





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

<u>Home</u> > <u>Forums</u> > <u>Software User Forums</u> > <u>OpenFOAM</u> > <u>OpenFOAM Programming & Development</u>

# Building a solver with fixedTemperatureConstraint using fvOptions



SEARCH THIS THREAD

DISPLAY MODES TO



Building a solver with fixedTemperatureConstraint using fvOptions



Fluido New Member

Join Date: Jul 2013

July 29, 2013, 12:41

Posts: 6 Rep Power: 9 Dear Foamers,

I have been using OpenFOAM for several months for a student project. Everything has been working quite well. But at the moment I am a bit stuck at a problem. Maybe someone can help...  $\bigcirc$ 

LINKBACK 🔝

THREAD TOOLS

For version 2.2.0 OpenFOAM seems to have introduced a very nice new feature of fvOptions called 'fixedTemperatureConstraint', to 'to fix the temperature to a given value' (see <a href="http://www.openfoam.org/version2.2.0/fvOptions.php">http://www.openfoam.org/version2.2.0/fvOptions.php</a>). I would like to use this feature to set the temperature of air flowing through a heater to a fixed temperature instead of having to model the heater as a heat source.

My input for the fvOptions file would look like this

### Code:

I would like to include the fixedTemperatureConstraint to a self-built solver for a decoupled, steady state, incompressible temperature equation. The solver is working already. It is just missing the

handling of sources. So, here is what I did:

- add fvOptions(T) to the temperature equation
- add fvOptions.constrain(TEqn) to TEqn.H
- #include "fvIOoptionList.H" to solver .C file
- #include "createFvOptions.H" to solver .C file

The code of the TEqn file looks like this now (not showing the definition of the coefficient alphaEff):

#### Code:

```
fvScalarMatrix TEqn
(
    fvm::div(phi, T)
    - fvm::laplacian(alphaEff, T)
    ==
    fvOptions(T)
);

TEqn. relax();
fvOptions. constrain(TEqn);

TEqn. solve();
```

Compiling of the solver works fine. But when trying to run a case, I get the following error massage.

#### Code:

```
--> FOAM FATAL ERROR:

request for basicThermo thermophysicalProperties from objectRegistry regionO failed available objects of type basicThermo are

0
(
(
)
```

From my understanding of this error message, OpenFOAM is missing a thermophysical model. So I am wondering:

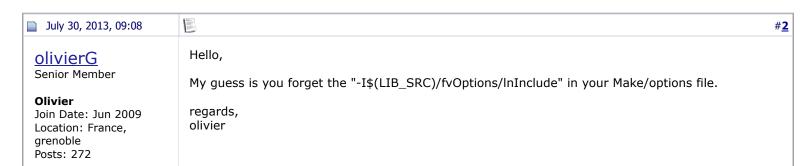
# Is there a thermophysical model necessary for the implementation of the fixedTemperatureConstraint?

If not, do you have any hints where the error messages might come from?

Thank you very much for your help! - Fluido -

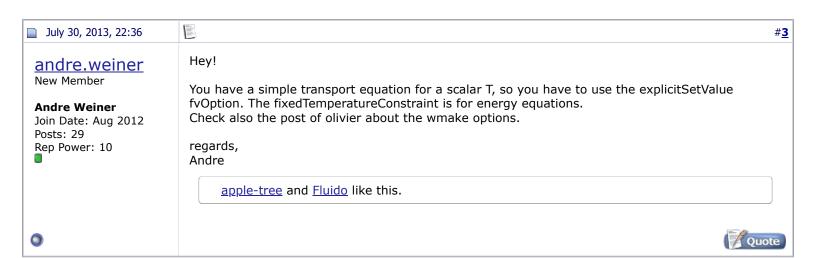
Last edited by Fluido; August 1, 2013 at 05:37.

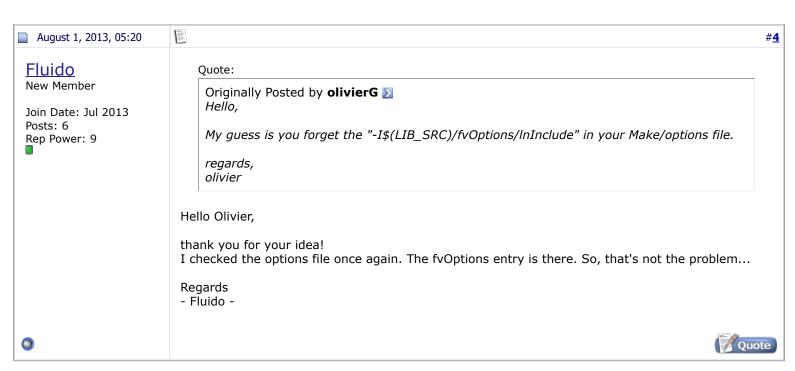


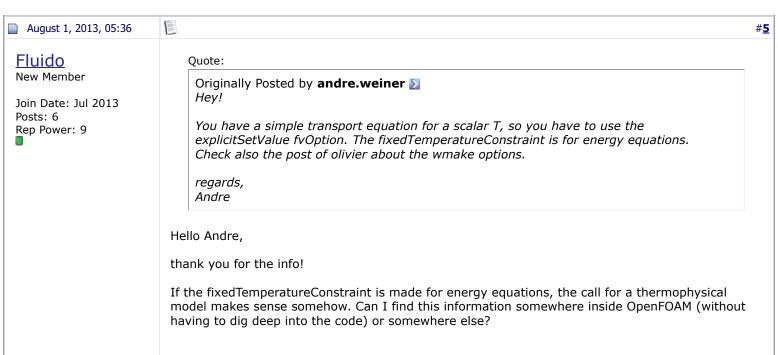










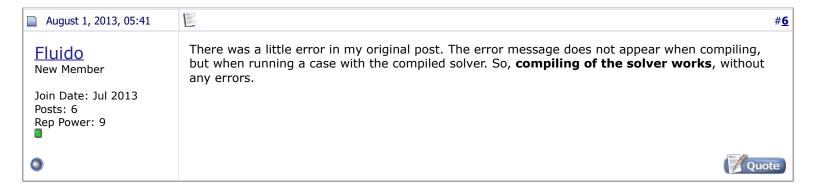


I will try the explicitSetValue option now...

Regards
- Fluido -



#<u>7</u>





August 1, 2013, 06:21

New Member

0

### Andre Weiner

Join Date: Aug 2012 Posts: 29 Rep Power: 10

### Quote:

Originally Posted by **Fluido D** *Hello Andre*,

thank you for the info!

If the fixedTemperatureConstraint is made for energy equations, the call for a thermophysical model makes sense somehow. Can I find this information somewhere inside OpenFOAM (without having to dig deep into the code) or somewhere else?

I will try the explicitSetValue option now...

### Regards

- Fluido -

You don't have to dig that deep :-)

### Code:

```
void Foam::fv::fixedTemperatureConstraint::setValue
(
    fvMatrix<scalar>& eqn,
    const label
)
{
    const basicThermo& thermo =
        mesh_.lookupObject<basicThermo>("thermophysicalProperties");
    if (eqn.psi().name() == thermo.he().name())
{...
```

This is a part of the member function that sets the value. As you see it looks for some thermophysicalProperties and then it checks your equation for a part called psi and so on. So this fixedTemperatureConstraint is a little more complicated than what you need.

Regards, Andre

Fluido likes this.



August 1, 2013, 11:23

Originally Posted by andre.weiner [3]

## <u>Fluido</u>

New Member

Join Date: Jul 2013 Posts: 6 Rep Power: 9 Quote:

```
You don't have to dig that deep :-)

Code:

void Foam::fv::fixedTemperatureConstraint::setValue
{
    fvMatrix<scalar>& eqn,
    const label
}

const basicThermo& thermo =
    mesh_.lookupObject<basicThermo>("thermophysicalProperties");

if (eqn.psi().name() == thermo.he().name())
{...
```

This is a part of the member function that sets the value. As you see it looks for some thermophysicalProperties and then it checks your equation for a part called psi and so on. So this fixedTemperatureConstraint is a little more complicated than what you need.

Regards, Andre

Ok. I had not looked at the .C file yet (or not close enough). But it is quite obvious from code you posted that OpenFOAM would look for a thermophysical model. Thanks!

Meanwhile I tried the explicitSetValue option. I just changed the input in the fvOptions file to:

Code:

And...it works perfectly! :-)

So, thank you again!

- Fluido -

luiscardona, apple-tree, peppino and 1 others like this.



#<u>9</u>

#<u>8</u>

February 14, 2014, 07:25



Dear All,

I am trying to do something like what you did, but instead of temperatures, I wanna add in a certain cellSet a constant bodyForce.

Samuele Z

Join Date: Oct 2009 Location: Mozzate - Co -

Italy Posts: 519 Rep Power: 15 Do you have an idea about I can do this?

Thanks a lot, Samuele





#**10** 

# February 15, 2018, 01:30

# ,

100

### <u>svramana</u>

Member

### Ramana

Join Date: Jul 2017 Location: India Posts: 58 Rep Power: 5

### Quote:

### Originally Posted by Fluido 💟

Ok. I had not looked at the .C file yet (or not close enough). But it is quite obvious from code you posted that OpenFOAM would look for a thermophysical model. Thanks!

Meanwhile I tried the explicitSetValue option. I just changed the input in the fvOptions file to:

### Code:

### And...it works perfectly! :-)

So, thank you again!

- Fluido -

 $\label{thm:linear} \mbox{Hi,i know i am digging into old thread but i am not able to set "scalarExplicitSetValue" in my fvOptions. \\$ 

any help is appreciated.

Regards, s.v.Ramana







### <u>Tags</u>

constraint, fixed temperature, fvoptions, sources, temperature source

« Previous Thread | Next Thread »





You <b>may not</b> post new thread
You may not post replies
You may not post attachment
You <b>may not</b> edit your posts
BB code is <b>On</b>
Smilies are On
[IMG] code is <b>On</b>
HTML code is <b>Off</b>
Trackbacks are Off
Pingbacks are On
Refbacks are <b>On</b>

**Posting Rules** 

Forum Rules

Similar Threads				8
Thread	Thread Starter	Forum	Replies	Last Post
thobois class engineTopoChangerMesh error	Peter_600	OpenFOAM	4	August 2, 2014 10:52
How do I install a custom solver?	NJG	OpenFOAM Programming & Development	5	January 30, 2013 20:03
Interfoam blows on parallel run	danvica	OpenFOAM Running, Solving & CFD	16	December 22, 2012 03:09
<u>Unexplained Error during Solver Runs</u>	cfb	CFX	6	November 9, 2012 16:42
why the solver reject it? Anyone with experience?	bearcat	CFX	6	April 28, 2008 15:08

All times are GMT -4. The time now is 03:45.

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

