

How to implement your own turbulence model (1/3)

- The implementations of the turbulence models are located in `$FOAM_SRC/turbulenceModels`
- Copy the source of the turbulence model that is most similar to what you want to do. In this case we will make our own copy of the `kOmegaSST` turbulence model:

```
cd $WM_PROJECT_DIR
cp -r --parents src/turbulenceModels/incompressible/RAS/kOmegaSST \
  $WM_PROJECT_USER_DIR
cd $WM_PROJECT_USER_DIR/src/turbulenceModels/incompressible/RAS
mv kOmegaSST mykOmegaSST
cd mykOmegaSST
```

How to implement your own turbulence model (2/3)

- We also need a Make/files and a Make/options

- Create a Make directory:

```
mkdir Make
```

- Create Make/files and add (c.f. original Make/files):

```
mykOmegaSST.C
```

```
LIB = $(FOAM_USER_LIBBIN)/libmyIncompressibleRASModels
```

(we are only *adding* mykOmegaSST)

- Create Make/options and add (c.f. original Make/options):

```
EXE_INC = \  
-I$(LIB_SRC)/turbulenceModels \  
-I$(LIB_SRC)/transportModels \  
-I$(LIB_SRC)/finiteVolume/lnInclude \  
-I$(LIB_SRC)/meshTools/lnInclude \  
-I$(LIB_SRC)/turbulenceModels/incompressible/RAS/lnInclude  
LIB_LIBS =
```

(the last -I is needed since mykOmegaSST uses include-files in the original directory)

How to implement your own turbulence model (3/3)

- We need to modify the file names of our new turbulence model. Note that the Linux command `rename` might work differently in different systems two examples are given here (See `man rename`. You can also use `mv` for each file/directory):

```
rename kOmegaSST mykOmegaSST * #Works in StuDat
rename 's/kOmegaSST/mykOmegaSST/' *
```

- In `mykOmegaSST.C` and `mykOmegaSST.H`, change all occurrences of `kOmegaSST` to `mykOmegaSST` so that we have a new class name:

```
sed -i s/kOmegaSST/mykOmegaSST/g mykOmegaSST.C
sed -i s/kOmegaSST/mykOmegaSST/g mykOmegaSST.H
```

- Introduce a small modification so that we can see if we use our new model. Add within the curly brackets of the constructor in `mykOmegaSST.C`:

```
Info << "Defining my own kOmegaSST model" << endl;
```

- Compile using:

```
wmake libso
```

which will build a dynamic library.

Test on the simpleFoam/pitzDaily tutorial

We will use our turbulence model on the simpleFoam/pitzDaily tutorial:

```
run
cp -r $FOAM_TUTORIALS/incompressible/simpleFoam/pitzDaily .
cd pitzDaily
blockMesh
```

How to use your own turbulence model

- Tell OpenFOAM to use your new library by adding a line to controlDict:

```
libs ("libmyIncompressibleRASModels.so");
```

- You choose turbulence model in the constant/RASProperties dictionary:

```
RASModel mykOmegaSST;
```

- You also need to generate a 0/omega file, and update fvSchemes and fvSolution

```
cp 0/epsilon 0/omega
sed -i s/epsilon/omega/g 0/omega
sed -i s/"0 2 -3 0 0 0 0"/"0 0 -1 0 0 0 0"/g 0/omega
sed -i s/14.855/440.15/g 0/omega
sed -i s/epsilon/omega/g system/fvSchemes
sed -i s/epsilon/omega/g system/fvSolution
```

(here $\nu_t = \frac{k}{\omega} = C_\mu \frac{k^2}{\varepsilon}$ is kept the same in both cases)

- Now you can run the simpleFoam/pitzDaily tutorial with your new turbulence model. Try both kOmegaSST and mykOmegaSST and look for your write-statement in the log file.
- Simply add appropriate source terms to implement a variant of kOmegaSST....

A note on new turbulence models

- The RAS turbulence models in OpenFOAM are sub-classes to the virtual class `RASModel`.
- You are only allowed to use the same member function definitions as in the `RASModel` class. If you need other member functions you will have to add those to the `RASModel` class, which requires that you copy and modify all of `$FOAM_SRC/turbulenceModels/incompressible/RAS`. You can recognize where the top-level of a class is located by locating the Make-directory.

We will now have a look at the implementation of the `kOmegaSST` model.

$k - \omega$ SST in OpenFOAM-1.6 (almost identical in 1.7.x)

From \$FOAM_SRC/turbulenceModels/incompressible/RAS/kOmegaSST/kOmegaSST.H:

- Menter, F., Esch, T.
"Elements of Industrial Heat Transfer Prediction"
16th Brazilian Congress of Mechanical Engineering (COBEM),
Nov. 2001
- Note that this implementation is written in terms of alpha diffusion coefficients rather than the more traditional sigma (alpha = 1/sigma) so that the blending can be applied to all coefficients in a consistent manner. The paper suggests that sigma is blended but this would not be consistent with the blending of the k-epsilon and k-omega models.
- Also note that the error in the last term of equation (2) relating to sigma has been corrected.
- Wall-functions are applied in this implementation by using equations (14) to specify the near-wall omega as appropriate.
- The blending functions (15) and (16) are not currently used because of the uncertainty in their origin, range of applicability and that as y^+ becomes sufficiently small blending u_τ in this manner clearly becomes nonsense.

$k - \omega$ SST in OpenFOAM-1.6, ν_t **Source code:**

```
$FOAM_SRC/turbulenceModels/incompressible/RAS/kOmegaSST
```

Kinematic eddy viscosity:

$$\nu_t = \frac{a_1 k}{\max(a_1 \omega, SF_2)}$$

kOmegaSST.C:

```
nut_ =
    a1_*k_/max(a1_*(omega_ + omegaSmall_), F2()*mag(symm(fvc::grad(U_))));
```

In kOmegaSST.C:

```
a1_(dimensioned<scalar>::lookupOrAddToDict("a1",coeffDict_,0.31))
```

In kOmegaSST.C ($S = \sqrt{S2}$):

```
volScalarField S2 = magSqr(symm(fvc::grad(U_)));
```

i.e. $S^2 = \left| \frac{1}{2}(\partial_j u_i + \partial_i u_j) \right|^2$ and $S = \sqrt{S^2} = \left| \frac{1}{2}(\partial_j u_i + \partial_i u_j) \right|$

F2() is a blending function, which is described on the next slide

$k - \omega$ SST in OpenFOAM-1.6, F2()

F2() is a blending function:

$$F_2 = \tanh \left[\left[\min \left(\max \left(\frac{2\sqrt{k}}{\beta^*\omega y}, \frac{500\nu}{y^2\omega} \right), 100 \right) \right]^2 \right]$$

In kOmegaSST.C:

```
tmp<volScalarField> kOmegaSST::F2() const
{
    volScalarField arg2 = min
    (
        max
        (
            (scalar(2)/betaStar_)*sqrt(k_)/(omega_*y_),
            scalar(500)*nu()/ (sqr(y_)*omega_)
        ),
        scalar(100)
    );

    return tanh(sqr(arg2));
}
```

$k - \omega$ SST in OpenFOAM-1.6, Turbulence kinetic energy eq.

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = P_k - \beta^* k \omega + \frac{\partial}{\partial x_j} \left[(\nu + \sigma_k \nu_T) \frac{\partial k}{\partial x_j} \right], \quad P_k = \min(G, 10\beta^* k \omega), \quad G = \nu_t \frac{\partial U_i}{\partial x_j} \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right)$$

In kOmegaSST.C:

```
tmp<fvScalarMatrix> kEqn
(
    fvm::ddt(k_)
  + fvm::div(phi_, k_)
  - fvm::Sp(fvc::div(phi_), k_)
  - fvm::laplacian(DkEff(F1), k_)
  ==
    min(G, c1_*betaStar_*k_*omega_)
  - fvm::Sp(betaStar_*omega_, k_)
);
```

The effective diffusivity for k , ($DkEff(F1)$), is described on a later slide.

$F1$ is obtained from $F1()$, which is a blending function for σ_k , and is described on the next slide,

where $CD_{k\omega} = 2\sigma_{\omega^2} \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i}$

```
volScalarField CDkOmega =
```

```
(2*alphaOmega2_)*(fvc::grad(k_) & fvc::grad(omega_))/omega_;
```

$F_1()$ is a blending function, $k\Omega$ SST.C (compressed here):

$$F_1 = \tanh \left\{ \left\{ \min \left(\min \left[\max \left(\frac{\sqrt{k}}{\beta^* \omega y}, \frac{500\nu}{y^2 \omega} \right), \frac{4\sigma_{\omega_2} k}{CD_{kw}^+ y^2} \right], 10 \right) \right\}^4 \right\}$$

```
tmp<volScalarField> kOmegaSST::F1(const volScalarField& CDkOmega) const
{
    volScalarField CDkOmegaPlus = max
    (
        CDkOmega,
        dimensionedScalar("1.0e-10", dimless/sqr(dimTime), 1.0e-10)
    );
    volScalarField arg1 = min
    (
        min
        (
            max
            (
                (scalar(1)/betaStar_)*sqrt(k_)/(omega_*y_),
                scalar(500)*nu()/ (sqr(y_)*omega_)
            ),
            (4*alphaOmega2_)*k_/ (CDkOmegaPlus*sqr(y_))
        ),
        scalar(10)
    );
    return tanh(pow4(arg1));
}
```

$F_1 = 0$ in the freestream ($k - \varepsilon$ model) and $F_1 = 1$ in the boundary layer ($k - \omega$ model)

$k - \omega$ SST in OpenFOAM-1.6, Effective diffusivity for k

The effective diffusivity for k , (DkEff (F1)), kOmegaSST.H:

```
tmp<volScalarField> DkEff(const volScalarField& F1) const
{
    return tmp<volScalarField>
    (
        new volScalarField("DkEff", alphaK(F1)*nut_ + nu())
    );
}
```

Blend alphaK1 and alphaK2 using blend function F1, kOmegaSST.H:

```
tmp<volScalarField> alphaK
(
    const volScalarField& F1
) const
{
    return blend(F1, alphaK1_, alphaK2_);
}
```

In kOmegaSST.C:

```
alphaK1_(dimensioned<scalar>::lookupOrAddToDict("alphaK1",coeffDict_,0.85034))
alphaK2_(dimensioned<scalar>::lookupOrAddToDict("alphaK2",coeffDict_,1.0))
```

$k - \omega$ SST in OpenFOAM-1.6, Specific dissipation rate eq.

$$\frac{\partial \omega}{\partial t} + U_j \frac{\partial \omega}{\partial x_j} = \alpha S^2 - \beta \omega^2 + \frac{\partial}{\partial x_j} \left[(\nu + \sigma_\omega \nu_T) \frac{\partial \omega}{\partial x_j} \right] + 2(1 - F_1) \sigma_\omega \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i}$$

In kOmegaSST.C:

```
tmp<fvScalarMatrix> omegaEqn
(
    fvm::ddt(omega_)
  + fvm::div(phi_, omega_)
  - fvm::Sp(fvc::div(phi_), omega_)
  - fvm::laplacian(DomegaEff(F1), omega_)
==
    gamma(F1)*2*S2
  - fvm::Sp(beta(F1)*omega_, omega_)
  - fvm::SuSp
    (
        (F1 - scalar(1))*CDkOmega/omega_,
        omega_
    )
);
```

Modify our mykOmegaSST model into kOmegaSSTF

- Gyllenram, W. and Nilsson, H., *Design and Validation of a Scale-Adaptive Filtering Technique for LRN Turbulence Modeling of Unsteady Flow*, JFE, May 2008, Vol.130.
- Upper limit (Δ_f or $1t$ below) to the modelled length scale (L_t or L_t below), applied to ν_t :

$$\Delta_f = \alpha \max \left\{ \left| \vec{U} \right| \delta t, \Delta^{1/3} \right\}, \quad \alpha = 3 \quad (\alpha > 1), \quad \hat{\nu}_t = \left(\frac{\min(L_t, \Delta_f)}{L_t} \right)^{4/3} \frac{k}{\omega}$$

kOmegaSST:

```
// Re-calculate viscosity
nut_ = a1_*k_/max(a1_*omega_, F2()*sqrt(S2));
```

kOmegaSSTF: (See the Turbulence WG Wiki. Please help me implement this more general!)

```
// Compute Filter
scalar alph = 3.0; //Should be in a dictionary
scalarField Lt = sqrt(k_)/(betaStar_*omega_);
scalarField lt = alph*Foam::max(Foam::pow(mesh_.V().field(), 1.0/3.0),
    (mag(U_)*runTime_.deltaT())->internalField());
```

```
// Re-calculate viscosity
nut_.internalField() = Foam::min(Foam::pow(lt/Lt, 4.0/3.0), 1.0)*
    (a1_*k_/max(a1_*omega_, F2()*sqrt(S2)))->internalField();
```

Modify our mykOmegaSST model into kOmegaSSTF

```
cd $WM_PROJECT_USER_DIR/src/turbulenceModels/incompressible/RAS/mykOmegaSST/
```

Find in mykOmegaSST.C the lines saying:

```
// Re-calculate viscosity
nut_ = a1_*k_/max(a1_*omega_, F2()*sqrt(S2));
```

Exchange those lines with:

```
// Compute Filter
scalar alph = 3.0; //Should be in a dictionary
scalarField Lt = sqrt(k_)/(betaStar_*omega_);
scalarField lt = alph*Foam::max(Foam::pow(mesh_.V().field(), 1.0/3.0),
                               (mag(U_)*runTime_.deltaT())->internalField());
```

```
// Re-calculate viscosity
nut_.internalField() = Foam::min(Foam::pow(lt/Lt, 4.0/3.0), 1.0)*
    (a1_*k_/max(a1_*omega_, F2()*sqrt(S2)))->internalField();
```

Compile with `wmake libso`

Modify the pitzDaily case for pimpleFoam

Make sure that you are in the `pitzDaily` case, and delete the previous results:

```
run ; cd pitzDaily ; rm -r [1-9]*
```

Modify the files in `system`, for use with `pimpleFoam`:

```
cp $FOAM_TUTORIALS/incompressible/pimpleFoam/TJunction/system/{fvSolution,fvSchemes} system
```

```
sed -i s/epsilon/omega/g system/fvSchemes
```

```
sed -i s/epsilon/omega/g system/fvSolution
```

```
sed -i s/simpleFoam/pimpleFoam/g system/controlDict
```

```
sed -i s/1000/0.3/g system/controlDict
```

```
sed -i s/"1;"/"0.0001;"/g system/controlDict
```

```
sed -i s/uncompressed/compressed/g system/controlDict
```

Add to `system/controlDict`:

```
adjustTimeStep no;
```

```
maxCo          5;
```

`pimpleFoam` needs one more dictionary:

```
cp $FOAM_TUTORIALS/incompressible/pimpleFoam/TJunction/constant/turbulenceProperties constant
```

We can re-use the same `0` directory that we modified before.

Make sure that you still specify `mykOmegaSST` in `constant/RASproperties`

Run the case with `pimpleFoam -noFunctionObjects` and make a nice movie of the results.

kOmegaSSTF

The kOmegaSSTF turbulence model is available for OpenFOAM-1.5 at OpenFOAM-extend:

http://openfoam-extend.svn.sourceforge.net/viewvc/openfoam-extend/trunk/Breeder_1.5/OSIG/Turbulence/src/turbulenceModels/RAS/incompressible/kOmegaSSTF

There is a pitzDaily tutorial for the turbFoam solver (no longer in OpenFOAM-1.6 and newer versions), and a utility for viewing the filter function.

It is also used in the Dellenback Abrupt Expansion case-study, which is described in the Turbulence Working Group Wiki:

http://openfoamwiki.net/index.php/Sig_Turbulence/_Dellenback_Abrupt_Expansion