex85	OpenFOAM Running, Solving & CFD	11	February 16, 2017 14:40
<a href="#">Extrusion with OpenFoam problem No. Iterations 0</a>	Lord Kelvin	OpenFOAM Running, Solving & CFD	8	March 28, 2016 12:08
<a href="#">pimpleFoam: turbulence-&gt;correct(); is not executed when using residualControl</a>	hfs	OpenFOAM Running, Solving & CFD	3	October 29, 2013 09:35
<a href="#">calculation stops after few time steps</a>	sivakumar	OpenFOAM Running, Solving & CFD	7	March 17, 2013 07:37
<a href="#">Orifice Plate with a fully developed flow - Problems with convergence</a>	jonmec	OpenFOAM Running, Solving & CFD	3	July 28, 2011 06:24

All times are GMT -4. The time now is 10:38.

[Contact Us](#) - [CFD Online](#) - [Privacy Statement](#) - [Top](#)



© CFD Online \_