Basics of C++ in OpenFOAM

Basics of C++ in OpenFOAM

Prerequisites

- You have some programming experience.
- You have experience in working in Linux.

Learning outcomes

- You will learn the basic syntax in C++
- You will learn how to use classes to implement simple C++ codes, and how member functions are called in the top-level code.
- You will learn how to implement functions in the top-level code, understand the difference between declaration and definition, and see how that can be practically used.
- You will learn how OpenFOAM compilation relates to compilation of a simple C++ code.

Note that you will be asked to pack up your final cleaned-up directories and submit them for assessment of completion.

Basics of C++ in OpenFOAM

- To begin with: The aim of this part of the course is not to teach all of C++, but to give a short introduction that is useful when trying to understand the contents of OpenFOAM.
- After this introduction you should be able to *recognize* and make *minor modifications* to most C++ features in OpenFOAM.
- Some books:
  - *C++ direkt* by Jan Skansholm (ISBN 91-44-01463-5)
  - *C++ from the Beginning* by Jan Skansholm (probably similar)
  - *C++ how to Program* by Paul and Harvey Deitel
  - *Object Oriented Programming in C++* by Robert Lafore
- A link:
  - `https://www.geeksforgeeks.org/c-plus-plus/`

C++ basics – types

- Variables can contain data of different *types*, for instance:

  ```
  int myInteger;
  ```

  for a declaration of an integer variable named `myInteger`, or

  ```
  const int myConstantInteger = 10;
  ```

  for a declaration of an *constant* integer variable named `myConstantInteger` with value 10.

- Variables can be added, substracted, multiplied and divided as long as they have the same type, or if the types have definitions on how to convert between the types.

- In C++ it is possible to define special *types* (classes), and there are many types defined for you in OpenFOAM.

- User-defined types must have the required conversions defined. Some of the types in OpenFOAM can be used together in arithmetic expressions, but not all of them.

C++ basics – Namespace

- When using pieces of C++ code developed by different programmers there is a risk that the same name has been used for different things.
- By associating a declaration with a namespace, the declaration will only be visible if that namespace is used. The standard declarations are used by starting with:

  ```
  using namespace std;
  ```

- OpenFOAM declarations belong to namespace Foam, so in OpenFOAM we use:

  ```
  using namespace Foam;
  ```

  to make all declarations in namespace `Foam` visible.

- Explicit naming in OpenFOAM:

  ```
  Foam::function();
  ```

  where `function()` is a function defined in namespace `Foam`. This must be used if any other namespace containing a declaration of another `function()` is also visible.

C++ basics – input/output

- Input and output can be done using the standard library `iostream`, using:

```
cout << "Please type an integer!" << endl;
cin >> myInteger;
```

  where `<<` and `>>` are output and input operators, and `endl` is a manipulator that generates a new line (there are many other manipulators).

- In OpenFOAM a new output stream `Info` is however defined, and it is recommended to use that one instead since it takes care of write-outs for parallel simulations.

C++ basics, main function

- All C++ codes must have at least one function:

```
int main()
{
return 0;
}
```

  in this case, `main` takes no arguments, but it may (as in OpenFOAM applications).

- Code appearing after the `return` statement is not executed!!!

C++ basics, Example code

In file `basic1.C`:

```
#include <iostream>
using namespace std;
int main()
{
int myInteger;
const int constantInteger=5;
const float constantFloat=5.1;
cout << "Please type an integer!" << endl;
cin >> myInteger;
cout << myInteger << " + " << constantInteger << " = "
     << myInteger+constantInteger << endl;
cout << myInteger << " + " << constantFloat << " = "
     << myInteger+constantFloat << endl;
return 0;
}
```

Compile and run with:

```
g++ basic1.C -o basic1;
./basic1
```

C++ basics – operators

- `+`, `-`, `*` and `/` are operators that define how the operands should be used.
- Other standard operators are:

  `%` (integer division modulus)
  `++` (add 1)
  `--` (substract 1)
  `+=` (`i+=2` adds 2 to i)
  `-=` (`i-=2` subtracts 2 from i)
  `*=` (`i*=2` multiplies i by 2)
  `/=` (`i/=2` divides i by 2)

  etc. User-defined types should define its operators.

- Comparing operators: `<` `>` `<=` `>=` `==` `!=` Generates `bool` (boolean)
- Logical operators: `&&` `||` `!` (or, for some compilers: `and` `or` `not`). Generates `bool` (boolean)

C++ basics – functions

- Mathematic standard functions are available in standard libraries. They are thus not part of C++ itself.
- Standard library `cmath` contains trigonometric functions, logaritmic functions and square root. (use `#include cmath;` if you need them)
- Standard library `cstdlib` contains general functions, and some of them can be used for arithmetics. (use `#include cstdlib;` if you need them)

C++ basics – if, for and while-statements

- if-statements:

```
if (variable1 > variable2) {...CODE...} else {...CODE...}
```

- for-statements:

```
for ( init; condition; change ) {...CODE...}
```

- while-statements:

```
while (...expression...) {...CODE...}
```

`break;` breaks the execution of `while`

C++ basics, Example code

In file `basic2.C`:

```
#include <iostream>
#include <cmath>
using namespace std;
int main()
{
float myFloat;
cout << "Please type a float!" << endl;
cin >> myFloat;
cout << "sin(" << myFloat << ") = " << sin(myFloat) << endl;
if (myFloat < 5.5){cout << myFloat << " is less than 5.5" << endl;} else
                  {cout << myFloat << " is not less than 5.5" << endl;};
for ( int i=0; i<myFloat; i++ ) {cout << "For-looping: " << i << endl;}
int j=0;
while (j<myFloat) {cout << "While-looping: " << j << endl; j++;}
return 0;
} //Note conversion of myFloat to int in loops!
```

Compile and run with:

```
g++ basic2.C -o basic2; ./basic2
```

C++ basics – arrays

- Arrays:
  `double f[5];` (Note: components numbered from 0!)
  `f[3] = 2.75;` (Note: no index control!)
  `int a[6] = {2, 2, 2, 5, 5, 0};` (declaration and initialization)
  The arrays have strong limitations, but serve as a base for array **templates**

- Array **templates** (example `vector`. other: `list, deque`):
  ```
  #include <vector>
  using namespace std
  ```
  The type of the vector must be specified upon declaration:
  `vector<double> v2(3);` gives {0, 0, 0}
  `vector<double> v3(4, 1.5);` gives {1.5, 1.5, 1.5, 1.5}
  `vector<double> v4(v3);` Constructs `v4` as a copy of `v3` (copy-constructor)

- Array template operations: The template classes define member functions that can be used for those types, for instance: `size()`, `empty()`, `assign()`, `push_back()`, `pop_back()`, `front()`, `clear()`, `capacity()` etc.
  `v.assign(4, 1.0);` gives {1.0, 1.0, 1.0, 1.0}

C++ basics, Example code

In file `basic3.C`:

```
#include <iostream>
#include <vector>
using namespace std;
int main()
{
vector<double> v2(3);
vector<double> v3(4, 1.5);
vector<double> v4(v3);
cout << "v2: (" << v2[0] << "," << v2[1] << "," << v2[2] << ")" << endl;
cout << "v3: (" << v3[0] << "," << v3[1] << "," << v3[2] << "," << v3[3] << ")" << endl;
cout << "v4: (" << v4[0] << "," << v4[1] << "," << v4[2] << "," << v4[3] << ")" << endl;
cout << "v2.size(): " << v2.size() << endl;
return 0;
}
//LineToFixCopyPasteProblem-----------------------------------------------------
```

Compile and run with:

```
g++ basic3.C -o basic3; ./basic3
```

Note that the standard `vector` class is **not** implemented to be able to execute:

```
cout << "v2: " << v2 << endl;
```

Such functionality is available in OpenFOAM.

C++ basics – function implementation

- Example function named `average`

```
double average (double x1, double x2)
{
  int nvalues = 2;
  return (x1+x2)/nvalues;
}
```

  takes two arguments of type `double`, and returns type `double`. The variable `nvalues` is a local variable, and is only visible inside the function. Note that any code after the `return` statement will not be executed.

- A function doesn't have to take arguments, and it doesn't have to return anything (the output type is then specified as `void`).

- There may be several functions with the same names, as long as there is a difference in the arguments to the functions - the number of arguments or the types of the arguments.

- Functions must be *declared* before they are used.

C++ basics, Example code

In file `basic4.C`:

```
#include <iostream>
using namespace std;
double average (double x1, double x2)
{
  int nvalues = 2;
  return (x1+x2)/nvalues;
}
int main()
{
double d1=2.1;
double d2=3.7;
cout << "Average: " << average(d1,d2) << endl;
return 0;
}
```

Compile and run with:

```
g++ basic4.C -o basic4; ./basic4
```

C++ basics – declaration and definition of functions

- The function *declaration* must be done before it is used, but the function `definition` can be done after it is used. Example:

```
double average (double x1, double x2); //Declaration
main ()
{
  mv = average(value1, value2)
}
double average (double x1, double x2) //Definition
{
  return (x1+x2)/2;
}
```

  The argument *names* may be omitted in the `declaration`.

- Declarations are often included from include-files:

```
#include "file.h"
#include <standardfile>
```

- A good way to program C++ is to make files in pairs, one with the `declaration`, and one with the `definition`. This is done throughout OpenFOAM.

C++ basics, Example code

In file `basic5.C`:

```
#include <iostream>
#include "basic5.H"
using namespace std;
int main()
{
double d1=2.1;
double d2=3.7;
cout << "Average: " << average(d1,d2) << endl;
return 0;
}
double average (double x1, double x2)
{
  int nvalues = 2;
  return (x1+x2)/nvalues;
}
```

In file `basic5.H`:

```
double average (double, double);
```

Compile and run with: `g++ basic5.C -o basic5; ./basic5`

C++ basics – function parameters / arguments reference and default value

- If an argument variable should be changed inside a function, the type of the argument must be a reference, i.e.

  ```
  void change(double& x1)
  ```

  The reference parameter `x1` will now be a reference to the argument to the function instead of a local variable in the function. (standard arrays are always treated as reference parameters).

- Reference parameters can also be used to avoid copying of large fields when calling a function. To avoid changing the parameter in the function it can be declared as `const`, i.e.
  ```
  void checkWord(const string& s)
  ```
  This often applies for parameters of class-type, which can be large.

- Default values can be specified, and then the function may be called without that parameter, i.e.
  ```
  void checkWord(const string& s, int nmbr=1)
  ```

C++ basics, Example code

In file `basic6.C`:

```
#include <iostream>
using namespace std;

double average (double& x1, double& x2, int nvalues=2)
{
  x1 = 7.5;
  return (x1+x2)/nvalues;
}

int main()
{
double d1=2.1;
double d2=3.7;
cout << "Modified average: " << average(d1,d2) << endl;
cout << "Half modified average: " << average(d1,d2,4) << endl;
cout << "d1: " << d1 << ", d2: " << d2 << endl;
return 0;}
```

Compile and run with: `g++ basic6.C -o basic6; ./basic6`

C++ basics – Pointers

- Pointers point at a memory location (while a reference is referring to another variable, as shown before, i.e. they are different). Example (in `basic7.C`):

```
#include <iostream>
using namespace std;
int main()
{
double d1=2.1;
double d2=3.7;
double* d3; //d3 is a pointer, currently not pointing at anything
d3 = &d1; //Now d3 points at the memory location of d1
cout << "d1: " << d1 << endl;
cout << "d2: " << d2 << endl;
cout << "d3: " << d3 << endl;
cout << "*d3: " << *d3 << endl;
d3 = &d2; //Now d3 points at the memory location of d2
cout << "d3: " << d3 << endl;
cout << "*d3: " << *d3 << endl;
return 0;}
```

Compile and run with: `g++ basic7.C -o basic7; ./basic7`

Pointers for turbulence models

- Turbulence models are treated with the `turbulence` pointer in OpenFOAM.
  In file:`$FOAM_SOLVERS/incompressible/simpleFoam/createFields.H`:

```
autoPtr<incompressible::turbulenceModel> turbulence
(
    incompressible::turbulenceModel::New(U, phi, laminarTransport)
);
```

  In file `$FOAM_SOLVERS/incompressible/simpleFoam/simpleFoam.C`:

```
turbulence->correct();
```

C++ basics – Types

- *Types* define what values a variable may obtain, and what operations may be made on the variable.
- Pre-defined C++ types are:

```
signed char                 unsigned int
short int                   unsigned long int
int                         float
unsigned char               double
unsigned short int          long double
```

- User defined types can be defined in *classes*. OpenFOAM provides many types/classes that are useful for solving partial differential equations.
- OpenFOAM classes are used by including the class declarations in the header of the code, and linking to the corresponding compiled OpenFOAM library at compilation.
- The path to included files that are in another path than the current directory must be specified by `-I`
- The path to libraries that are linked to is specified with `-L`

C++ basics, Example code

In file `basic8.C`:

```
#include <iostream>      //Just for cout
using namespace std;     //Just for cout
#include "tensor.H"      //From OpenFOAM
#include "symmTensor.H" //From OpenFOAM
using namespace Foam;    //From OpenFOAM
int main()
{    tensor t1(1, 2, 3, 4, 5, 6, 7, 8, 9); //From OpenFOAM
     cout << "t1[0]: " << t1[0] << endl;
     symmTensor st1(1, 2, 3, 4, 5, 6);      //From OpenFOAM
     cout << "st1[5]: " << st1[5] << endl;
     return 0;
}
```

Make sure that you have sourced OpenFOAM in your terminal window.
Compile and run with (some trial-and-error, looking at output from wmake for test/tensor):

```
g++ -std=c++0x basic8.C -DWM_DP -DWM_LABEL_SIZE=32 -I$FOAM_SRC/OpenFOAM/lnInclude \
    -L$WM_PROJECT_DIR/lib/$WM_OPTIONS/libOpenFOAM.so -o basic8; ./basic8
```

Here, `-DWM_DP` is for double precision floats and `-DWM_LABEL_SIZE=32` is for 32 bit `int`.
We include header files (`declarations`) from `$FOAM_SRC/OpenFOAM/lnInclude`
We link to library (`definitions`) `$WM_PROJECT_DIR/lib/$WM_OPTIONS/libOpenFOAM.so`