GWDG
Gesellschaft für wissenschaftliche Datenverarbeitung mbH Göttingen

OpenFOAM Debugging

Patrick Höhn (patrick.hoehn@uni-goettingen.de)

02/08/2023

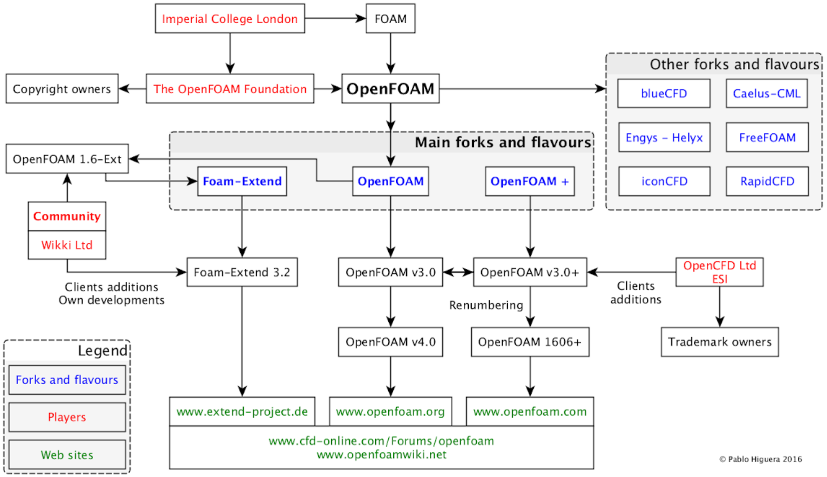
Agenda

- Motivation
- Requirements
- Tools
  - basic debugging using information written to the terminal
  - `DebugSwitches` in controlDict
  - terminal based debugging using gdb and gdbOF
  - memory checking using valgrind
  - GUI based debugging using Visual Studio Code
  - GUI based debugging using Qtcreator
- Further Reading and References

Motivation

- Code not working as intented
- Unterstanding of running code
- Fixing of runtime error
- difficult to study compiled code

OpenFOAM



Source: https://olaflow.github.io/blog/what-is-what-and-who-is-who-in-the-openfoam-environment/

Requirements - OpenFOAM base installation Ubuntu 20.04 LTS

Install base packages

```
sudo apt-get update
sudo apt-get install git-core build-essential binutils-dev \
      cmake flex zlib1g-dev libncurses5-dev curl bison \
      libxt-dev rpm mercurial graphviz python python-dev \
      gcc-7 g++-7 paraview gdb
```

clone foam-extend repository

```
mkdir ~/foam
cd ~/foam
git clone http://git.code.sf.net/p/foam-extend/foam-extend-4.1 \
      foam-extend-4.1
```

Requirements - OpenFOAM base installation Ubuntu 20.04 LTS

Modify settings

```
cd ~/foam/foam-extend-4.1
echo "export WM_CC='gcc-7'"  >> etc/prefs.sh
echo "export WM_CXX='g++-7'"  >> etc/prefs.sh
source etc/bashrc
sed -i -e 's=rpmbuild --define=rpmbuild --define \
      "_build_id_links none" --define=' \
      ThirdParty/tools/makeThirdPartyFunctionsForRPM
sed -i -e 's/gcc/\$(WM_CC)/' wmake/rules/linux64Gcc/c
sed -i -e 's/g++/\$(WM_CXX)/' wmake/rules/linux64Gcc/c++
```

Requirements - OpenFOAM base installation Ubuntu 20.04 LTS

Define aliases

```
echo "alias fe41='source \
      \$HOME/foam/foam-extend-4.1/etc/bashrc'" >> ~/.bashrc
echo "alias fe41_debug='source \
      \$HOME/foam/foam-extend-4.1/etc/bashrc \
      WM_COMPILE_OPTION=Debug'" >> ~/.bashrc
```

Compile foam-extend

```
cd ~/foam/foam-extend-4.1
source ~/foam/foam-extend-4.1/etc/bashrc
./Allwmake.firstInstall
WM_COMPILE_OPTION=Debug ./Allwmake
```

Requirements - Tools

Using package management system

```
sudo apt-get install valgrind qtcreator
snap install code --classic
```

Manual installation - gdbOF

```
source $HOME/foam/foam-extend-4.1/etc/bashrc \
      WM_COMPILE_OPTION=Debug
cd ~/foam/
git clone https://gitlab.com/flowcrunchpublic/gdbof.git gdbOF
cd gdbOF
./installgdbOF.sh
```

Tools - basic debugging by text messages to terminal

```
Info<< "\nStarting time loop\n" << endl;

while (runTime.loop())
{
    Info<< "Time = " << runTime.timeName() << nl << endl;
    ...
}
```

Downsides:

- no control of code during execution
- requires modification of code and recompilation
- cleaning of code required after successful debugging

Tools - DebugSwitches

- 0 means no debug information
- different debug levels available (1,2,3...)
- no recompilation of coded needed unlike `Info` statements
- list and activate debugSwitches of solver (OpenFOAM foundation and ESI)

```
pisoFoam -listRegisteredSwitches
pisoFoam -debug-switch <name=val>
```

- list and activate debugSwitches of solver (foam-extend)

```
pisoFoam -dumpControlSwitches
pisoFoam -DebugSwitches <key1=val1,key2=val2, ...>
```

Hands-on - debugSwitches

```
fe41_debug
cd $FOAM_TUTORIALS/basic/scalarTransportFoam/pitzDaily/
blockMesh
scalarTransportFoam —DebugSwitches volScalarField=1,volVectorField=2

# Output without DebugSwitches:
Time = 0.05

BiCGStab:  Solving for T, Initial residual = 1, Final residual = 0, No Iterations 1

# Output with DebugSwitches:
Time = 0.05

GeometricField<Type, PatchField, GeoMesh>::GeometricBoundaryField::updateCoeffs()
GeometricField<Type, PatchField, GeoMesh>::GeometricBoundaryField::updateCoeffs()
GeometricField<Type, PatchField, GeoMesh>::GeometricBoundaryField::updateCoeffs()
GeometricField<Type, PatchField, GeoMesh>::GeometricBoundaryField::GeometricBoundaryField(const
      ↪ GeometricBoundaryField<Type, PatchField, BoundaryMesh>&)
GeometricField<Type, PatchField, GeoMesh>::GeometricField : constructing as copy resetting IO params
IOobject: volScalarField T_0 "/home/hoehn7/foam/foam—extend—4.1/tutorials/basic/scalarTransportFoam/swirlTest/0.05"

BiCGStab:  Solving for T, Initial residual = 1, Final residual = 0, No Iterations 1
GeometricField<Type, PatchField, GeoMesh>::GeometricBoundaryField::evaluate()
```

Tools - gdb

- study code run-time behaviour
- examining variables at run-time
- changing program at run-time
- BUT: more disc space required and slower execution time
- for own solver or library modification of `Make/options` needed:

```
EXE_INC = -O0 -fdefault-inline -ggdb -DFULLDEBUG
```

Tools - gdbOF and valgrind

gdbOF

- simplification of commands from pure GDB usage by additionally implemented additional macros
- easier inspection of data structures of OF at run-time
- dumping of data at run-time
- currently unmaintained and buggy

valgrind

- Open-Source framework for memory debugging, memory leak detection and profiling
- no profiling demonstrated as part of this talk

Tools - Visual Studio Code

- multiplatform (Windows, macOS, Linux) IDE (Integrated Development Environment) Visual Studio Code
- open source variant without telemetry VSCodium
- supported languages: C, C#, C++, JavaScript, Julia, Perl, Rust, ...
- syntax highlighting, auto completion, revision management
- graphical debugging

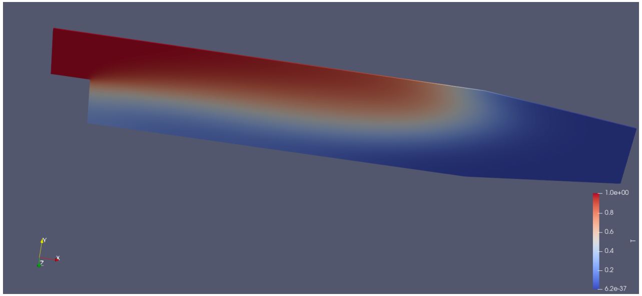
Tools - Qtcreator

- multiplatform (Windows, macOS, Linux) IDE (Integrated Development Environment)
- based on and integrated in multiplatform Qt framework
- supported languages: C++, Java, Markdown, JavaScript, Python, QML, ...
- syntax highlighting, auto completion, revision management
- graphical debugging

Hands-on - Pitzdaily

scalarTransportFoam:

$$\frac{\delta T}{\delta t} + \nabla \cdot (uT) - \nabla \cdot (\nabla D_T T) = S$$



Hands-on - gdb

```
fe41_debug
cd $FOAM_TUTORIALS/basic/scalarTransportFoam/pitzDaily/
blockMesh
gdb scalarTransportFoam
```

command	**meaning**
[h]elp	list all classes of commands
help class	one-line description for commands in class
help command	description of command
[l]ist	shows ten lines after previous listing
list main	shows first ten lines around function
list 30,50	shows defined line range
list createTime.H:1,20	shows first twenty lines of createTime.H

Hands-on - gdb 2

command	meaning
[r]un	start your program with current argument list
run arglist	start your program with arglist
[c]ontinue	resumes execution of program until next breakpoint
ctrl-x ctrl-a	starting of Text User Inferface (TUI) Mode
tui enable	starting of Text User Inferface (TUI) Mode
ctrl-l	redraw TUI window
[b]ack[t]race	print trace of all frames in stack
backtrace n	print trace of n frames in stack
frame n	select frame number n

Hands-on - gdb 3

command	**meaning**
[b]reak (file:)line break (file:)line if var==value	set breakpoint at line (in file) conditional breakpoint at line (in file) if var equal value
clear	clear all breakpoints
whatis expr ptype expr	show datatype of expression show more details on datatype compared to whatis
watchpoint expr	set a watchpoint for expression expr
[s]tep step n	run next line, stepping into function calls run n lines, stepping into function calls

Hands-on - gdb 4

command	**meaning**
[n]ext	run next line, stepping over function calls
next n	run n lines, stepping over function calls
info break	show defined breakpoints
info watch	show defined watch points
finish	run until stack frame returns
[p]rint *T_.v_@nvalues	print first n values of T_ variable
p *T_.v_@(begin,end)	print values from begin to end of T_ variable
call Foam::min(1,2)	call min function from inside OpenFOAM
[q]uit	quit gdb

Hands-on - Valgrind

```
fe41_debug
cd $FOAM_TUTORIALS/basic/scalarTransportFoam/pitzDaily/
blockMesh
valgrind --leak-check=full --show-leak-kinds=all \
    --track-origins=yes --log-file="memcheck.txt" \
    myScalarTransportFoam
```

```
fe41_debug
cd $WM_PROJECT_USER_DIR/applications/
tar xzvf my.tar.gz
cd myScalarTransportFoam/
{vi|nano|emacs|gedit} # editor of your choice
wmake
```

Hands-on - Visual Studio Code - General instructions

- run ’fe41_debug’ in shell
- run ’blockMesh" in same terminal
- run ’code .’ from same shell

Hands-on - Visual Studio Code - tasks.json

```
{
    "version": "2.0.0",
    "tasks": [
        {
            "type": "shell",
            "label": "wmake-build",
            "command": "wmake",
            "problemMatcher": [],
            "group": {
                "kind": "build",
                "isDefault": true
            }
        }
    ]
}
```

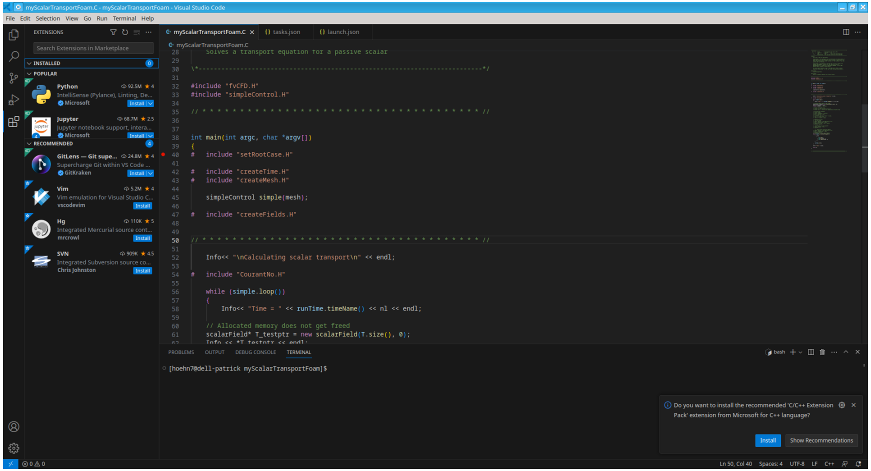
Hands-on - Visual Studio Code - launch.json

```
 1 {
 2   "version": "0.2.0",
 3   "configurations": [
 4         {
 5             "name": "OF—Debug",
 6             "type": "cppdbg",
 7             "request": "launch",
 8             "program": "${env:FOAM_APPBIN}/myScalarTransportFoam",
 9             "args": [],
10             "stopAtEntry": false,
11             "cwd": "${env:FOAM_TUTORIALS}/basic/scalarTransportFoam/pitzDaily/",
12             "environment": [],
13             "externalConsole": false,
14             "MIMode": "gdb",
15             "miDebuggerPath": "/usr/bin/gdb",
16             "setupCommands": [
17                 {
18                     "description": "Enable pretty—printing for gdb",
19                     "text": "—enable—pretty—printing",
20                     "ignoreFailures": true
21                 }
22             ],
23             "preLaunchTask": "wmake—build"
24         }
25     ]
26 
27 }
```

Description:
8: Location of executable program
9: Arguments to pass to the programm
10: Should program stop at the beginning of the main function?
11: Path to case folder
12: Environment variables to add to the environment of the program
14: What debugger should Visual Studio Code connect to?
15: Path of debugger. Don't know? Use which gdb in another terminal
16-22: Array of commands to execute in order to setup GDB
23: Task to be performed before debugging

Hands-on - Visual Studio Code - Steps

After first start after installation click "Install" to add c++ specific modules



myScalarTransportFoam.C - myScalarTransportFoam - Visual Studio Code

File Edit Selection View Go Run Terminal Help
EXTENSIONS
Search Extensions in Marketplace
INSTALLED 0
POPULAR
RECOMMENDED 4
myScalarTransportFoam.C
tasks.json
launch.json
PROBLEMS OUTPUT DEBUG CONSOLE TERMINAL
bash
Ln 50, Col 40 Spaces: 4 UTF-8 LF C++

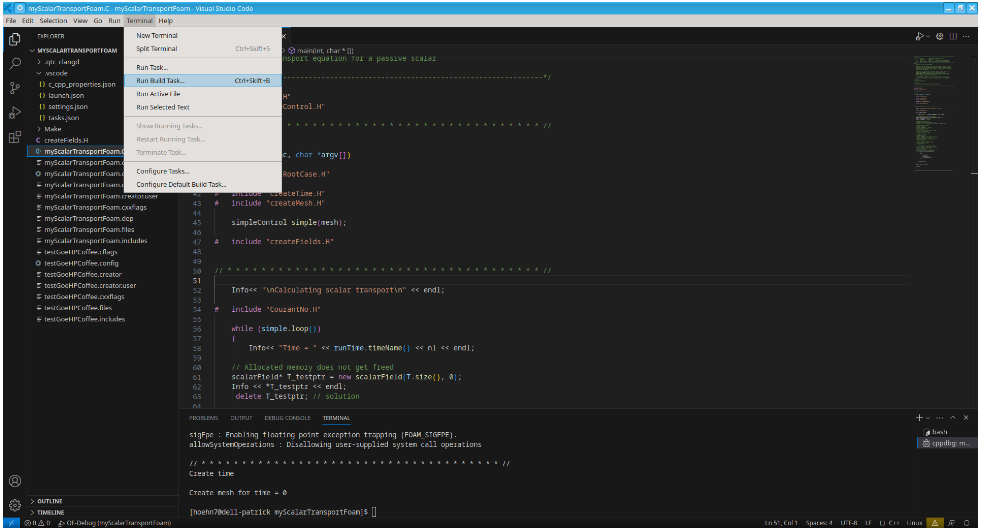
Python	IntelliSense (Pylance), Linting, De...	Microsoft · 92.5M · ★ 4 · Install
Jupyter	Jupyter notebook support, intera...	Microsoft · 68.7M · ★ 2.5 · Install
GitLens — Git supe...	Supercharge Git within VS Code ...	GitKraken · 24.8M · ★ 4 · Install
Vim	Vim emulation for Visual Studio C...	vscodevim · 5.2M · ★ 4 · Install
Hg	Integrated Mercurial source cont...	mrcrowl · 110K · ★ 5 · Install
SVN	Integrated Subversion source co...	Chris Johnston · 909K · ★ 4.5 · Install

```
    Solves a transport equation for a passive scalar

\*---------------------------------------------------------------------------*/

#include "fvCFD.H"
#include "simpleControl.H"

// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //


int main(int argc, char *argv[])
{
#   include "setRootCase.H"

#   include "createTime.H"
#   include "createMesh.H"

    simpleControl simple(mesh);

#   include "createFields.H"


// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

    Info<< "\nCalculating scalar transport\n" << endl;

#   include "CourantNo.H"

    while (simple.loop())
    {
        Info<< "Time = " << runTime.timeName() << nl << endl;

    // Allocated memory does not get freed
    scalarField* T_testptr = new scalarField(T.size(), 0);
    Info << *T_testptr << endl;
```

```
[hoehn7@dell-patrick myScalarTransportFoam]$
```

Do you want to install the recommended 'C/C++ Extension Pack' extension from Microsoft for C++ language?

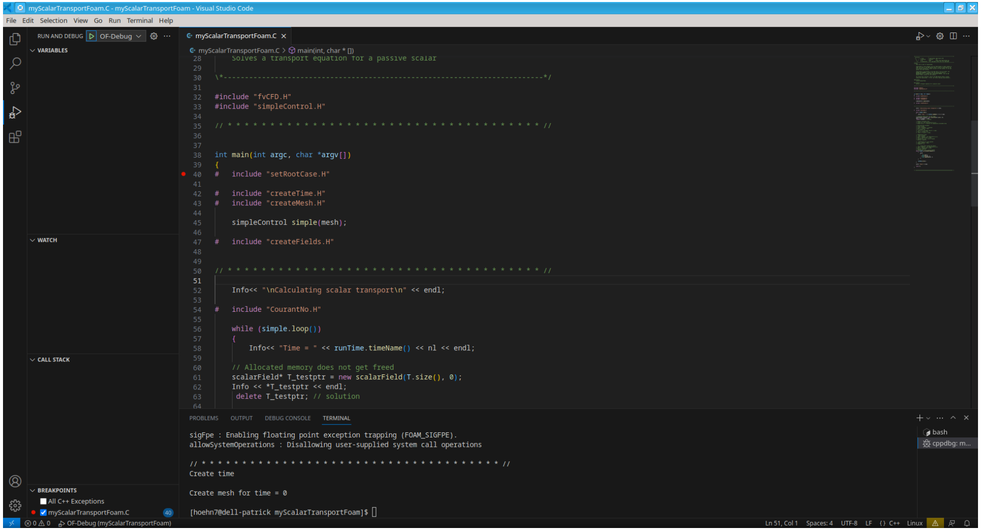
Hands-on - Visual Studio Code - Steps


myScalarTransportFoam.C - myScalarTransportFoam - Visual Studio Code
File Edit Selection View Go Run Terminal Help
New Terminal
Split Terminal Ctrl+Skift+5
Run Task...
Run Build Task... Ctrl+Skift+B
Run Active File
Run Selected Text
Show Running Tasks...
Restart Running Task...
Terminate Task...
Configure Tasks...
Configure Default Build Task...
EXPLORER
MYSCALARTRANSPORTFOAM
.qtc_clangd
.vscode
c_cpp_properties.json
launch.json
settings.json
tasks.json
Make
createFields.H
myScalarTransportFoam.C
myScalarTransportFoam.creator.user
myScalarTransportFoam.cxxflags
myScalarTransportFoam.dep
myScalarTransportFoam.files
myScalarTransportFoam.includes
testGoeHPCoffee.cflags
testGoeHPCoffee.config
testGoeHPCoffee.creator
testGoeHPCoffee.creator.user
testGoeHPCoffee.cxxflags
testGoeHPCoffee.files
testGoeHPCoffee.includes
OUTLINE
TIMELINE
main(int, char * [])
PROBLEMS OUTPUT DEBUG CONSOLE TERMINAL
bash
cppdbg: m...
OF-Debug (myScalarTransportFoam)
Ln 51, Col 1 Spaces: 4 UTF-8 LF C++ Linux

```
nsport equation for a passive scalar

---------------------------------------------------------------*/

H"
Control.H"

 * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //


c, char *argv[])

RootCase.H"

#   include "createTime.H"
#   include "createMesh.H"

    simpleControl simple(mesh);

#   include "createFields.H"


// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

    Info<< "\nCalculating scalar transport\n" << endl;

#   include "CourantNo.H"

    while (simple.loop())
    {
        Info<< "Time = " << runTime.timeName() << nl << endl;

    // Allocated memory does not get freed
    scalarField* T_testptr = new scalarField(T.size(), 0);
    Info << *T_testptr << endl;
     delete T_testptr; // solution
```

```
sigFpe : Enabling floating point exception trapping (FOAM_SIGFPE).
allowSystemOperations : Disallowing user-supplied system call operations

// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //
Create time

Create mesh for time = 0

[hoehn7@dell-patrick myScalarTransportFoam]$
```

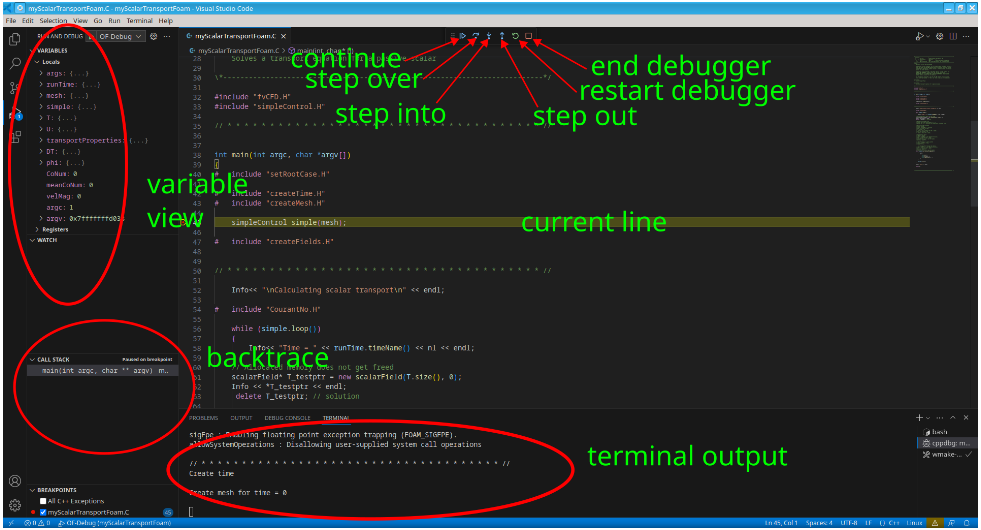
Hands-on - Visual Studio Code - Steps



```
    Solves a transport equation for a passive scalar

\*---------------------------------------------------------------------------*/

#include "fvCFD.H"
#include "simpleControl.H"

// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //


int main(int argc, char *argv[])
{
#   include "setRootCase.H"

#   include "createTime.H"
#   include "createMesh.H"

    simpleControl simple(mesh);

#   include "createFields.H"


// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //


    Info<< "\nCalculating scalar transport\n" << endl;

#   include "CourantNo.H"

    while (simple.loop())
    {
        Info<< "Time = " << runTime.timeName() << nl << endl;

    // Allocated memory does not get freed
    scalarField* T_testptr = new scalarField(T.size(), 0);
    Info << *T_testptr << endl;
     delete T_testptr; // solution
```

```
sigFpe : Enabling floating point exception trapping (FOAM_SIGFPE).
allowSystemOperations : Disallowing user-supplied system call operations

// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //
Create time

Create mesh for time = 0

[hoehn7@dell-patrick myScalarTransportFoam]$ 
```

Hands-on - Visual Studio Code - Steps



```
VARIABLES
  Locals
    > args: {...}
    > runTime: {...}
    > mesh: {...}
    > simple: {...}
    > T: {...}
    > U: {...}
    > transportProperties: {...}
    > DT: {...}
    > phi: {...}
      CoNum: 0
      meanCoNum: 0
      velMag: 0
      argc: 1
    > argv: 0x7fffffffd03
  > Registers
WATCH
```

```
CALL STACK
  main(int argc, char ** argv) m..
```

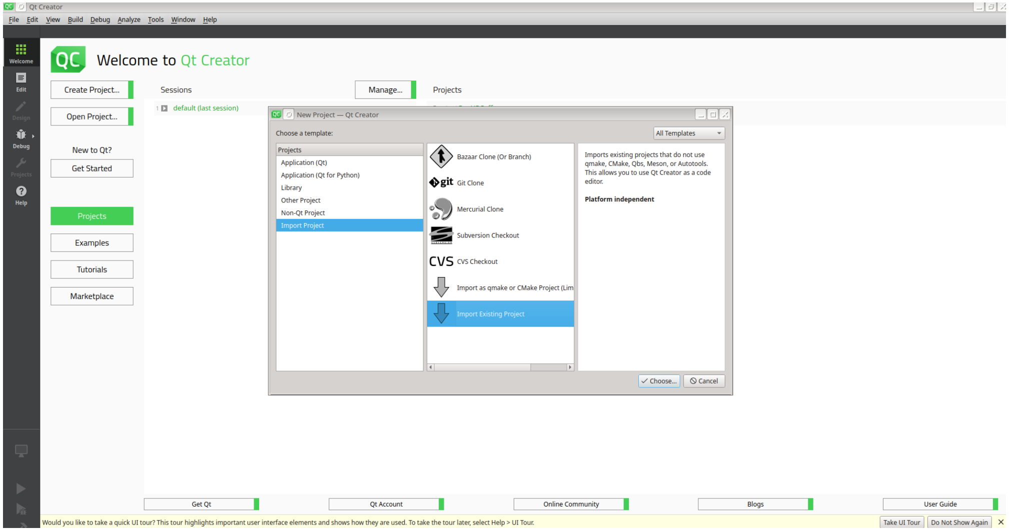
```cpp
    Solves a transport equation for a passive scalar

\*---------------------------------------------------------------------------*/

#include "fvCFD.H"
#include "simpleControl.H"

// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //


int main(int argc, char *argv[])
{
#   include "setRootCase.H"

#   include "createTime.H"
#   include "createMesh.H"

    simpleControl simple(mesh);

#   include "createFields.H"


// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

    Info<< "\nCalculating scalar transport\n" << endl;

#   include "CourantNo.H"

    while (simple.loop())
    {
        Info<< "Time = " << runTime.timeName() << nl << endl;

        // Allocated memory does not get freed
    scalarField* T_testptr = new scalarField(T.size(), 0);
    Info << *T_testptr << endl;
     delete T_testptr; // solution
```

```
sigFpe : Enabling floating point exception trapping (FOAM_SIGFPE).
allowSystemOperations : Disallowing user-supplied system call operations

// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //
Create time

Create mesh for time = 0

```

Hands-on - QtCreator - General instructions

- run ’fe41_debug’ in shell
- run ’blockMesh" in same terminal
- run ’find $FOAM_SRC -type d -iname "lnInclude"’ in same shell
- run ’qtcreator .’ from same shelll
- changing of files possible by double click on filename

Hands-on - QtCreator - Steps

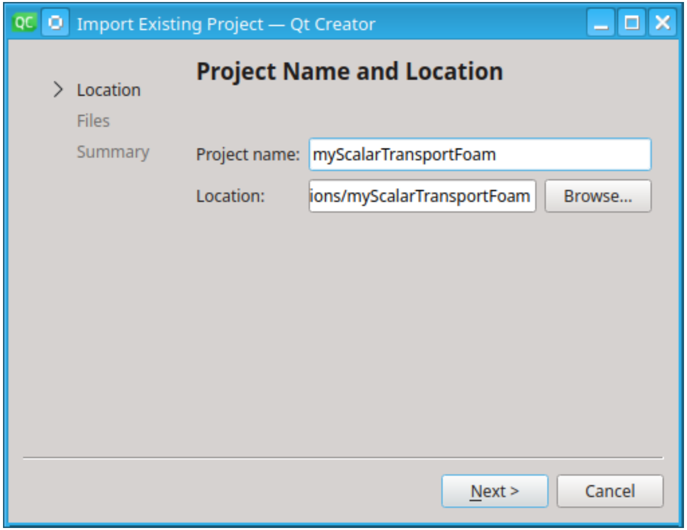


Imports existing projects that do not use qmake, CMake, Qbs, Meson, or Autotools. This allows you to use Qt Creator as a code editor.

Platform independent

Would you like to take a quick UI tour? This tour highlights important user interface elements and shows how they are used. To take the tour later, select Help > UI Tour.

Hands-on - QtCreator - Steps



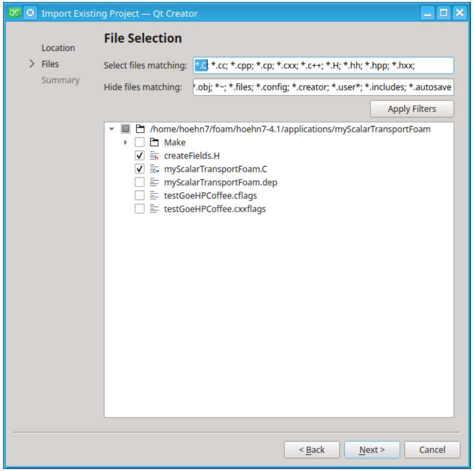
Project Name and Location

Project name: myScalarTransportFoam

Location: ions/myScalarTransportFoam

Hands-on - QtCreator - Steps

make sure that the filter includes "*.C" and "*.H" because Linux is case-sensitive


Import Existing Project — Qt Creator

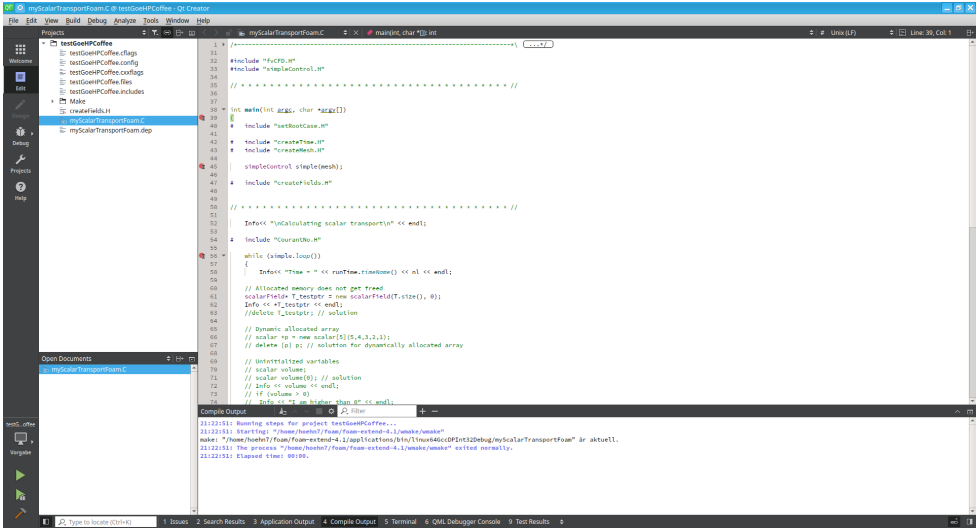
File Selection

Select files matching: *.C; *.cc; *.cpp; *.cp; *.cxx; *.c++; *.H; *.hh; *.hpp; *.hxx;

Hide files matching: *.obj; *~; *.files; *.config; *.creator; *.user*; *.includes; *.autosave

```
▾ ▣ /home/hoehn7/foam/hoehn7-4.1/applications/myScalarTransportFoam
   ▸ ☐ Make
     ☑ createFields.H
     ☑ myScalarTransportFoam.C
     ☐ myScalarTransportFoam.dep
     ☐ testGoeHPCoffee.cflags
     ☐ testGoeHPCoffee.cxxflags
```

Hands-on - QtCreator - Steps

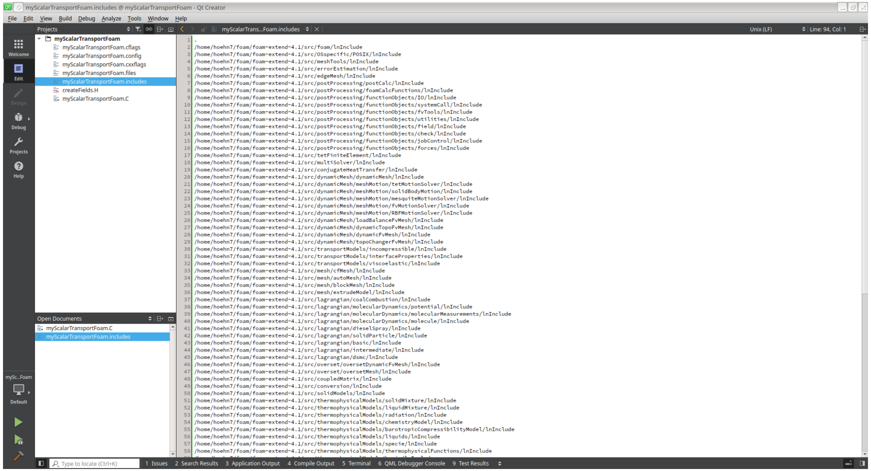


```
/*---------------------------------------------------------------------------*\ ...*/

#include "fvCFD.H"
#include "simpleControl.H"

// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //


int main(int argc, char *argv[])
{
#   include "setRootCase.H"

#   include "createTime.H"
#   include "createMesh.H"

    simpleControl simple(mesh);

#   include "createFields.H"


// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

    Info<< "\nCalculating scalar transport\n" << endl;

#   include "CourantNo.H"

    while (simple.loop())
    {
        Info<< "Time = " << runTime.timeName() << nl << endl;

    // Allocated memory does not get freed
    scalarField* T_testptr = new scalarField(T.size(), 0);
    Info << *T_testptr << endl;
    //delete T_testptr; // solution

    // Dynamic allocated array
    // scalar *p = new scalar[5](5,4,3,2,1);
    // delete [p] p; // solution for dynamically allocated array

    // Uninitialized variables
    // scalar volume;
    // scalar volume{0}; // solution
    // Info << volume << endl;
    // if (volume > 0)
    //  Info << "I am higher than 0" << endl;
```

```
21:22:51: Running steps for project testGoeHPCoffee...
21:22:51: Starting: "/home/hoehn7/foam/foam-extend-4.1/wmake/wmake"
make: "/home/hoehn7/foam/foam-extend-4.1/applications/bin/linux64GccDPInt32Debug/myScalarTransportFoam" är aktuell.
21:22:51: The process "/home/hoehn7/foam/foam-extend-4.1/wmake/wmake" exited normally.
21:22:51: Elapsed time: 00:00.
```

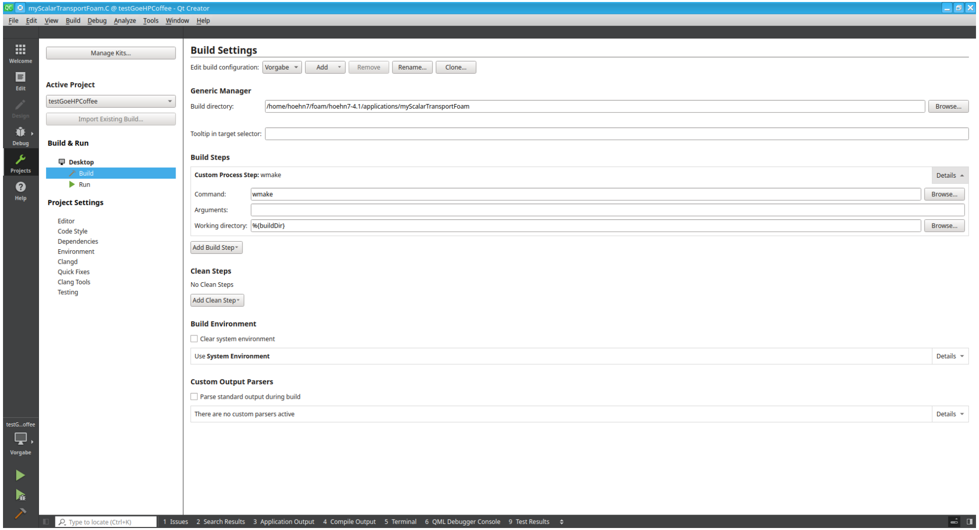
Hands-on - QtCreator - Steps

paste output from find command in file


myScalarTransportFoam.includes @ myScalarTransportFoam - Qt Creator
File Edit View Build Debug Analyze Tools Window Help
Projects
myScalarTransportFoam
myScalarTransportFoam.cflags
myScalarTransportFoam.config
myScalarTransportFoam.cxxflags
myScalarTransportFoam.files
myScalarTransportFoam.includes
createFields.H
myScalarTransportFoam.C
myScalarTrans...Foam.includes
Unix (LF)
Line: 94, Col: 1
Welcome
Edit
Design
Debug
Projects
Help
Open Documents
myScalarTransportFoam.C
myScalarTransportFoam.includes
mySc...Foam
Default
Type to locate (Ctrl+K)
1 Issues
2 Search Results
3 Application Output
4 Compile Output
5 Terminal
6 QML Debugger Console
9 Test Results

```
.
/home/hoehn7/foam/foam-extend-4.1/src/foam/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/OSspecific/POSIX/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/meshTools/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/errorEstimation/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/edgeMesh/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/postProcessing/postCalc/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/postProcessing/foamCalcFunctions/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/postProcessing/functionObjects/IO/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/postProcessing/functionObjects/systemCall/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/postProcessing/functionObjects/fvTools/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/postProcessing/functionObjects/utilities/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/postProcessing/functionObjects/field/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/postProcessing/functionObjects/check/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/postProcessing/functionObjects/jobControl/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/postProcessing/functionObjects/forces/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/tetFiniteElement/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/multiSolver/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/conjugateHeatTransfer/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/dynamicMesh/dynamicMesh/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/dynamicMesh/meshMotion/tetMotionSolver/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/dynamicMesh/meshMotion/solidBodyMotion/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/dynamicMesh/meshMotion/mesquiteMotionSolver/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/dynamicMesh/meshMotion/fvMotionSolver/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/dynamicMesh/meshMotion/RBFMotionSolver/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/dynamicMesh/loadBalanceFvMesh/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/dynamicMesh/dynamicTopoFvMesh/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/dynamicMesh/dynamicFvMesh/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/dynamicMesh/topoChangerFvMesh/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/transportModels/incompressible/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/transportModels/interfaceProperties/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/transportModels/viscoelastic/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/mesh/cfMesh/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/mesh/autoMesh/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/mesh/blockMesh/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/mesh/extrudeModel/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/lagrangian/coalCombustion/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/lagrangian/molecularDynamics/potential/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/lagrangian/molecularDynamics/molecularMeasurements/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/lagrangian/molecularDynamics/molecule/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/lagrangian/dieselSpray/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/lagrangian/solidParticle/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/lagrangian/basic/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/lagrangian/intermediate/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/lagrangian/dsmc/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/overset/oversetDynamicFvMesh/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/overset/oversetMesh/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/coupledMatrix/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/conversion/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/solidModels/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/thermophysicalModels/solidMixture/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/thermophysicalModels/liquidMixture/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/thermophysicalModels/radiation/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/thermophysicalModels/chemistryModel/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/thermophysicalModels/barotropicCompressibilityModel/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/thermophysicalModels/liquids/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/thermophysicalModels/specie/lnInclude
/home/hoehn7/foam/foam-extend-4.1/src/thermophysicalModels/thermophysicalFunctions/lnInclude
```

Hands-on - QtCreator - Steps


myScalarTransportFoam.C @ testGoeHPCoffee - Qt Creator
File Edit View Build Debug Analyze Tools Window Help
Welcome
Edit
Design
Debug
Projects
Help
Manage Kits...
Import Existing Build...
testG...offee
Vorgabe
Type to locate (Ctrl+K)
1 Issues 2 Search Results 3 Application Output 4 Compile Output 5 Terminal 6 QML Debugger Console 9 Test Results

Active Project

testGoeHPCoffee

Build & Run

- Desktop
  - Build
  - Run

Project Settings

- Editor
- Code Style
- Dependencies
- Environment
- Clangd
- Quick Fixes
- Clang Tools
- Testing

Build Settings

Edit build configuration: Vorgabe | Add | Remove | Rename... | Clone...

Generic Manager

Build directory: /home/hoehn7/foam/hoehn7-4.1/applications/myScalarTransportFoam

Tooltip in target selector:

Build Steps

Custom Process Step: wmake

Command: wmake

Arguments:

Working directory: %{buildDir}

Add Build Step

Clean Steps

No Clean Steps

Add Clean Step

Build Environment

☐ Clear system environment

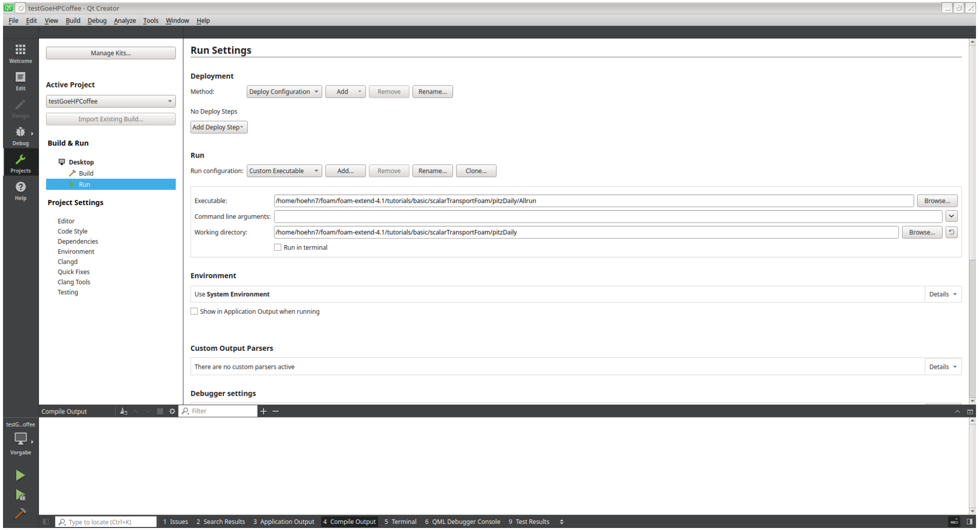
Use **System Environment**

Custom Output Parsers

☐ Parse standard output during build

There are no custom parsers active

Hands-on - QtCreator - Steps



testGoeHPCoffee - Qt Creator

Run Settings

Deployment

Method: Deploy Configuration

No Deploy Steps

Run

Run configuration: Custom Executable

Executable: /home/hoehn7/foam/foam-extend-4.1/tutorials/basic/scalarTransportFoam/pitzDaily/Allrun

Command line arguments:

Working directory: /home/hoehn7/foam/foam-extend-4.1/tutorials/basic/scalarTransportFoam/pitzDaily

Run in terminal

Environment

Use **System Environment**

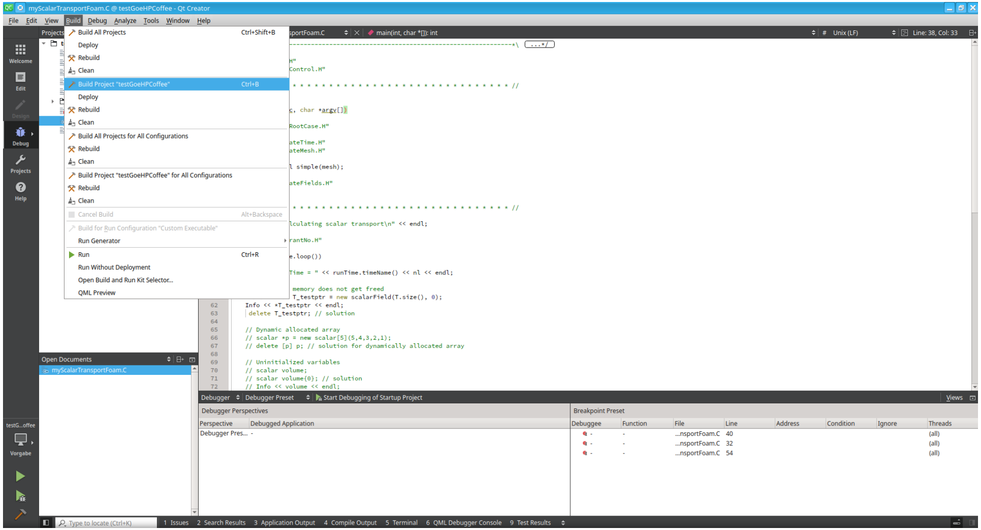
Show in Application Output when running

Custom Output Parsers

There are no custom parsers active

Debugger settings

Hands-on - QtCreator - Steps

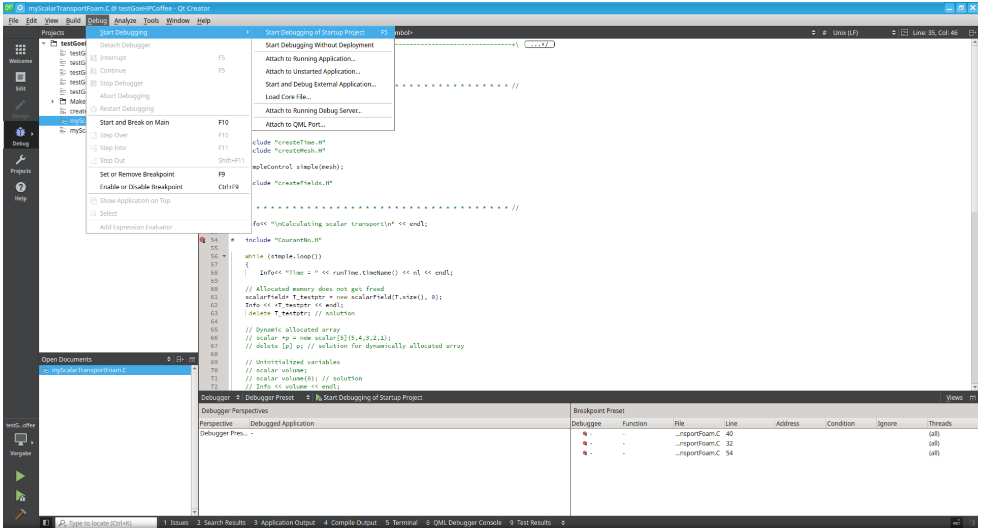


```
-------------------------------------------------------------*\  ...*/

H"
Control.H"

 * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //


c, char *argv[])

RootCase.H"

ateTime.H"
ateMesh.H"

l simple(mesh);

ateFields.H"


 * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

lculating scalar transport\n" << endl;

rantNo.H"

e.loop())

Time = " << runTime.timeName() << nl << endl;

 memory does not get freed
 T_testptr = new scalarField(T.size(), 0);
Info << *T_testptr << endl;
 delete T_testptr; // solution

// Dynamic allocated array
// scalar *p = new scalar[5](5,4,3,2,1);
// delete [p] p; // solution for dynamically allocated array

// Uninitialized variables
// scalar volume;
// scalar volume{0}; // solution
// Info << volume << endl;
```

Perspective	Debugged Application
Debugger Pres...	-

Debuggee	Function	File	Line	Address	Condition	Ignore	Threads
-	-	...nsportFoam.C	40				(all)
-	-	...nsportFoam.C	32				(all)
-	-	...nsportFoam.C	54				(all)

Hands-on - QtCreator - Steps

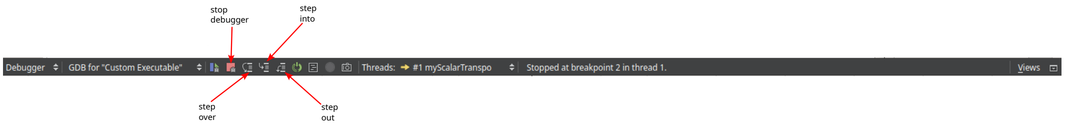

myScalarTransportFoam.C @ testGoeHPCoffee - Qt Creator
File Edit View Build Debug Analyze Tools Window Help
Projects
Welcome
Edit
Design
Debug
Projects
Help
Start Debugging
Detach Debugger
Interrupt F5
Continue F5
Stop Debugger
Abort Debugging
Restart Debugging
Start and Break on Main F10
Step Over F10
Step Into F11
Step Out Shift+F11
Set or Remove Breakpoint F9
Enable or Disable Breakpoint Ctrl+F9
Show Application on Top
Select
Add Expression Evaluator
Start Debugging of Startup Project F5
Start Debugging Without Deployment
Attach to Running Application...
Attach to Unstarted Application...
Start and Debug External Application...
Load Core File...
Attach to Running Debug Server...
Attach to QML Port...
Unix (LF)
Line: 35, Col: 46
Open Documents
myScalarTransportFoam.C
Debugger
Debugger Preset
Start Debugging of Startup Project
Views
Debugger Perspectives
Breakpoint Preset
testG...offee
Vorgabe
Type to locate (Ctrl+K)
1 Issues 2 Search Results 3 Application Output 4 Compile Output 5 Terminal 6 QML Debugger Console 9 Test Results

```
clude "createTime.H"
clude "createMesh.H"

mpleControl simple(mesh);

clude "createFields.H"


 * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

fo<< "\nCalculating scalar transport\n" << endl;

#   include "CourantNo.H"

    while (simple.loop())
    {
        Info<< "Time = " << runTime.timeName() << nl << endl;

    // Allocated memory does not get freed
    scalarField* T_testptr = new scalarField(T.size(), 0);
    Info << *T_testptr << endl;
     delete T_testptr; // solution

    // Dynamic allocated array
    // scalar *p = new scalar[5](5,4,3,2,1);
    // delete [p] p; // solution for dynamically allocated array

    // Uninitialized variables
    // scalar volume;
    // scalar volume{0}; // solution
    // Info << volume << endl;
```

Perspective	Debugged Application
Debugger Pres...	-

Debuggee	Function	File	Line	Address	Condition	Ignore	Threads
● -	-	...nsportFoam.C	40				(all)
● -	-	...nsportFoam.C	32				(all)
● -	-	...nsportFoam.C	54				(all)

Hands-on - QtCreator - Steps



Stopped at breakpoint 2 in thread 1.

References

- Training Session "OpenFOAM Code Debugging and Profiling" 18th OpenFOAM Workshop
- `https://www.tfd.chalmers.se/~hani/kurser/OS_CFD_2022/lectureNotes/24_debugging.pdf`
- `https://openfoamwiki.net`
- `https://cs.brown.edu/courses/cs033/docs/guides/gdb.pdf`
- R. Stallman, R. Pesch, and S. Shebs. Debugging with GDB: The GNU Source-Level debugger. GNU Press, Free Software Foundation Inc., 9th edition, 2002.

References

- https://wikis.ovgu.de/lss/doku.php?id=guide:qtcreater_for_openfoam
- https://github.com/Rvadrabade/Debugging-OpenFOAM-with-Visual-Studio-Code
- Damián, S. M., Giménez, J. M., Nigro, N. M. (2012). gdbOF: A debugging tool for OpenFOAM®. Advances in Engineering Software, 47(1), 17-23.
- https://github.com/FoamScience/foamUT
- https://users.ece.utexas.edu/~adnan/gdb-refcard.pdf
- https://info.gwdg.de/news/en/configuring-vscode-to-access-gwdgs-hpc-cluster

Thank you for your attention

Questions?