

Challenge 5

OpenFoam development
Om Mihani

09/05/2022

Code:

```
#include "fvCFD.H"
int main( int argc, char *argv[] )
{
    Foam::argList::noBanner();
    Foam::argList args(argc, argv);
    ...
}
```

Target:

To create a .cpp file that prints out the value of temperature stored in a dictionary file thermalProperties.

Notes on the solution:

First of all, we need to make a runtime object.

Motivation: OpenFOAM-6/src/OpenFOAM/include/createTime.H

Next, we need to create a mesh object.

Motivation: OpenFOAM-6/src/OpenFOAM/include/createMesh.H

Finally, we create our own IObject T

Note that here we just need to print the values already present in a field. So we do not need to alter the code a lot. All the code is being used from the lectures.

Difficulties faced:

Locating the files

Still figuring out why we didn't give the name or mention that T is a volScalarField anywhere in the code.

Github link for the code: [Assignment 5](#)