Understanding temperature coupling BCs

October 28, 2014, 08:44 **Understanding temperature coupling BCs** #1

chriss85
Senior Member

Join Date: Oct 2013
Posts: 394
Rep Power: 15

I'm currently trying to understand the boundary conditions for temperature coupling between two regions, namely compressible::turbulentTemperatureCoupledBaffleMixed and compressible::turbulentTemperatureRadCoupledMixed (source files located in turbulenceModels/compressible/turbulenceModel/derivedFvPatchFields/). In the first BC, radiation is neglected and the problem thus simplifies. In the .C file, one finds the following comment:

Quote:

```
// Both sides agree on
// - temperature : (myKDelta*fld +
nbrKDelta*nbrFld)/(myKDelta+nbrKDelta)
// - gradient : (temperature-fld)*delta
// We've got a degree of freedom in how to implement this in a mixed bc.
// (what gradient, what fixedValue and mixing coefficient)
// Two reasonable choices:
// 1. specify above temperature on one side (preferentially the high side)
// and above gradient on the other. So this will switch between pure
// fixedvalue and pure fixedgradient
// 2. specify gradient and temperature such that the equations are the
// same on both sides. This leads to the choice of
// - refGradient = zero gradient
// - refValue = neighbour value
// - mixFraction = nbrKDelta / (nbrKDelta + myKDelta())
```

I completely agree to and understand the value of the temperature on both sides of the patch. This is just a weighted average of the temperatures in the cells weighted with the heat conductivities divided by the length from center of the cell to the patch

face.
I then assume that the gradient which is written here is only meant for the first side, and that the neighbour side is meant to be (temperature-nbrFld)*nbrDelta (possibly negative when the direction of the surface normal vector is considered).

Now the two strategies below are meant to be used in iterative solvers I suppose. Does anyone know how the second one can be derived from the formulas? Or is it somewhat empirically determined to accelerate convergence maybe?

For the second BC, things become more difficult because there are additional radiative heat fluxes, and no explaining comment. The strategy which is used here appears to be a bit different. The formulas used are:

$$refValue_o = \frac{K_{\Delta_n} \cdot T_n}{\alpha} = \frac{\frac{\kappa_o}{\Delta_o} \cdot T_n}{\frac{\kappa_n}{\Delta_n} - \frac{Qr_o + Qr_n}{T_o}}$$

$$refGrad = 0$$

$$valueFraction = \frac{\alpha}{\alpha + K_{\Delta_o}} = \frac{\frac{\kappa_n}{\Delta_n} - \frac{Qr_o + Qr_n}{T_o}}{\frac{\kappa_n}{\Delta_n} - \frac{Qr_o + Qr_n}{T_o} + \frac{\kappa_o}{\Delta_o}}$$

, where o=owner, n=neighbour, Delta=distance between cell center and patch face, kappa heat conductivity and Qr radiative heat flux.

Does anyone understand this or has seen a derivative? I think this could be interesting for different heat transfer applications at boundaries.

roucho, ahmmedshakil, zfaraday and 8 others like this.

Quote

October 31, 2014, 09:31 #2

chriss85
Senior Member

Join Date: Oct 2013
Posts: 394
Rep Power: 15

I've spent some more time thinking about this. Using the equations above inserted into the mixed BC formula:

$$T_{F_0} = \frac{\alpha}{\alpha + \frac{\kappa_1}{\Delta_1}} \frac{\frac{\kappa_1}{\Delta_1} T_{F_1}}{\alpha} + (1 - \frac{\alpha}{\alpha + \frac{\kappa_1}{\Delta_1}}) T_0 = \frac{\alpha}{\alpha + \frac{\kappa_1}{\Delta_1}} \frac{\frac{\kappa_1}{\Delta_1} T_{F_1}}{\alpha} + \frac{\frac{\kappa_0}{\Delta_0}}{\alpha + \frac{\kappa_1}{\Delta_1}} T_0 = \frac{\frac{\kappa_1}{\Delta_1}}{\alpha + \frac{\kappa_1}{\Delta_1}} T_{F_1} + \frac{\frac{\kappa_0}{\Delta_0}}{\alpha + \frac{\kappa_1}{\Delta_1}} T_0$$

, here with 0: first region, 1: second region, so T0: temperature in first region cell, T_F_0: temperature in first region patch.
I'm wondering if the signs at the radiation fluxes are correct. For positive Q_r (meaning outgoing radiation) an iteration procedure using this formula is bound to diverge, because the sum of the two weighting factors in front of the temperatures is not 1???

As an example: Consider radiation coming out of a gas (0) being absorbed completely by a solid (1). In this case, Qr_0 > 0, Qr_1 = 0 and the sum of both factors is larger than 1. In my understanding this should result in an increasing temperature on each iteration.

I'm going to test this series one dimensionally with MATLAB to investigate the convergence properties. Does anyone have any better explanation for the reasoning behind these formulas or maybe a literature recommendation?

roucho, atulkjoy, Zhiheng Wang and 2 others like this.

Quote

November 3, 2014, 12:06 #3

chriss85
Senior Member

Join Date: Oct 2013
Posts: 394
Rep Power: 15

See http://www.cfd-online.com/Forums/ope...tml#post517173 for further results.

atulkjoy and Zhiheng Wang like this.

Quote

November 26, 2018, 01:20 #4

atulkjoy
Member

Atul Kumar
Join Date: Dec 2015
Location: National Centre for Combustion Research and Development
Posts: 48
Rep Power: 6

Quote:

Originally Posted by **chriss85**
I've spent some more time thinking about this. Using the equations above inserted into the mixed BC formula:

$$T_{F0} = \frac{\alpha}{\alpha + \frac{\kappa_1}{\Delta_1}} \frac{\frac{\kappa_1}{\Delta_1} T_{F1}}{\alpha} + (1 - \frac{\alpha}{\alpha + \frac{\kappa_1}{\Delta_1}}) T_0 = \frac{\alpha}{\alpha + \frac{\kappa_1}{\Delta_1}} \frac{\frac{\kappa_1}{\Delta_1} T_{F1}}{\alpha} + \frac{\frac{\kappa_0}{\Delta_0}}{\alpha + \frac{\kappa_1}{\Delta_1}} T_0 = \frac{\frac{\kappa_1}{\Delta_1}}{\alpha + \frac{\kappa_1}{\Delta_1}} T_{F1} + \frac{\frac{\kappa_0}{\Delta_0}}{\alpha + \frac{\kappa_1}{\Delta_1}} T_0$$

, here with 0: first region, 1: second region, so T0: temperature in first region cell, T_F_0: temperature in first region patch.
I'm wondering if the signs at the radiation fluxes are correct. For positive Q_r (meaning outgoing radiation) an iteration procedure using this formula is bound to diverge, because the sum of the two weighting factors in front of the temperatures is not 1???

As an example: Consider radiation coming out of a gas (0) being absorbed completely by a solid (1). In this case, Qr_0 > 0, Qr_1 = 0 and the sum of both factors is larger than 1. In my understanding this should result in an increasing temperature on each iteration.

I'm going to test this series one dimensionally with MATLAB to investigate the convergence properties. Does anyone have any better explanation for the reasoning behind these formulas or maybe a literature recommendation?

HI Hi chriss

Did you find a way to couple radiation of solid and gas phase. ????

Quote

November 26, 2018, 01:24 **Coupled Boundaries** #5

atulkjoy
Member

Atul Kumar
Join Date: Dec 2015
Location: National Centre for

Quote:

Originally Posted by **chriss85**
I've spent some more time thinking about this. Using the equations above inserted into the mixed BC formula:

Combustion Research and Development
Posts: 48
Rep Power: 6

$$T_{F0} = \frac{\alpha}{\alpha + \frac{\kappa_1}{\Delta_1}} \frac{\frac{\kappa_1}{\Delta_1} T_{F1}}{\alpha} + (1 - \frac{\alpha}{\alpha + \frac{\kappa_1}{\Delta_1}}) T_0 = \frac{\alpha}{\alpha + \frac{\kappa_1}{\Delta_1}} \frac{\frac{\kappa_1}{\Delta_1} T_{F1}}{\alpha} + \frac{\frac{\kappa_0}{\Delta_0}}{\alpha + \frac{\kappa_1}{\Delta_1}} T_0 = \frac{\frac{\kappa_1}{\Delta_1}}{\alpha + \frac{\kappa_1}{\Delta_1}} T_{F1} + \frac{\frac{\kappa_0}{\Delta_0}}{\alpha + \frac{\kappa_1}{\Delta_1}} T_0$$

, here with 0: first region, 1: second region, so T0: temperature in first region cell, T_F_0: temperature in first region patch.
I'm wondering if the signs at the radiation fluxes are correct. For positive Q_r (meaning outgoing radiation) an iteration procedure using this formula is bound to diverge, because the sum of the two weighting factors in front of the temperatures is not 1???

As an example: Consider radiation coming out of a gas (0) being absorbed completely by a solid (1). In this case, Qr_0 > 0, Qr_1 = 0 and the sum of both factors is larger than 1. In my understanding this should result in an increasing temperature on each iteration.

I'm going to test this series one dimensionally with MATLAB to investigate the convergence properties. Does anyone have any better explanation for the reasoning behind these formulas or maybe a literature recommendation?

Quote:

Originally Posted by **chriss85**
I'm currently trying to understand the boundary conditions for temperature coupling between two regions, namely compressible::turbulentTemperatureCoupledBaffleMix ed and compressible::turbulentTemperatureRadCoupledMixed (source files located in turbulenceModels/compressible/turbulenceModel/derivedFvPatchFields/). In the first BC, radiation is neglected and the problem thus simplifies. In the .C file, one finds the following comment:

*I completely agree to and understand the value of the temperature on both sides of the patch. This is just a weighted average of the temperatures in the cells weighted with the heat conductivities divided by the length from center of the cell to the patch face. I then assume that the gradient which is written here is only meant for the first side, and that the neighbour side is meant to be (temperature-nbrFld)*nbrDelta (possibly negative when the direction of the surface normal vector is considered).*

Now the two strategies below are meant to be used in iterative solvers I suppose. Does anyone know how the second one can be derived from the formulas? Or is it somewhat empirically determined to accelerate convergence maybe?

For the second BC, things become more difficult because there are additional radiative heat fluxes, and no explaining comment. The strategy which is used here appears to be a bit different. The formulas used are:

$$refValue_o = \frac{K_{\Delta_n} \cdot T_n}{\alpha} = \frac{\frac{\kappa_o}{\Delta_o} \cdot T_n}{\frac{\kappa_n}{\Delta_n} - \frac{Qr_o + Qr_n}{T_o}}$$

$$refGrad = 0$$

$$valueFraction = \frac{\alpha}{\alpha + K_{\Delta_o}} = \frac{\frac{\kappa_n}{\Delta_n} - \frac{Qr_o + Qr_n}{T_o}}{\frac{\kappa_n}{\Delta_n} - \frac{Qr_o + Qr_n}{T_o} + \frac{\kappa_o}{\Delta_o}}$$

, where o=owner, n=neighbour, Delta=distance between cell center and patch face, kappa heat conductivity and Qr radiative heat flux.

Does anyone understand this or has seen a derivative? I think this could be interesting for different heat transfer applications at boundaries.

Nice work Chriss

Quote

November 26, 2018, 12:38

#6

Bloerb
Senior Member

Join Date: Sep 2013
Posts: 255
Rep Power: 14

At the interface between fluid and solid (or solid and solid) the following is true:

$Q_s = -Q_f$ and $T_s = T_f$

This means, that the heat flux exiting one domain enters the other and that both regions agree on temperature.

The heat flux can be written as:

$\kappa_s \nabla T_s = Q_s = -Q_f = -\kappa_f \nabla T_f$

The gradient at the wall can be expressed as the difference between the value at the cell center and wall face devided by the distance between those. OpenFOAM uses the inverse of that however: $\Delta = \frac{1}{\delta}$.

$$\nabla T = \frac{T_c - T_f}{\delta} = \Delta(T_c - T_f)$$

From here it is only a bit of math. We summarize the above condition:

$$\kappa_s \Delta_s (T_{cs} - T_s) = -\kappa_f \Delta_f (T_{cf} - T_f)$$

and simplify with $T_s = T_f = T_{wall}$. This depends on the region you want to use the boundary condition for. For illustration we'll use the fluid side.

$$\kappa_s \Delta_s (T_{cs} - T_f) = -\kappa_f \Delta_f (T_{cf} - T_f)$$

$$\Rightarrow T_f = T_{cs} \left(\frac{\kappa_s \Delta_s}{\kappa_s \Delta_s + \kappa_f \Delta_f} \right) + T_{cf} \left(\frac{\kappa_f \Delta_f}{\kappa_s \Delta_s + \kappa_f \Delta_f} \right)$$

a mixed boundary condition in OpenFOAM is defined as follows:

$$T_f = \text{valueFraction} \cdot \text{refValue} + (1 - \text{valueFraction})(T_c + \Delta \cdot \text{refGrad})$$

We can rewrite the above formula to match this:

$$\Rightarrow T_f = T_{cs} \left(\frac{\kappa_s \Delta_s}{\kappa_s \Delta_s + \kappa_f \Delta_f} \right) + T_{cf} \left(1 - \frac{\kappa_s \Delta_s}{\kappa_s \Delta_s + \kappa_f \Delta_f} \right)$$

The actual implementation

Code:

```
this->refValue() = nbrIntFld(); // This is T_s

this->refGrad() = 0.0;

this->valueFraction() = nbrKDelta()/(nbrKDelta() + myKDelta()); This is kappa*Delta
```

This is the derivation for the boundary condition.

For adding in heat fluxes due to radiation you'd slightly modify the initial set up
$Q_s + Q_{rads} = -Q_f - Q_{radf}$

I have however not referenced this to the code. And am unsure about the sign of the radiation flux definition in OpenFOAM, so this might differ. It should however be a starting point for understanding the code.

snak, Daniel_Khazaei, Munki and 10 others like this.

Last edited by Bloerb; November 27, 2018 at 04:05.

Quote

Post Reply

« 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

Similar Threads

Thread	Thread Starter	Forum	Replies	Last Post
[openSmoke] libOpenSMOKE	Tobi	OpenFOAM Community Contributions	553	May 27, 2020 10:25
Static Temperature / Opening Temperature	JulianP	CFX	12	April 10, 2019 19:00
UDF for Back-flow Temperature	G340	Fluent UDF and Scheme Programming	3	August 21, 2013 05:56
Fluent Ansys temperature coupling	tensun	Fluent UDF and Scheme Programming	0	November 14, 2010 06:30
high temperature in a coupling simulation	sheintz	STAR-CCM+	3	September 30, 2010 14:56

All times are GMT -4. The time now is 06:32.