

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

[Home](#) > [Forums](#) > [Software User Forums](#) > [OpenFOAM](#) > [OpenFOAM Programming & Development](#)

Building a solver with fixedTemperatureConstraint using fvOptions

User Name ☒ Remember Me
 Password

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



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

 July 29, 2013, 12:41

 **Building a solver with fixedTemperatureConstraint using fvOptions**

#1

[Fluido](#)

New Member

Join Date: Jul 2013

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...😊

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 <http://www.openfoam.org/version2.2.0/fvOptions.php>). 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:

```
fixedTemperaure1
{
    type                fixedTemperatureConstraint;
    active              true;
    selectionMode       cellZone;
    cellZone            heater;

    fixedTemperatureConstraintCoeffs
    {
        mode            uniform;
        temperature     350;
    }
}
```

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 message.

Code:

```
--> FOAM FATAL ERROR:

request for basicThermo thermophysicalProperties from objectRegistry region0 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.



 July 30, 2013, 09:08



#2

[olivierG](#)

Senior Member

Olivier

Join Date: Jun 2009
Location: France,
grenoble
Posts: 272

Hello,

My guess is you forget the "-I\$(LIB_SRC)/fvOptions/lnInclude" in your Make/options file.

regards,
olivier

Rep Power: 14



July 30, 2013, 22:36



#3

[andre.weiner](#)

New Member

Andre Weiner

Join Date: Aug 2012

Posts: 29

Rep Power: 10



Hey!

You have a simple transport equation for a scalar T, so you have to use the explicitSetValue fvOption. The fixedTemperatureConstraint is for energy equations. Check also the post of olivier about the wmake options.

regards,
Andre

[apple-tree](#) and [Fluido](#) like this.



August 1, 2013, 05:20



#4

[Fluido](#)

New Member

Join Date: Jul 2013

Posts: 6

Rep Power: 9



Quote:

Originally Posted by **olivierG**

Hello,

My guess is you forget the "-I\$(LIB_SRC)/fvOptions/lnInclude" in your Make/options file.

*regards,
olivier*

Hello Olivier,

thank you for your idea!

I checked the options file once again. The fvOptions entry is there. So, that's not the problem...

Regards
- Fluido -



August 1, 2013, 05:36



#5

[Fluido](#)

New Member

Join Date: Jul 2013

Posts: 6

Rep Power: 9



Quote:

Originally Posted by **andre.weiner**

Hey!

You have a simple transport equation for a scalar T, so you have to use the explicitSetValue fvOption. The fixedTemperatureConstraint is for energy equations. Check also the post of olivier about the wmake options.

*regards,
Andre*

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 -



August 1, 2013, 05:41

#6

[Fluido](#)

New Member

Join Date: Jul 2013

Posts: 6

Rep Power: 9

There was a little error in my original post. The error message does not appear when compiling, but when running a case with the compiled solver. So, **compiling of the solver works**, without any errors.



August 1, 2013, 06:21

#7

[andre.weiner](#)

New Member

Andre Weiner

Join Date: Aug 2012

Posts: 29

Rep Power: 10

Quote:

Originally Posted by **Fluido**

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.




Fluido

New
Member

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

Quote:

Originally Posted by **andre.weiner** 
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:

```
source1
{
    type            scalarExplicitSetValue;
    active          true;
    selectionMode    cellZone;
    cellZone         fluid-porous;

    scalarExplicitSetValueCoeffs
    {
        volumeMode    absolute;
        injectionRate
        {
            T          323;
        }
    }
}
```

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

So, thank you again!

- Fluido -

[luiscardona](#), [apple-tree](#), [peppino](#) and [1 others](#) like this.



samiam1000

Senior Member

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



February 15,
2018, 01:30



#10

[svramana](#)

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:

```
source1
{
    type            scalarExplicitSetValue;
    active          true;
    selectionMode    cellZone;
    cellZone        fluid-porous;

    scalarExplicitSetValueCoeffs
    {
        volumeMode    absolute;
        injectionRate
        {
            T          323;
        }
    }
}
```

And...it works perfectly! :-)

So, thank you again!

- Fluido -

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

**Tags**

[constraint](#), [fixed temperature](#), [fvoptions](#), [sources](#), [temperature source](#)

« [Previous Thread](#) | [Next Thread](#) »

Posting Rules

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](#)

Similar Threads



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
Unexplained Error during Solver Runs	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.

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

[twitter](#)  [facebook](#) 

© CFD Online _