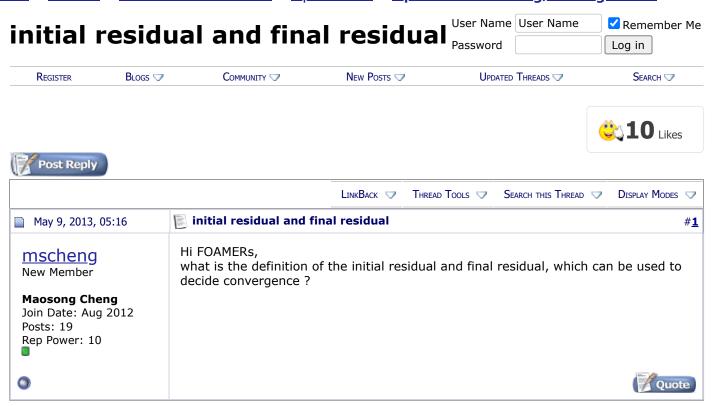
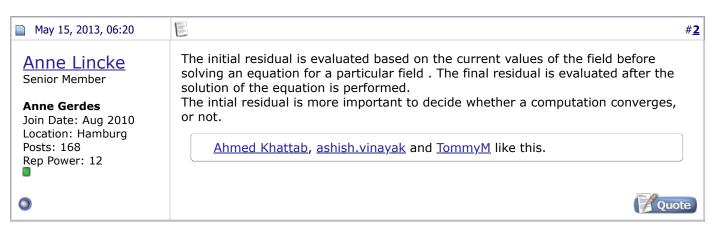




- Home Home
- News
- Forums
- Wiki
- Links
- Jobs
- Books
- Events
- Tools
- Feeds
- About
- Search

Home > Forums > Software User Forums > OpenFOAM > OpenFOAM Running, Solving & CFD







## timo IHS

Member

#### Timo K.

Join Date: Feb 2010 Location: University of

Stuttgart Posts: 66 Rep Power: 12



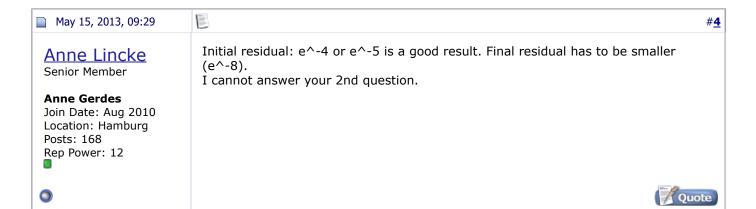
Hi Anne,

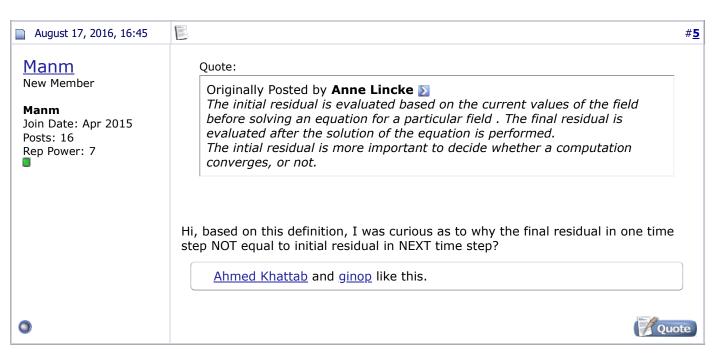
can you give some rule of thumbs for values, which should be achieved for the initial/final residual and how many iterations should be used for pEgn per cycle e.g. for a transient simulation with a standard PCG solver.

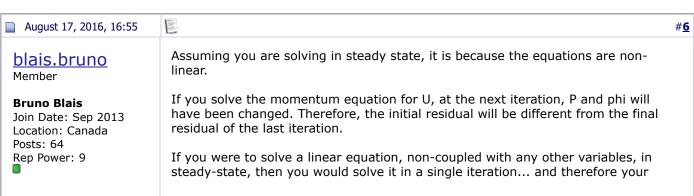
Best Timo











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

## Quote:

Originally Posted by Manm D

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.



#**7** 



# August 17, 2016, 18:13

Manm New Member

## Manm

Join Date: Apr 2015

Posts: 16 Rep Power: 7

## Quote:

1

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 t0 = 0:

During 1 st iteration, we start with a p0, phi0 and dt time step and get U1 = f(p0, phi0,dt) as well as p1, phi1 values. The corresponding final error/ residual is calculated from U1 and true U expected.

Before 2nd iteration starts, a (say) U12 is calculated based on new available values:

U12 = f(p1,phi1,dt) and this is used to calculate the initial error/ residual before the 2nd time step starts.

and then we go on to calculate U2 = f(p1, phi1, dt+dt) and corresponding p2, phi2 and final error/ residual at the end of 2nd time step and so on....

ginop likes this.



#<mark>8</mark>



## August 18, 2016, 09:31

<u>blais.bruno</u> Member 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.

**Bruno Blais** 

Join Date: Sep 2013 Location: Canada Posts: 64 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.

1 0313. 0

## Rep Power: 9

### Quote:

Originally Posted by Manm [3]

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 t0 = 0:

During 1 st iteration, we start with a p0, phi0 and dt time step and get U1 = f(p0, phi0, dt) as well as p1, phi1 values. The corresponding final error/residual is calculated from U1 and true U expected.

Before 2nd iteration starts, a (say) U12 is calculated based on new available values:

U12 = f(p1,phi1,dt) and this is used to calculate the initial error/residual before the 2nd time step starts.

and then we go on to calculate U2 = f(p1, phi1, dt+dt) and corresponding p2, phi2 and final error/residual at the end of 2nd time step and so on....







## « Previous Thread | Next Thread »

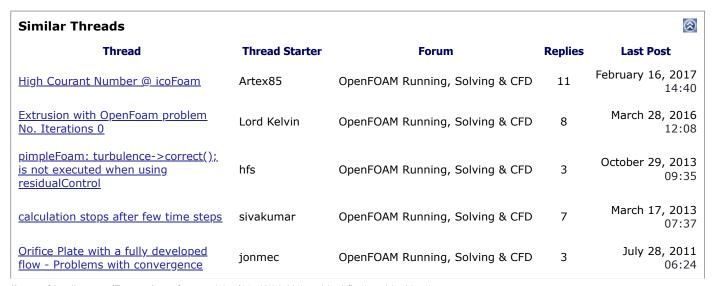
### **Posting Rules**



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 Refbacks are On

Forum Rules



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

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

© CFD Online \_