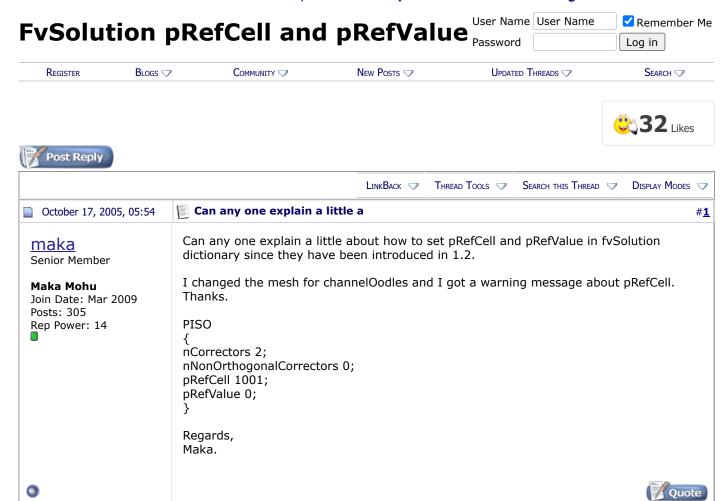
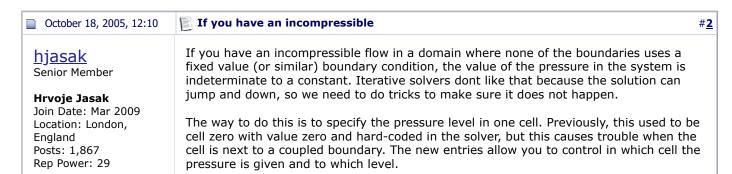




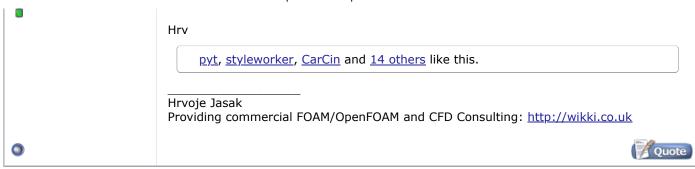
- Home
- News
- Forums
- Wiki
- Links
- Jobs
- Books
- Events
- Tools
- Feeds
- About
- Search

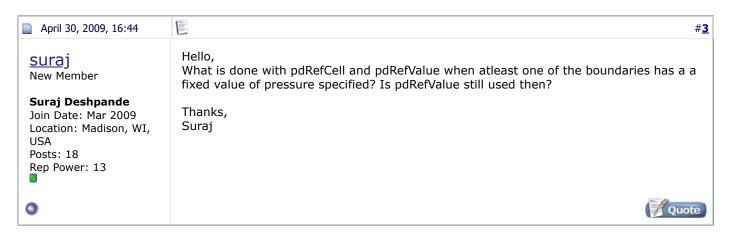
Home > Forums > Software User Forums > OpenFOAM > OpenFOAM Pre-Processing

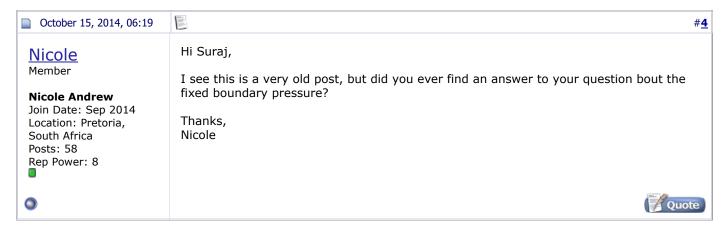


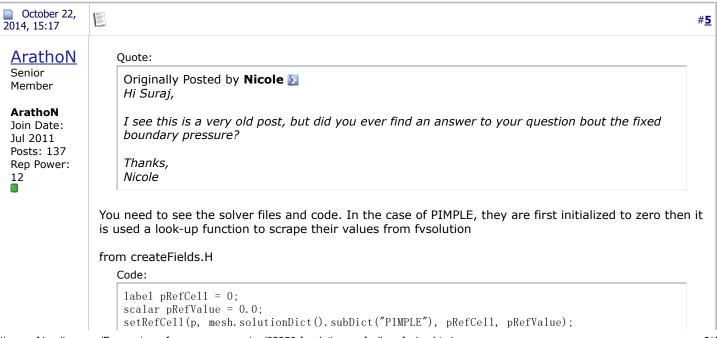


[Sponsors]









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 created by createFields.H

```
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 define, whis 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>());

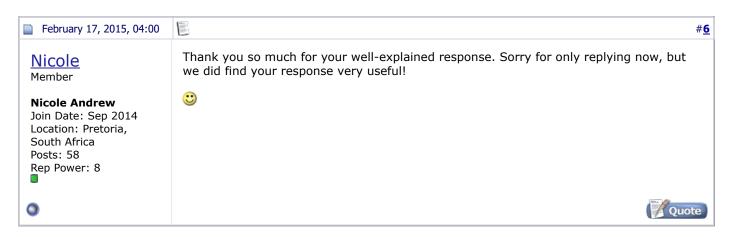
return needRef;
}
```

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

zjdedongxi, Daniel Khazaei, Nicole and 12 others like this.









« Previous Thread | Next Thread »



Similar Threads				(2)
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

All times are GMT -4. The time now is 22:35.

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

