Cite as: Muye, G.: Improve sheet cavitation inception prediction by taking laminar separation into consideration. In Proceedings of CFD with OpenSource Software, 2018, Edited by Nilsson. H.,
http://dx.doi.org/10.17196/OS_CFD#YEAR_2018

CFD WITH OPENSOURCE SOFTWARE

A COURSE AT CHALMERS UNIVERSITY OF TECHNOLOGY
TAUGHT BY HÅKAN NILSSON

# Improve sheet cavitation inception prediction by taking laminar separation into consideration

Developed for OpenFOAM-v1806+

*Author:*
Muye GE
Chalmers University of Technology
muye.ge@chalmers.se

*Peer reviewed by:*
EBRAHIM GHAHRAMANI
FEDERICO ZABALETA

Licensed under CC-BY-NC-SA, https://creativecommons.org/licenses/

Disclaimer: This is a student project work, done as part of a course where OpenFOAM and some other OpenSource software are introduced to the students. Any reader should be aware that it might not be free of errors. Still, it might be useful for someone who would like learn some details similar to the ones presented in the report and in the accompanying files. The material has gone through a review process. The role of the reviewer is to go through the tutorial and make sure that it works, that it is possible to follow, and to some extent correct the writing. The reviewer has no responsibility for the contents.

December 21, 2018

# Learning outcomes

The reader will learn:

- how the interPhaseChangeFoam solver works
- how to use the $\gamma - Re_\theta$ transition sensitive turbulence model
- the theory of laminar separation and its relationship with sheet cavitation inception
- the theory background of the Schenerr-Sauer mass transfer model
- the implementation of the combination of laminar separation indicator and cavitation vaporization

# Prerequisties

The reader is expected to have the following knowledge to follow this report

- Fundamentals of Volume of Fluid approach
- Basic knowledge of Computational Fluid Dynamics and Finite Volume Method
- Basic knowledge of running cases using OpenFOAM

# Contents


**1 Theoretical Background** **4**
- 1.1 Introduction . . . 4
- 1.2 Govering Equations . . . 4
  - 1.2.1 VoF treatment in OpenFOAM . . . 4
  - 1.2.2 Schnerr-Sauer cavitation model . . . 5
  - 1.2.3 Laminar separation in $\gamma - Re_\theta$ model . . . 6
- 1.3 Combination of laminar separation and cavitation . . . 7

**2 Implementation** **8**
- 2.1 Modification of $\alpha$ transportation . . . 9
- 2.2 Laminar separation indicator . . . 12
- 2.3 Schnerr Sauer cavitation model . . . 14
- 2.4 Modification of cavitation model . . . 15

**3 Test Case** **17**
- 3.1 Case description . . . 17
- 3.2 Mesh and boundary conditions . . . 18
- 3.3 Simulation results . . . 21
  - 3.3.1 Non-cavitating condition with Re = 300,000 . . . 21
  - 3.3.2 Non-cavitating condition with Re = 1,000,000 . . . 21
  - 3.3.3 Cavitating condition with Re = 1,000,000 . . . 22

# Chapter 1

# Theoretical Background

## 1.1 Introduction

When the local pressure in the liquid falls below a certain threshold, the liquid will change its phase and vaporize. This phenomena is called cavitation. Cavitation usually happens on marine propeller blades, impellers , engine nozzles and hydrofoils under certain conditions.

Due to the complexicity of the cavitation physics, cavitation mass transfer models are usually used in the FVM-based simulations. In most cavitation mass transfer models, the pressure is used as the only or major vaporization indicator. The cavitation may develop remains same position relative with the geometry profile, which is called sheet cavitation and it is one of the common type of cavitation types occuring on lifting surfaces. Though the behind mechanisms are still not fully understood, many researches revealed that the detachment of the sheet cavity usually occurs slightly downstream of the laminar separation point. The lack of sensitivity to other flow features like laminar separation would lead to massively over-prediction of cavitaiton pattern([Arakeri(1975)] [Casey(1974)] [Franc and Michel(1985)] [Wang et al.(2001)Wang, Senocak, Shyy, Ikohagi, and Cao] ).

The $\gamma - Re_\theta$ is a local correlation based model in RANS for the prediction of laminar to turbulence transition phenomenon. The correlation based method accounts for transition due to free stream turbulence, pressure gradient and separation.

In the present study, the NACA 16012 hydrofoil will be studied [Franc and Michel(1985)]. The experiments were performed by J.P.Franc and J.M.Michel. From the numerical side, firstly the simulation will be performed in the common way to illustrate the cavitation over-prediction problem, i.e. using $k\omega - SST$ turbulence model, VoF (Volume of Fluid) approach with Schenerr-Sauer mass transfer model for cavitaiton conditions; secondly the $\gamma - Re_\theta$ will be used instead and the vaporization factor will be adopted according to laminar separation.

## 1.2 Govering Equations

### 1.2.1 VoF treatment in OpenFOAM

Based on the VoF approach, the transport equation of $\alpha$ could be written as

$$\frac{\partial \alpha}{\partial t} + \bigtriangledown \cdot \alpha \mathbf{u} = 0, \tag{1.1}$$

and the properties of the mixture fluid could be calculated based on the homogeneous flow assumption,

$$\rho_m = \alpha_1 \rho_1 + (1 - \alpha_1)\rho_2, \quad \mu_m = \alpha_1 \mu_1 + (1 - \alpha_1)\mu_2, \tag{1.2}$$

in which $\rho$ denotes the density, $\mu$ denotes the dynamic viscosity and subscriptions $1, 2, m$ denote phase 1, phase 2 and mixture phase.

To matain boundness and accuracy, a Flux Corrected Transport (FCT) based method was used to solve this transport equation, which is MUlti-dimensional Limiter for Explicit Solution (MULES). An updated version, CMULES was released after the original MULES solver, which firstly solves Equation. 1.3 using a diffusive implicit approach (implicit Euler time stepping and upwind convection) and then solve Equation. 1.4 explicitly with higher order schemes,

$$\frac{\alpha_{i(upwind)}^{n+1} - \alpha_i^n}{\Delta t} V + \phi_{upwind}^n = 0, \tag{1.3}$$

$$\frac{\alpha_i^{n+1} - \alpha_{i(upwind)}^{n+1}}{\Delta t} V + \lambda(\phi_{highOrder}^n - \phi_{upwind}^n) = 0, \tag{1.4}$$

where $\phi$ represents the summation of the face normal flux of $\alpha$ in a cell transported by the velocity.

### 1.2.2 Schnerr-Sauer cavitation model

In reality, rather than a perfect purified medium, water contains many other other facts, and the small bubbles are one of the major source to break the water medium under tension. The small vapor bubbles are also refereed to nuclei. Its distribution, number and size could be determined experimentally.

As mentioned earlier, the FVM-based approaches rely on cavitation models to simulate the process. There are many cavitation mass transfer models available, and the Schnerr-Sauer cavitation model is one of the widely used models. In the Schenerr Sauer cavitation model, the distribution of nuclei are assumed to be spherical and uniformly distributed in the water medium.

For uniformly distributed nuclei, the volume of nuclei with diameter $d$ and number $n$ in one unit could be estimated as

$$V_{nuc} = \frac{1}{6} n\pi d^3, \tag{1.5}$$

thus the volume fraction of nuclei could be expressed as

$$\alpha_{nuc} = \frac{V_{nuc}}{1 + V_{nuc}} = \frac{\frac{1}{6} n\pi d^3}{1 + \frac{1}{6} n\pi d^3}. \tag{1.6}$$

The bubble growth rates was estimated using a simplified Rayleigh relation,

$$\frac{dR}{dt} = \sqrt{\frac{2}{3} \frac{p_{sat} - p_\infty}{\rho_l}}, \tag{1.7}$$

and the vapor production rate could be calculated from the following equation,

$$\frac{d\alpha}{dt} = (1 - \alpha) \frac{4\pi n R^2 \frac{dR}{dt}}{1 + 4\pi n R^3/3} \tag{1.8}$$

Unlike Eq. 1.1 which is a purely convection equation, the mass transfer between different phases (water and vapor) needs to be added as a source term. The transport equation, say for phase l (liquid) could be written as

$$\frac{\partial \alpha_l}{\partial t} + \bigtriangledown \cdot \alpha_l \mathbf{u} = \frac{\dot{m}}{\rho_l}. \tag{1.9}$$

The source mass transfer term could be further re-organized as

$$\frac{\dot{m}}{\rho_l} = \frac{\dot{m}}{\rho_l} - \alpha_l(\frac{1}{\rho_l} - \frac{1}{\rho_v}) + \alpha_l(\frac{1}{\rho_l} - \frac{1}{\rho_v}) = (\frac{1}{\rho_l} - \alpha_l(\frac{1}{\rho_l} - \frac{1}{\rho_v}))\dot{m} + \alpha_l(\frac{1}{\rho_l} - \frac{1}{\rho_v})\dot{m}. \tag{1.10}$$

The first term in Eq. 1.10 on the right hand side could be decomposed into two terms (one for vaporization and one for condensation) with expression

$$\dot{m}_{ac} = C_c \alpha_l \frac{3\rho_l \rho_v}{\rho_m R} \sqrt{\frac{2}{3\rho_l}} \sqrt{\frac{1}{|p - p_{threhold}|}} max(p - p_{threhold}, 0), \tag{1.11}$$

$$\dot{m}_{av} = C_v (1 + \alpha_{nuc} - \alpha_l) \frac{3\rho_l \rho_v}{\rho_m R} \sqrt{\frac{2}{3\rho_l}} \sqrt{\frac{1}{|p - p_{threhold}|}} min(p - p_{threhold}, 0). \tag{1.12}$$

Then the total rate of mass transfer could be modified accordingly as

$$\dot{m} = \alpha_l \dot{m}_{av} + (1 - \alpha_l)\dot{m}_{ac} = \alpha_l(\dot{m}_{av} - \dot{m}_{ac}) + \dot{m}_{ac} \tag{1.13}$$

Finally, introducing $\dot{V} = (\frac{1}{\rho_l} - \alpha_l(\frac{1}{\rho_l} - \frac{1}{\rho_v}))$ and $\frac{\partial u_i}{\partial x_i} = (\frac{1}{\rho_l} - \frac{1}{\rho_v})\dot{m}$, the final transport equation could be written as

$$\frac{\partial \alpha_l}{\partial t} + \bigtriangledown \cdot (\alpha_l \mathbf{u}) = (\bigtriangledown \cdot \mathbf{u} + \dot{V}_v - \dot{V}_c)\alpha_l + \dot{V}_c \tag{1.14}$$

### 1.2.3 Laminar separation in $\gamma - Re_\theta$ model

The transition sensitive turbulence model, $\gamma - Re_\theta$ model, is established based on the Local Correlation based Transition Modelling (LCTM) concept, which means the model is based on correlations and the quantities used are locally determined for general usage of parallel computation. The model links the flow transition characteristics (transition onset, length of transition, etc.,) to flow features (pressure gradient, free stream turbulence intensity level, etc.,) based on series of two dimensional experimental data. The model is coupled with the $k - \omega SST$ RANS turbulence model using two additional equations. The first equation is the intermittency equation, which controls the production of kinetic energy inside the boundary layer; the second equation is a transport equation for transition momentum thickness Reynolds number, which is used to find the onset of transition and takes the non-local empirical correlations and transfers them into local qualities, as shown in Equation 1.15, the transport equation of intermittency ($\gamma$) and Equation 1.16, the transport equation of momentum thickness Reynolds number($\widetilde{R}e_{\theta t}$),

$$\frac{\partial(\rho\gamma)}{\partial t} + \frac{\partial \rho U_j \gamma}{\partial x_j} = P_\gamma - E_\gamma + \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_f}\right)\frac{\partial \gamma}{\partial x_j}\right], \tag{1.15}$$

$$\frac{\partial(\rho\widetilde{R}e_{\theta t})}{\partial t} + \frac{\rho U_j \widetilde{R}e_{\theta t}}{\partial x_j} = P_{\theta t} + \frac{\partial}{\partial x_j}\left[\sigma_{\theta t}(\mu + \mu_t)\frac{\partial \widetilde{R}e_{\theta t}}{\partial x_j}\right], \tag{1.16}$$

in which $\rho$ denotes density, $P$ and $E$ are the production and destruction terms, $\mu$ is the molecular dynamic viscosity, $\sigma_f$ and $\sigma_{\theta t}$ are model constants.

In this model, the foundation is to use the correlation of a scaled ratio of vorticity Reynolds number $Re_v = \frac{\rho y^2}{\mu} S$ and momentum thickness Reynolds number $Re_\theta$ to represent the boundary layer shape factor, as $\frac{max(Re_v)}{2.193 Re_\theta} \sim H$. Inside the boundary layer, $y^2 S$, the product of squared wall distance and strain rate magnitude, could be regarded as the growth of disturbances, and the term $\mu/\rho$ could be regarded as its damping; and transition will be triggered when a critical value of $Re_v$ is reached. In a laminar boundary layer, with the increase of adverse pressure gradient, the flow would be separating when shape factor H is higher than 3.5. Thus the scaling constant would be changed to 3.235 to get a better accuracy in between the target shape factor range. In the orginal formulation of the $\gamma - Re_\theta$ model, the separation induced transition is formulated as

$$\gamma_{sep} = min(s_1 max[0, \frac{Re_v}{3.235 Re_{\theta c}} - 1] F_{reattach}, 2) F_{\theta t}, \tag{1.17}$$

in which $s_1 = 2, F_{reattach} = e^{-\frac{R_T}{20}^4}$, and the blending function $F_{\theta t}$ is used to control $\gamma_{Sep}$ is only active inside the boundary layer. $F_{\theta t}$ is equal to zero in the free stream and one inside boundary layer, which formulation

$$F_{\theta t} = min(max(F_{wake} \cdot e^{-(\frac{y}{\delta})^4}), 1.0 - (\frac{\gamma - 1/c_{e2}}{1.0 - 1/c_{e2}})^2, 1.0),$$

$$\theta_{BL} = \frac{\tilde{Re}_{\theta t}\mu}{\rho U}, \quad \delta_{BL} = 7.5\theta_{BL}, \quad \delta = \frac{50\Omega y}{U} \cdot \delta_{BL},$$

$$Re_\omega = \frac{\rho\omega y^2}{\mu}, \quad F_{wake} = e^{-(\frac{Re_\omega}{1E5})^2}.$$

For the rest part of the model, the author refers to the orignal publication [Langtry and Menter(2009)].

## 1.3 Combination of laminar separation and cavitation

As mentioned in section 1.2.2, one of the assumption of the cavitation model is the uniform initial distribution of nuclei. One of the consequences of this assumption could be seen from the final formulation of vapor production and condensation, in which the sign of $(p - p_{sat})$ is the only factor to determine the cavitation is starting or not.

While under certain conditions this is not true. Many researches revealed that in most situations the sheet cavity actually detach shortly downstream the location of laminar separation and it is widely believed and observed, though the behind mechanism is still kind of unclear and also several limited researches seemingly proposed other possible mechanism and claims the existence of laminar separation is not a must for sheet cavity detachment.

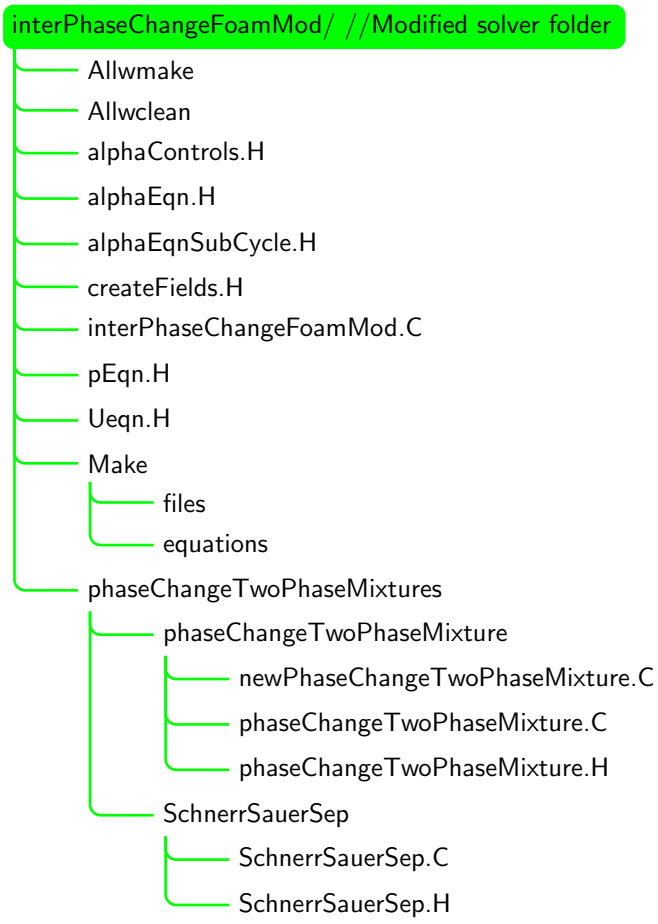
The combination of the laminar separation indicator and cavitation model could be achieved via the modification of the two constants, $C_c$ and $C_v$, which are the vaporization and condensation coefficients. This is somewhat desired, since the two constants could be modified according to the user; while taking into consideration the fact that the laminar separation indicator would be modified to be one or zero, the original form of the cavitation model should be maintained as much as possible.

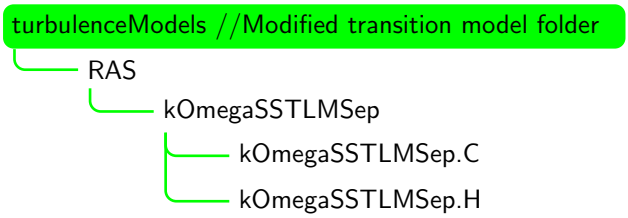
# Chapter 2

# Implementation

In this chapter, the implementation of the VOF treatment, the Scheneer - Sauer cavitation mass transfer model, the laminar separation indicator and the combination of the laminar separation and cavitation model will be explained step by step. The implementation is based on OpenFOAM version v1806.

The implementation of VoF part will be discussed and modified for the current application usage. For the transition model, the implementation of laminar separation induced transition part will be discussed and possible modification will be described. The implementation of the Scheneer Sauer cavitation model will be described, and the modification will be presented. The final structure of the modified folders are shown in below.






## 2.1 Modification of $\alpha$ transportation

As described in Section 1.2.2, the transport equation of $\alpha$ could be written as

$$\frac{\partial \alpha_l}{\partial t} + \bigtriangledown \cdot \alpha \mathbf{u} = (\bigtriangledown \cdot \mathbf{u} + \dot{V}_v - \dot{V}_c)\alpha_l + \dot{V}_c, \tag{2.1}$$

which is implemented in the original solver, interPhaseChangeFoam. When using CMULES algorithm, the equation would be solved first with the first order Euler Scheme for the first term and upwind flux for the second term, as shown in the code located in the alphaEqn.H,

```
if (MULESCorr)
{
    fvScalarMatrix alpha1Eqn
    (
        fv::EulerDdtScheme<scalar>(mesh).fvmDdt(alpha1)
      + fv::gaussConvectionScheme<scalar>
        (
            mesh,
            phi,
            upwind<scalar>(mesh, phi)
        ).fvmDiv(phi, alpha1)
      - fvm::Sp(divU, alpha1)
     ==
        fvm::Sp(vDotvmcAlphal, alpha1)
      + vDotcAlphal
    );

    alpha1Eqn.solve();

    Info<< "Phase-1 volume fraction = "
        << alpha1.weightedAverage(mesh.Vsc()).value()
        << "  Min(" << alpha1.name() << ") = " << min(alpha1).value()
        << "  Max(" << alpha1.name() << ") = " << max(alpha1).value()
        << endl;

    talphaPhi = alpha1Eqn.flux();
}
```

Then the flux using higher order schemes would be calculated, as

```
    tmp<surfaceScalarField> talphaPhiCorr
    (
        fvc::flux
        (
            phi,
            alpha1,
            alphaScheme
```

```
            )
          + fvc::flux
            (
                -fvc::flux(-phir, alpha2, alpharScheme),
                alpha1,
                alpharScheme
            )
```

And the MULES correction would be performed with possibly under-relaxation

```
if (MULESCorr)
       {
           talphaPhiCorr.ref() -= talphaPhi();

           volScalarField alpha100("alpha100", alpha10);
           alpha10 = alpha1;

           MULES::correct
           (
               geometricOneField(),
               alpha1,
               talphaPhi(),
               talphaPhiCorr.ref(),
               vDotvmcAlphal,
               (
                   divU*(alpha10 - alpha100)
                 - vDotvmcAlphal*alpha10
               )(),
               1,
               0
           );

           // Under-relax the correction for all but the 1st corrector
           if (aCorr == 0)
           {
               talphaPhi.ref() += talphaPhiCorr();
           }
           else
           {
               alpha1 = 0.5*alpha1 + 0.5*alpha10;
               talphaPhi.ref() += 0.5*talphaPhiCorr();
           }
       }
```

Since only two phases exist in the simulation, the second phase volume fraction could be updated as `alpha2 = 1.0 - alpha1`. Lastly the flux could be updated as

```
rhoPhi = talphaPhi()*(rho1 - rho2) + phi*rho2;
```

However, as mentioned in the orignal publication of Schenerr-Sauer cavitation model, the upwind differencing scheme should be used [Schnerr and Sauer(2001)] instead of possible High Resolution Schemes like HRIC or CICSAM. CMULES is also intended to keep a 'sharp' interface which conflicts with the homogeneous assumption, thus the alphaEqn.H could actually be modified based on their suggestions.

For the modification, firstly copy the original solver to `$WM_PROJECT_USER_DIR`.

```
cd $WM_PROJECT_USER_DIR
mkdir applications; cd $WM_PROJECT_DIR/applications
cp -r --parents ./solvers/multiphase/interPhaseChangeFoam $WM_PROJECT_USER_DIR/applications
cd $WM_PROJECT_USER_DIR/applications/solvers/multiphase/
```

Rename the solver to be modified;

```
mv interPhaseChangeFoam interPhaseChangeFoamMod; cd interPhaseChangeFoamMod
mv interPhaseChangeFoam.C interPhaseChangeFoamMod.C
sed -i s/interPhaseChangeFoam/interPhaseChangeFoamMod/g interPhaseChangeFoamMod.C
sed -i s/interPhaseChangeFoam/interPhaseChangeFoamMod/g Make/*
sed -i s/FOAM_APPBIN/FOAM_USER_APPBIN/g interPhaseChangeFoamMod/Make/files
```

Modify `alphaEqnSubCycle.H`, remove the MULES related parts:

```
{
    volScalarField divU(fvc::div(phi));

    if (nAlphaSubCycles > 1)
    {
        dimensionedScalar totalDeltaT = runTime.deltaT();
        surfaceScalarField rhoPhiSum("rhoPhiSum", rhoPhi);

        for
        (
            subCycle<volScalarField> alphaSubCycle(alpha1, nAlphaSubCycles);
            !(++alphaSubCycle).end();
        )
        {
            #include "alphaEqn.H"
            rhoPhiSum += (runTime.deltaT()/totalDeltaT)*rhoPhi;
        }

        rhoPhi = rhoPhiSum;
    }
    else
    {
        #include "alphaEqn.H"
    }

    rho == alpha1*rho1 + alpha2*rho2;
}
```

Then remove unnecessary parts in `alphaControls.H`

```
echo "const dictionary& alphaControls = mesh.solverDict(alpha1.name());" > alphaControls.H
echo "label nAlphaCorr(readLabel(alphaControls.lookup(\"nAlphaCorr\")));" >> alphaControls.H
echo "label nAlphaSubCycles(readLabel(alphaControls.loo
kup(\"nAlphaSubCycles\")));" >> alphaControls.H
```

Lastly modify `alphaEqn.H` to remove MULES related parts and artificial compression term in the $\alpha$ equation, with following content:

```
{
    {
     word alphaScheme("div(phi,alpha)");
     for (int aCorr=0; aCorr<nAlphaCorr; aCorr++)
     {
```

```
        Pair<tmp<volScalarField>> vDotAlphal =
        mixture->vDotAlphal();
        const volScalarField& vDotcAlphal = vDotAlphal[0]();
        const volScalarField& vDotvAlphal = vDotAlphal[1]();
        const volScalarField vDotvmcAlphal(vDotvAlphal - vDotcAlphal);
        fvScalarMatrix alpha1Eqn
        (
            fvm::ddt(alpha1)
          + fvm::div(phi, alpha1, "div(phi,alpha)")
          - fvm::Sp(divU, alpha1)
          ==
            fvm::Sp(vDotvmcAlphal, alpha1)
          + vDotcAlphal
        );
        alpha1Eqn.solve();
        alpha1 = min(max(alpha1, scalar(0)), scalar(1));
        alpha2 = scalar(1) - alpha1;
    }
    rhoPhi = fvc::flux(phi,alpha1, "div(phi,alpha)")*(rho1 - rho2) + phi*rho2;
    Info<< "Liquid phase volume fraction = "
        << alpha1.weightedAverage(mesh.V()).value()
        << "  Min(" << alpha1.name() << ") = " << min(alpha1).value()
        << "  Max(" << alpha1.name() << ") = " << max(alpha1).value()
        << endl;
}
```

## 2.2 Laminar separation indicator

The $\gamma - Re_\theta$ transition model is located in

`$WM_PROJECT_DIR/src/TurbulenceModels/turbulenceModels/RAS/kOmegaSSTLM`

Inside the file `kOmegaSSTLM.C`, just following the transport equation of intermittency, the laminar separation induced transition is calculated via `gammaSep`, as

```
    const volScalarField::Internal gammaSep
    (
        min(2*max(Rev/(3.235*ReThetac) - 1, scalar(0))*Freattach, scalar(2))
       *Fthetat
    );
    gammaIntEff_ = max(gammaInt_(), gammaSep);
```

in which it means when $Re_v/(3.235Re_{\theta c})$ is larger than one, locally the gamma value will increase and start producting kinetic energy $k$ as a source term in $k$ equation, thus lead to a turbulent flow.

However, there is an issue if directly using this as a cavitation indicator, due to the fact that for this group of LCTM, the transition starts roughly at the middle of boundary layer. This is undesired serving as an indicator for cavitation, due to the fact that it is believed the detachment of cavity should start on the wall instead of from the middle of boundary layer profile. The cavity inception is highly related to the laminar separation and reverse flow, which forms kind of a 'stable' region in which nuclei could grow and stay there to form a sheet cavity or it would be sweep away. Thus the cavitation indicator need to be modified based on `gammaSep`.

In the original code, the $\gamma - Re_\theta$ model was coupled with the $k - \omega SST$ turbulence model; and its implementation was templated with the `kOmegaSSTBase`.

For the implementation, the original turbulence model should be copied to the `$WM_PROJECT_USER_DIR`,
`cd $WM_PROJECT_DIR; cp -r --parents ./src/TurbulenceModels $WM_PROJECT_USER_DIR`

```
cd $WM_PROJECT_USER_DIR/src/TurbulenceModels
```

Modify the file `incompressible/turbulentTransportModels/turbulentTransportModels.C`, add following sentences after `makeRASModel(kOmegaSSTLM);`

```
#include "kOmegaSSTLMSep.H"
makeRASModel(kOmegaSSTLMSep);
```

```
sed -i s/FOAM_LIBBIN/FOAM_USER_LIBBIN/g turbulenceModels/Make/files
cd turbulenceModels/RAS; cp -r kOmegaSSTLM kOmegaSSTLMSep; cd kOmegaSSTLMSep
sed -i s/FOAM_LIBBIN/FOAM_USER_LIBBIN/g ./*/Make/files
mv *.C kOmegaSSTLMSep.C; mv *.H kOmegaSSTLMSep.H
sed -i s/kOmegaSSTLM/kOmegaSSTLMSep/g *
```

After the intermittency equation, below the sentence `gammaIntEff_ = max(gammaInt_(), gammaSep);`, add following

```
// Modifying the laminar separation indictor
        SepInd_.ref() = gammaSep;
        dimensionedScalar udimk("udimk", dimensionSet(0,-2,2,0,0,0,0), scalar(1.0));
        SepInd_ =  min( (SepInd_ * max((k - kInf*2)*udimk, scalar(0)))*1e6, scalar(1.0));
        volScalarField Rev_y = 1.0/y_ ;
        volVectorField gradRev_y = fvc::grad(Rev_y);
        volVectorField nor_gradRev_y = gradRev_y/mag(gradRev_y);
        surfaceScalarField yPhi
        (
        linearInterpolate(nor_gradRev_y) & this->mesh_.Sf()
        );
        surfaceScalarField yPhio(min(yPhi,scalar(0.0)*yPhi));
       dimensionedScalar udiml("udiml", dimensionSet(0,-1,0,0,0,0,0), scalar(1.0));
        // Projection equation to the wall
        tmp<fvScalarMatrix> SepIndEqn
        (
            fvm::div(yPhi, SepInd_)
        ==
            SepInd_*scalar(100.0)*udiml
        );
        solve(SepIndEqn);
        SepInd_ = min(SepInd_*scalar(1e16), scalar(1.0));
```

Then add the missing constructors for the newly added variables, i.e. `SepInd_` and `kInf`; below the constructor of `ReThetat_`, add following sentences

```
    SepInd_
    (
        IOobject
        (
            IOobject::groupName("SepInd", alphaRhoPhi.group()),
            this->runTime_.timeName(),
            this->mesh_,
            IOobject::NO_READ,
            IOobject::AUTO_WRITE
        ),
        this->mesh_
    ),
    y_(wallDist::New(this->mesh_).y()),
```

```
    kInf
      (
        dimensioned<scalar>::lookupOrDefault
        (
            "kInf",
            this->coeffDict_,
            dimensionSet( 0, 2, -2, 0, 0, 0, 0),
            0.001
        )
     ),
```

And declare them in `kOmegaSSTLMSep.H`, after the declaration of `ReThetat_`:

```
            volScalarField SepInd_;
            const volScalarField& y_;
            volScalarField kInf;
```

Lastly compile the code.
`cd $WM_PROJECT_USER_DIR/src/TurbulenceModels; ./Allwmake`

## 2.3 Schnerr Sauer cavitation model

As shown in Eqn 1.14, the alpha transport equation in OpenFOAM is implemented in the `interPhaseChangeFoam` solver as

```
        const volScalarField& vDotcAlphal = vDotAlphal[0]();
        const volScalarField& vDotvAlphal = vDotAlphal[1]();
        const volScalarField vDotvmcAlphal(vDotvAlphal - vDotcAlphal);
        fvScalarMatrix alpha1Eqn
        (
            fvm::ddt(alpha1)
          + fvm::div(phi, alpha1, "div(phi,alpha)")
          - fvm::Sp(divU, alpha1)
          ==
            fvm::Sp(vDotvmcAlphal, alpha1)
          + vDotcAlphal
        );
```

The mass transfer part was calculated by the Schnerr Sauer phase transfer model. Inside the `phaseChangeTwoPhaseMixtures` folder, under `SchnerrSauer`, one could find following content inside the .C file,

```
Foam::Pair<Foam::tmp<Foam::volScalarField>>
Foam::phaseChangeTwoPhaseMixtures::SchnerrSauer::mDotAlphal() const
{
    const volScalarField& p = alpha1_.db().lookupObject<volScalarField>("p");
    volScalarField pCoeff(this->pCoeff(p));

    volScalarField limitedAlpha1(min(max(alpha1_, scalar(0)), scalar(1)));

    return Pair<tmp<volScalarField>>
    (
        Cc_*limitedAlpha1*pCoeff*max(p - pSat(), p0_),

        Cv_*(1.0 + alphaNuc() - limitedAlpha1)*pCoeff*min(p - pSat(), p0_)
    );
```

```
}
Foam::Pair<Foam::tmp<Foam::volScalarField>>
Foam::phaseChangeTwoPhaseMixtures::SchnerrSauer::mDotP() const
{
    const volScalarField& p = alpha1_.db().lookupObject<volScalarField>("p");
    volScalarField pCoeff(this->pCoeff(p));

    volScalarField limitedAlpha1(min(max(alpha1_, scalar(0)), scalar(1)));
    volScalarField apCoeff(limitedAlpha1*pCoeff);

    return Pair<tmp<volScalarField>>
    (
        Cc_*(1.0 - limitedAlpha1)*pos0(p - pSat())*apCoeff,

        (-Cv_)*(1.0 + alphaNuc() - limitedAlpha1)*neg(p - pSat())*apCoeff
    );
}
```

Inside `interPhaseChangeFoam/phaseChangeTwoPhaseMixtures/phaseChangeTwoPhaseMixture`, one could find

```
Foam::Pair<Foam::tmp<Foam::volScalarField>>
Foam::phaseChangeTwoPhaseMixture::vDotAlphal() const
{
    volScalarField alphalCoeff(1.0/rho1() - alpha1_*(1.0/rho1() - 1.0/rho2()));
    Pair<tmp<volScalarField>> mDotAlphal = this->mDotAlphal();

    return Pair<tmp<volScalarField>>
    (
        alphalCoeff*mDotAlphal[0],
        alphalCoeff*mDotAlphal[1]
    );
}

Foam::Pair<Foam::tmp<Foam::volScalarField>>
Foam::phaseChangeTwoPhaseMixture::vDotP() const
{
    dimensionedScalar pCoeff(1.0/rho1() - 1.0/rho2());
    Pair<tmp<volScalarField>> mDotP = this->mDotP();

    return Pair<tmp<volScalarField>>(pCoeff*mDotP[0], pCoeff*mDotP[1]);
}
```

## 2.4 Modification of cavitation model

Now the laminar separation indicator and cavitaion model could be combined together.

```
cd $WM_PROJECT_USER_DIR/applications/solvers/multiphase
cd interPhaseChangeFoamMod/phaseChangeTwoPhaseMixtures/
cp -r SchnerrSauer SchnerrSauerSep; cd SchnerrSauerSep
mv *C SchnerrSauerSep.C; mv *H SchnerrSauerSep.H
sed -i s/SchnerrSauer/SchnerrSauerSep/g *
```

Inside the `SchnerrSauerSep.C` file, under Member Functions, below the function `pCoeff`, add

```
Foam::tmp<Foam::volScalarField>
```

```
Foam::phaseChangeTwoPhaseMixtures::SchnerrSauerSep::CvSep
(
    const volScalarField& sepind
) const
{
    return
    Cv_*sepind;
}
```

Also modify the sentence in function `mDotAlphal()`, replace

```
Cv_*(1.0 + alphaNuc() - limitedAlpha1)*pCoeff*min(p - pSat(), p0_)
```

with

```
CvSep*(1.0 + alphaNuc() - limitedAlpha1)*pCoeff*min(p - pSat(), p0_)
```

The same for the sentence in function `mDotP()`, replace

```
(-Cv_)*(1.0 + alphaNuc() - limitedAlpha1)*neg(p - pSat())*apCoeff
```

with

```
(-CvSep)*(1.0 + alphaNuc() - limitedAlpha1)*neg(p - pSat())*apCoeff
```

The modified sentences actually modified the user-defined vaporization rate while the condensation rate is kept unchanged. According the fact the modified indicator is either one or zero, the modification only controls the region of vaporization and the rest of the code are kept as much as similar with the original formulation and functionality, i.e. the user still could able to set the `Cc` and `Cv` according to the need.

Then modify necessary changes for the wmake options.

```
cd $WM_PROJECT_USER_DIR/applications/solvers/multiphase/interPhaseChangeFoamMod
cd phaseChangeTwoPhaseMixtures
rm Make/files; touch Make/files
echo "phaseChangeTwoPhaseMixture/phaseChangeTwoPhaseMixture.C" > Make/files
echo "phaseChangeTwoPhaseMixture/newPhaseChangeTwoPhaseMixture.C" >> Make/files
echo "Kunz/Kunz.C " >> Make/files
echo "Merkle/Merkle.C" >> Make/files
echo "SchnerrSauer/SchnerrSauer.C" >> Make/files
echo "SchnerrSauerSep/SchnerrSauerSep.C" >>Make/files
echo "LIB = \$(FOAM_USER_LIBBIN)/libphaseChangeTwoPhaseMixtures" >> Make/files

cd ..; ./Allwmake
```

# Chapter 3

# Test Case

## 3.1 Case description

Many researches have been performed to study the sheet cavitation inception, both experimentally and numerically. The study case used here comes from the previously study by J. P. Franc and J. M. Michel. The experimental studies of the NACA 16012 hydrofoil were preformed in the second free-surface channel of the hydrodynamic tunnel at Grenoble University. The visualization of the boundary layer was achieved by dye injection at the leading edge.

As shown in Figure 3.1, with the AoA=3deg and Re=300,000, which is one condition of the performed experiments, the separation starts at the rear part of the foil at the location roughly $x/C \sim 0.76 - 0.8$.

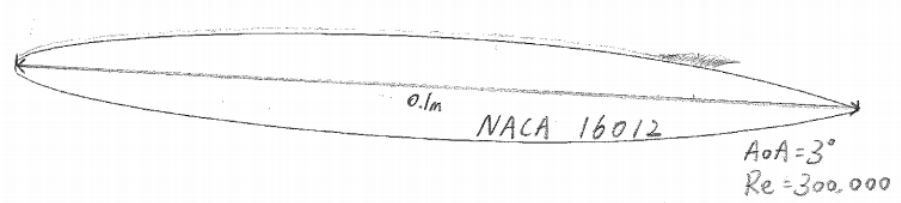



Figure 3.1: NACA 16012, non-cavitating flow for Re = 300,000 with angle of attack = 3 degree. Reproduced from [Franc and Michel(1985)].

One of the photoed cavitation patterns is shown in Figure 3.2. The Reynolds number is actually increased to 1,000,000. The laminar separation point is moved upstream to about 25% $\sim$ 30% chord length. Since the location of laminar separation point is a weak function with Reynolds number, mainly the cavity changed the location of laminar separation point to upstream.

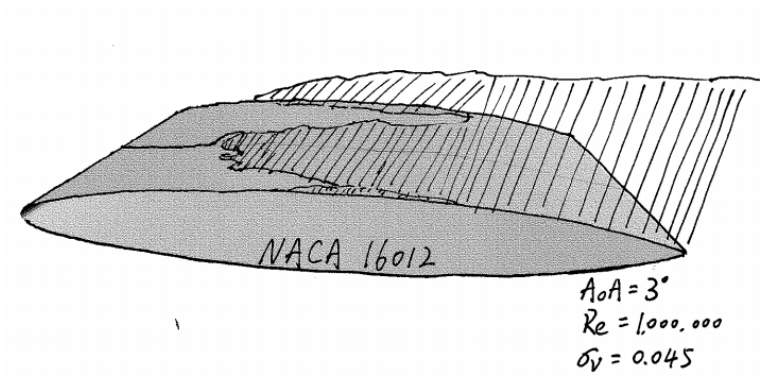



Figure 3.2: NACA 16012, Re = 1,000,000 with angle of attack = 3 degree and $\sigma_v$ = 0.045. [Franc and Michel(1985)].

## 3.2 Mesh and boundary conditions

The mesh used for the current numerical simulation is shown in Figure 3.3. As a requirement of the transition model, the y+ should be below 1 and the boundary prism layer growth ratio should ideally be between 1.05 to 1.1. Sufficient y and z direction resolution is also required for the accurate prediction of laminar boundary layer to turbulence boundary layer. As shown in Figure 3.2, the cavitation pattern showed some 3D effect, while the study showing here is based on two dimensional simulations. This is somewhat limited, but as described in the experiments that the influence of 3D effect is rather insignificant for the current case, and the origin of sheet cavity inception was not caused by the vortex formation which makes the 2D simulation acceptable.

The current mesh was generated using `Pointwise` with structured mesh, and the boundary layer was extruded from the foil surface using hyperbolic extrusion method. The mesh consists 80,758 hexahedra cells.

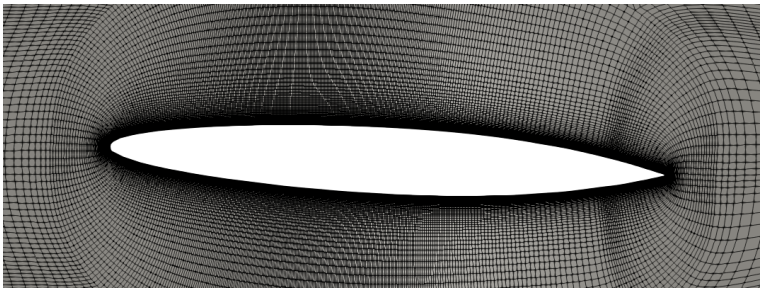


Figure 3.3: 2D Mesh used for numerical simulation.

The simulation domain and boundary conditions are shown in Figure 3.4.

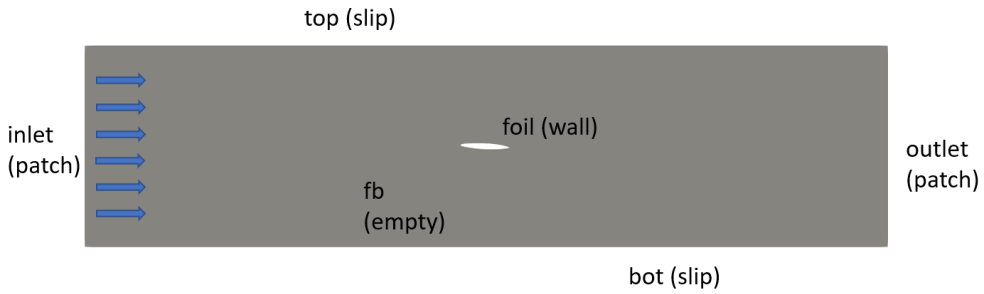



Figure 3.4: Simulation domain and boundary conditions

The turbulence intensity is reported and fixed at the inlet with value equals to 0.15% which is rather low. For $Tu < 1.3\%$, the inlet value of $Re_\theta$ should be calculated as

$$Re_\theta = 1173.51 - 589.428Tu + \frac{0.2196}{Tu^2} \quad Tu < 1.3\%;$$

$$Re_\theta = 331.5[Tu - 0.5658]^{-0.671} \quad Tu > 1.3\%.$$

Accordingly, the $k$ and $\omega$ value at inlet could be calculated based on the assumption of $R_T = 5$ using the formulation

$$k_{inlet} = 1.5(U_{inlet} Tu_{inlet})^2$$

$$R_T = \frac{\nu_t}{\nu} = \frac{\rho k}{\nu\omega}, \quad \omega_{inlet} = \frac{\rho k}{\nu R_T}$$

The inlet velocity could be calculated based on Reynolds number and water properties. `Slip` boundary condition is used on the `top` and `bot` boundaries. Non-slip condition should be used on the foil; the `k` and `nut` are set to fixed value 0 on the wall boundaries. The `omegaWallFunction` should be used on the foil, according top the formulation

$$\omega = \frac{6\nu}{\beta_1(\Delta y_1)^2}$$

The boundary conditions are summarized in table

| | U | p | nut | alpha.water |
|---|---|---|---|---|
| inlet | fixedValue | zeroGradient | Calculated | fixedValue |
| outlet | zeroGradient | fixedValue | Calculated | zeroGradient |
| top | slip | slip | slip | slip |
| bot | slip | slip | slip | slip |
| foil | fixedValue | zeroGradient | fixedValue | zeroGradient |
| fb | empty | empty | empty | empty |

| | k | omega | gammaInt | ReThetat | SepInd |
|---|---|---|---|---|---|
| inlet | fixedValue | fixedValue | fixedValue | fixedValue | Calculated |
| outlet | zeroGradient | zeroGradient | zeroGradient | zeroGradient | Calculated |
| top | slip | slip | slip | slip | Calculated |
| bot | slip | slip | slip | slip | Calculated |
| foil | fixedValue | omegaWallFunction | zeroGradient | zeroGradient | Calculated |
| fb | empty | empty | empty | empty | Calculated |

Table 3.1: Summary of boundary conditions

The turbulence decay control should be activated since the decay of turbulence terms with a $k\omega - SST$ based model is rather significant and the accurate prediction of laminar - transition is highly dependent on the turbulence intensity level at the object leading edge. This could be achieved by specifying the entry in `constant/turbulenceProperties`, set `decayControl` to `yes` and define `kInf` and `omegaInf` according to inlet conditions. Also, the `maxLambdaIter` should be set to a higher value like 30 instead of the default value, which controls the maximum steps to solve a root finding problem via iteration.

## 3.3 Simulation results

### 3.3.1 Non-cavitating condition with Re = 300,000

Simulation should be performed firstly for the condition with Reynolds number 300,000 under non-cavitating condition. A steady-state simulation should be performed using the `SIMPLEFOAM` solver. The predicted location of laminar separation region starts around $x/C \sim 0.78$. The prediction is quite consistent with experimental visualization, as shown in Figure 3.1.

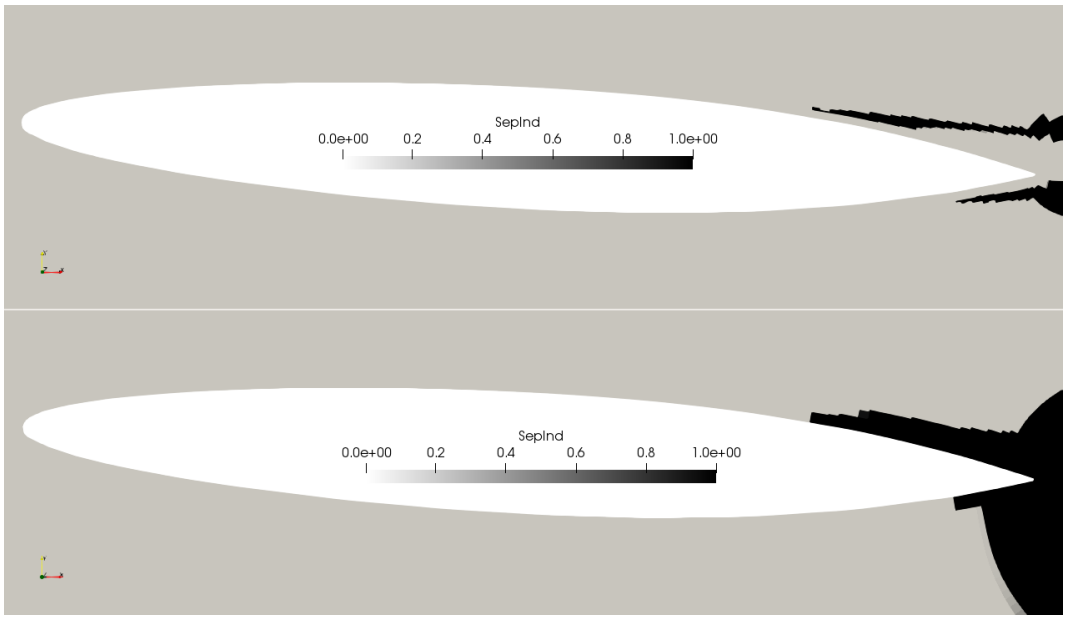



Figure 3.5: The laminar separation indicator, top: original predicted region; bottom: predicted region with modified code. Non-cavitation condition, Re = 300,000, AoA = 3deg.

### 3.3.2 Non-cavitating condition with Re = 1,000,000

The non-cavitating condition simulation for the condition with Reynolds number of 1,000,000 to have an idea of the location of laminar separation. It could be seen that the location of laminar separation moved upstream (about $x/c \sim 0.65$) due to the increase of Reynolds number.

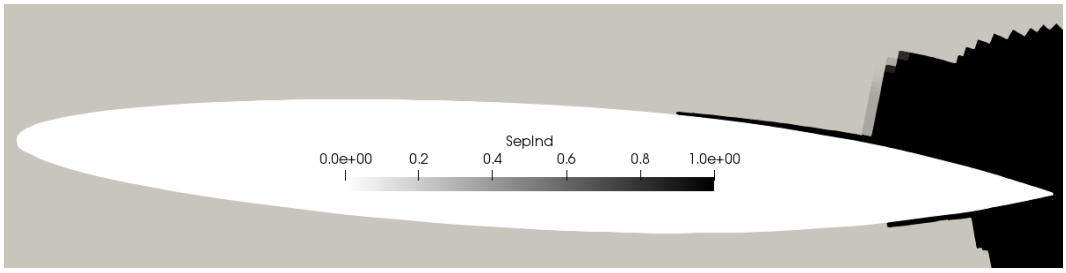



Figure 3.6: The laminar separation indicator with modified code, non-cavitating condition, Re = 1,000,000, AoA = 3deg.

### 3.3.3 Cavitating condition with Re = 1,000,000

Simulations were performed for the cavitating condition, with the modified solver and original solver. For the cavitating case, the `interPhaseChangeFoamMod` solver should be used with suitable time advancing scheme. The `backward` time scheme was used for the current study.

The steady-state simulation results were used as initial condition for the transient case. It should be mention that the pressure acquired from `SIMPLEFOAM` is $p/\rho$, thus the conversion should be performed to get `p_rgh` based on the formulation

$$p_{rgh} = p_{simpleFoam} * \rho + p_{ref},$$

in which $p_{ref}$ should be calculated based on the cavitation number. The boundary condition of `p_rgh` should be modified, i.e. fixedValue with $p_{ref}$ at the outlet; and `zeroGradient` should be changed to `fixedFluxPressure` for the cases in which gravity is considered.

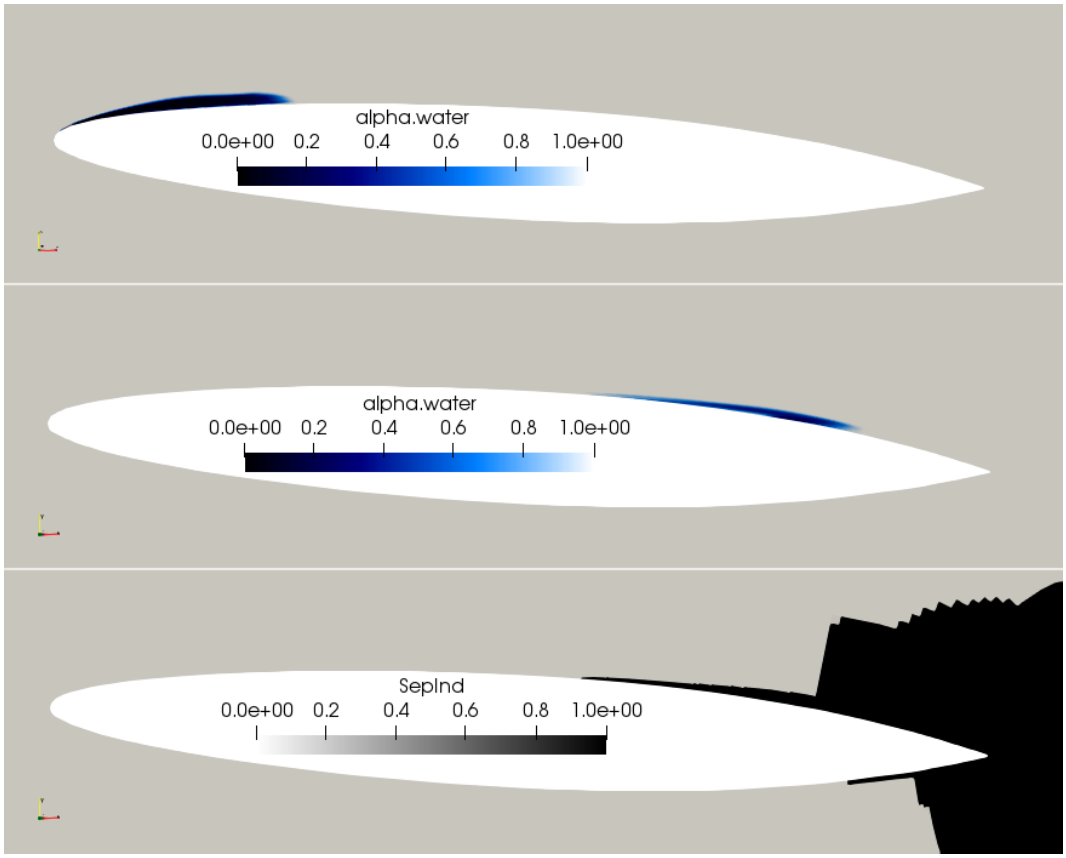



Figure 3.7: Predicted cavitation pattern and separation indicator at time 0.002s. From top to bottom: cavitation pattern, original solver; cavitation pattern, modified solver; separation indicator, modified solver.

As shown in Figure 3.7, for the modified solver, the cavitation starts at the location of laminar separation instead of from the minimum pressure, which should be the case in the experiments.

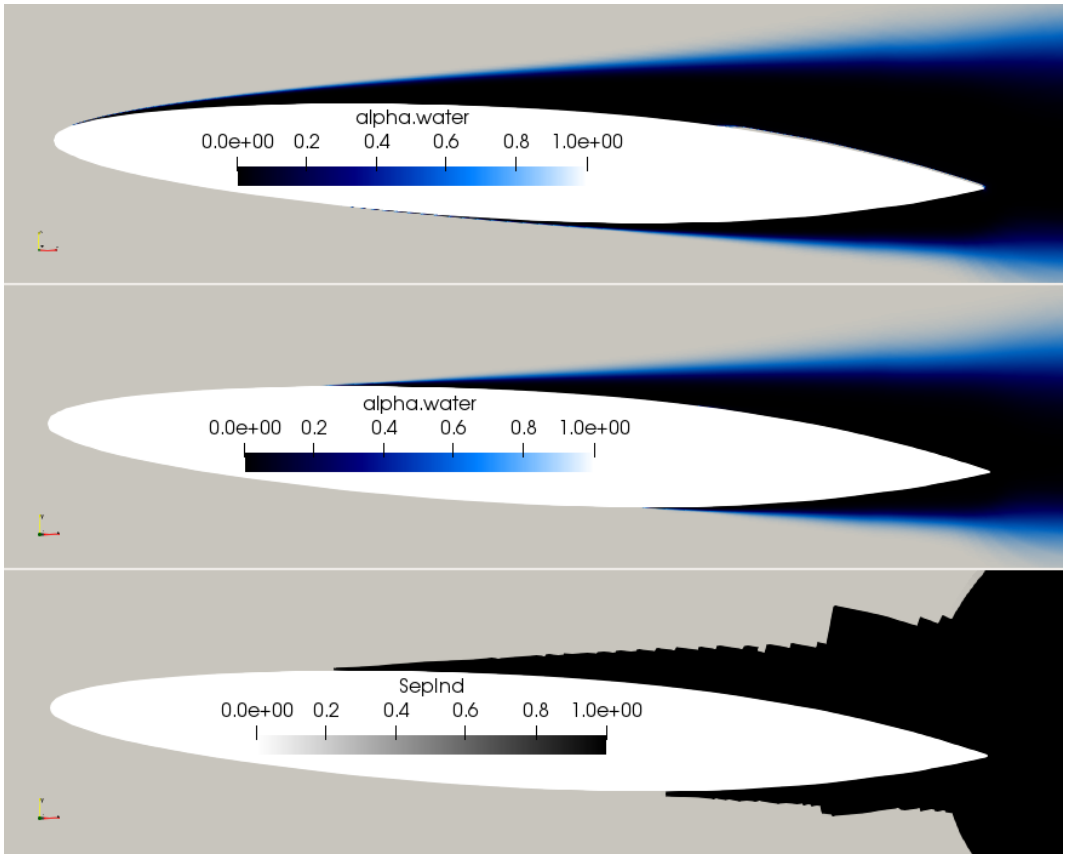



Figure 3.8: Predicted cavitation pattern and separation indicator at time 0.1s. From top to bottom: cavitation pattern, original solver; cavitation pattern, modified solver; separation indicator, modified solver.

After about 0.03s a kind of steady-state solution could be researched. As shown in Figure 3.8, at time 0.1s, a new state of equilibrium could be reached for the modified solver. The predicted cavitation agreed well compared to experimental observations, which is shown in Figure 3.2.

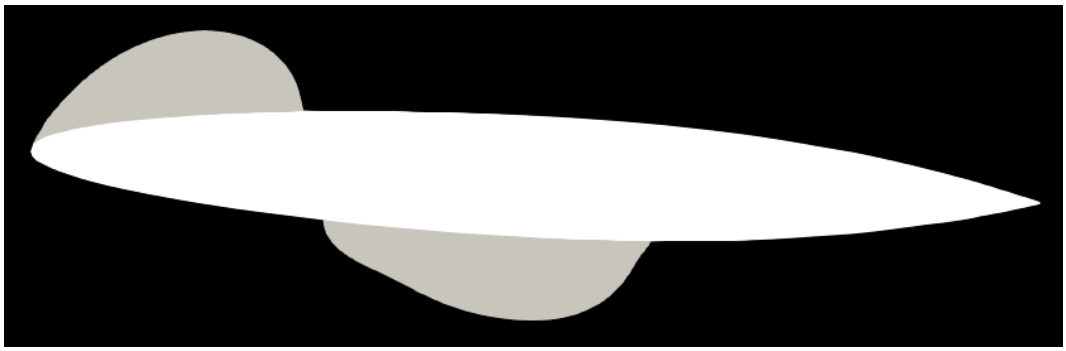


Figure 3.9: Pressure contour at time 0.1s. The grey color region show the pressure below saturation pressure.

The Figure 3.9 shows the pressure contour at time 0.1s, at which the solution becomes steady. The black region shows the pressure higher than the satuation pressure and the grey region shows the pressure is below satuation pressure.

# Bibliography

[Arakeri(1975)] Vijay H Arakeri. Viscous effects on the position of cavitation separation from smooth bodies. *Journal of Fluid Mechanics*, 68(4):779–799, 1975.

[Casey(1974)] M V. Casey. The inception of attached cavitation from laminar separatiohn bubbnles on hydrofoils. In *Proc. of Conf. on Cavitation*, Edinburgh, 1974.

[Franc and Michel(1985)] Jean-Pierre Franc and Jean-Marie Michel. Attached cavitation and the boundary layer: experimental investigation and numerical treatment. *Journal of Fluid Mechanics*, 154:63–90, 1985.

[Langtry and Menter(2009)] Robin B Langtry and Florian R Menter. Correlation-based transition modeling for unstructured parallelized computational fluid dynamics codes. *AIAA journal*, 47 (12):2894–2906, 2009.

[Schnerr and Sauer(2001)] Günter H Schnerr and Jürgen Sauer. Physical and numerical modeling of unsteady cavitation dynamics. In *Fourth International Conference on Multiphase Flow*, volume 1. ICMF New Orleans, 2001.

[Wang et al.(2001)Wang, Senocak, Shyy, Ikohagi, and Cao] Guoyu Wang, Inanc Senocak, Wei Shyy, Toshiaki Ikohagi, and Shuliang Cao. Dynamics of attached turbulent cavitating flows. *Progress in Aerospace sciences*, 37(6):551–581, 2001.

# Study questions

1. What is the mesh requirements for the use of $\gamma - Re_\theta$ transition model?
2. How is the laminar separation induced transition predicted in the transition model?
3. Which scheme should be used for the calculation of $\alpha$ convection when using the Schnerr Sauer cavitation model?
4. How are the laminar separation indicator and cavitation model combined in the present report?
5. List the assumptions of the Schnerr-Sauer cavitation model.