



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

[Home](#) > [Forums](#) > [Software User Forums](#) > [OpenFOAM](#) > [OpenFOAM Pre-Processing](#)

FvSolution pRefCell and pRefValue

User Name ☒ Remember Me
 Password

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

32 Likes

Post Reply

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

October 17, 2005, 05:54

Can any one explain a little a

#1

[maka](#)

Senior Member

Maka Mohu

Join Date: Mar 2009

Posts: 305

Rep Power: 14



Can any one explain a little about how to set pRefCell and pRefValue in fvSolution dictionary since they have been introduced in 1.2.

I changed the mesh for channelOodles and I got a warning message about pRefCell. Thanks.

```
PISO
{
  nCorrectors 2;
  nNonOrthogonalCorrectors 0;
  pRefCell 1001;
  pRefValue 0;
}
```

Regards,
Maka.

Quote

October 18, 2005, 12:10

If you have an incompressible

#2

[hjasak](#)

Senior Member

Hrvoje Jasak

Join Date: Mar 2009

Location: London,
England

Posts: 1,867

Rep Power: 29

If you have an incompressible flow in a domain where none of the boundaries uses a fixed value (or similar) boundary condition, the value of the pressure in the system is indeterminate to a constant. Iterative solvers don't like that because the solution can jump and down, so we need to do tricks to make sure it does not happen.

The way to do this is to specify the pressure level in one cell. Previously, this used to be cell zero with value zero and hard-coded in the solver, but this causes trouble when the cell is next to a coupled boundary. The new entries allow you to control in which cell the pressure is given and to which level.

Hrv

[pyt](#), [styleworker](#), [CarCin](#) and [14 others](#) like this.

Hrvoje Jasak

 Providing commercial FOAM/OpenFOAM and CFD Consulting: <http://wikki.co.uk>


April 30, 2009, 16:44



#3

[suraj](#)

New Member

Suraj Deshpande

Join Date: Mar 2009

Location: Madison, WI, USA

Posts: 18

Rep Power: 13



Hello,

What is done with pRefCell and pRefValue when atleast one of the boundaries has a fixed value of pressure specified? Is pRefValue still used then?

 Thanks,
Suraj


October 15, 2014, 06:19



#4

[Nicole](#)

Member

Nicole Andrew

Join Date: Sep 2014

Location: Pretoria, South Africa

Posts: 58

Rep Power: 8



Hi Suraj,

I see this is a very old post, but did you ever find an answer to your question bout the fixed boundary pressure?

 Thanks,
Nicole


October 22, 2014, 15:17



#5

[ArathoN](#)

Senior Member

ArathoN

Join Date: Jul 2011

Posts: 137

Rep Power: 12



Quote:

Originally Posted by **Nicole** *Hi Suraj,*
I see this is a very old post, but did you ever find an answer to your question bout the fixed boundary pressure?
*Thanks,
Nicole*

You need to see the solver files and code. In the case of PIMPLE, they are first initialized to zero then it is used a look-up function to scrape their values from fvSolution

from createFields.H

Code:

```
label pRefCell = 0;
scalar pRefValue = 0.0;
setRefCell(p, mesh.solutionDict().subDict("PIMPLE"), pRefCell, pRefValue);
```

Now if you look at the pEqn.H file, in the pressure corrector loop:

Code:

```
fvScalarMatrix pEqn
(
    fvm::laplacian(rAUf, p) == fvc::div(phiHbyA)
);

pEqn.setReference(pRefCell, pRefValue);

pEqn.solve(mesh.solver(p.select(pimple.finalInnerIter())));
```

And here it is used the referenced pressure.

IMO if there is no declaration of pRefCell or pRefValue in pimple they will not be created by createFields.H

Code:

```
setRefCell(p, mesh.solutionDict().subDict("PIMPLE"), pRefCell, pRefValue);
```

So in case of a well-conditioned problem with dirichlet (or mixed) pressure BC, the pRefValue should not be considered.

EDIT: at fvCFD.H included in pimplefoam you'll find the findRefCell.C where it is defined the serRefCell function:

Code:

```
if (fieldRef.needReference() || forceReference)
{.....}
```

So if there is a need to define a reference pressure it will be used otherwise they are neglected. However I can't find where ".needReference()" is defined, which is the function that will tell the solver if it needs or not pRefCell. If I found out something I'll update the post.

EDIT2: I've found it, the function "needReference" is defined GeometricField.C as such:

Code:

```
bool Foam::GeometricField<Type, PatchField, GeoMesh>::needReference() const
{
    // Search all boundary conditions, if any are
    // fixed-value or mixed (Robin) do not set reference level for solution.

    bool needRef = true;

    forAll(boundaryField_, patchi)
    {
        if (boundaryField_[patchi].fixesValue())
        {
            needRef = false;
            break;
        }
    }

    reduce(needRef, andOp<bool>());

    return needRef;
}
```

Now you have the complete picture of how the "referencing process" works on OF.

#6

Post Reply

Forum Rules

Thread	Thread Starter	Forum	Replies	Last Post
Periodic pressure in turb channel_pRefCell	maka	OpenFOAM Running, Solving & CFD	14	October 21, 2010 10:02
General help for fvSchemes and fvSolution settings	harly	OpenFOAM Running, Solving & CFD	4	September 7, 2009 11:31
Programmatically create fvSolution dictionary	cliffoi	OpenFOAM Running, Solving & CFD	14	February 12, 2009 03:50
What must be defined in fvSolution	Marco Kupiainen (Kupiainen)	OpenFOAM Running, Solving & CFD	1	February 21, 2005 06:03

twitter  facebook

© CFD Online