# Warsaw University of Technology

FACULTY OF POWER AND AERONAUTICAL ENGINEERING

Institute of Aeronautics and Applied Mechanics

# Bachelor’s diploma seminar

in the field of study Mechanics and machine Design and specialisation
Computer-Aided Engineering

An EARSM Turbulent modeling in OpenFOAM

Purn Prakash
student record book number 281142

Thesis supervisor
Associate Professor Sławomir Kubacki

Warsaw, 2019

# 1. Theoratical backgroud

## 1.1 The physics of transition

The transition mechanism of a laminar flow into turbulent flow, with the objective of modeling with Reynolds-averaged Navier-Strokes (RANS) description of a flow, is categorized into four forms:

### 1.1.1 Natural Transition

This kind of transition mechanism is characterized as a transition in an attached boundary layer when two-dimensional Tollmien-Schlichting instability waves appears and grows in downstream direction to for a three-dimensional instability. This forms a periodic hairpin vortices in the span-wise direction. This hairpin vortices breakdown the laminar flow in the downstream direction causing turbulent spots. This kind of transition appears when the flow has a statically steady and very low mean-flow of low turbulence level. This is a slow process. This kind of transition is possible in cases like external aerodynamics.

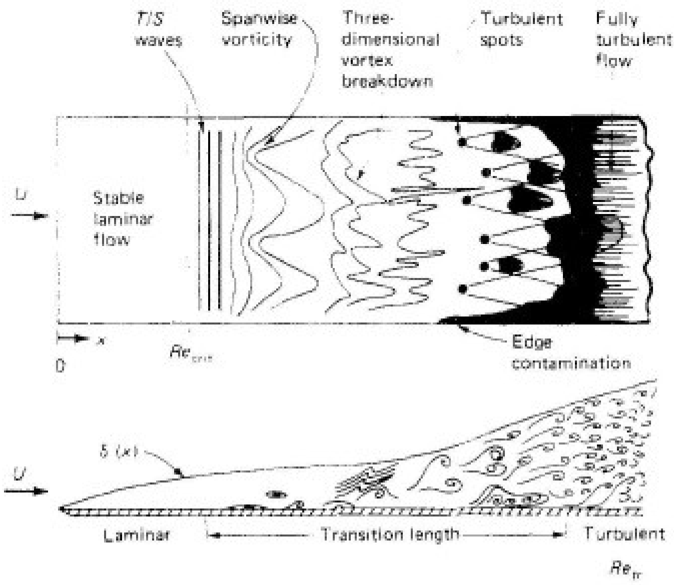
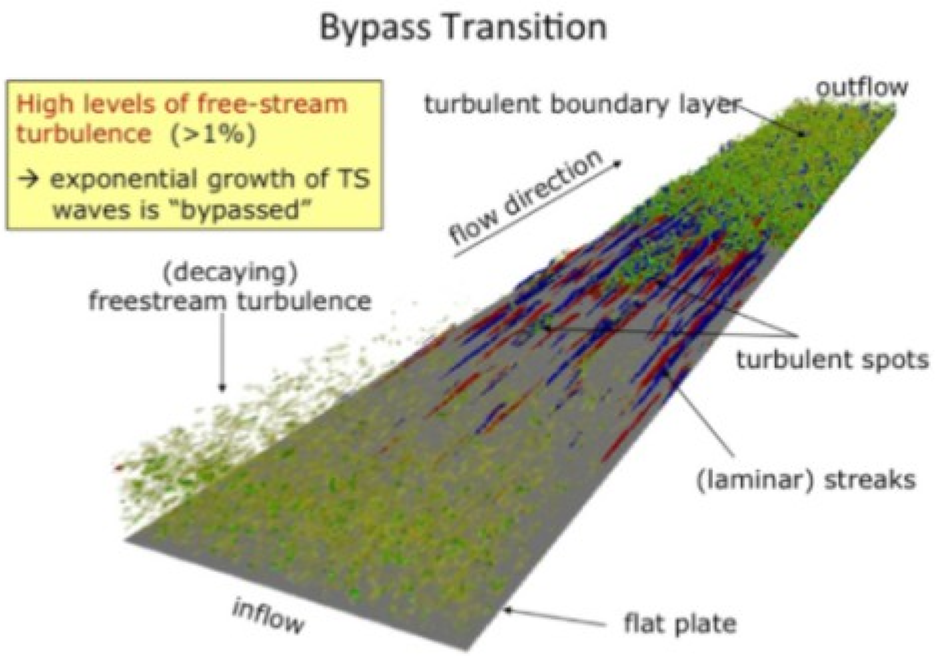



Fig 1. natural transition. source: liverpool.ac.uk

### 1.1.2 1.1.2. Bypass Transition

This type of transition appears when a sufficiently high level of turbulence is present in the flow, in the range of 0.5% to 1 % of turbulent intensity. This type of transition earns its name because the instability causing turbulence is not generated by the Tollmein-Schlichting waves like in the above type of transition called the natural transition. In this case, in the near-wall region of the laminar flow attached to a surface, there appears some stream-wise elongated disturbances, which are zones of forward and backward jet-like perturbations. They are called Klebanoff distortions or simple streaks. These streaks grow downstream in length and amplitude. Further, they breakdown forming turbulent spots in the flow. These perturbations have the following properties:

- They have alternate span-wise direction.
- They are almost periodic.
- They posses wavelength of the order of the boundary layer thickness in their vicinity.




Bypass-transition. Source- tomer-cfdisrael-tenzorblog.com

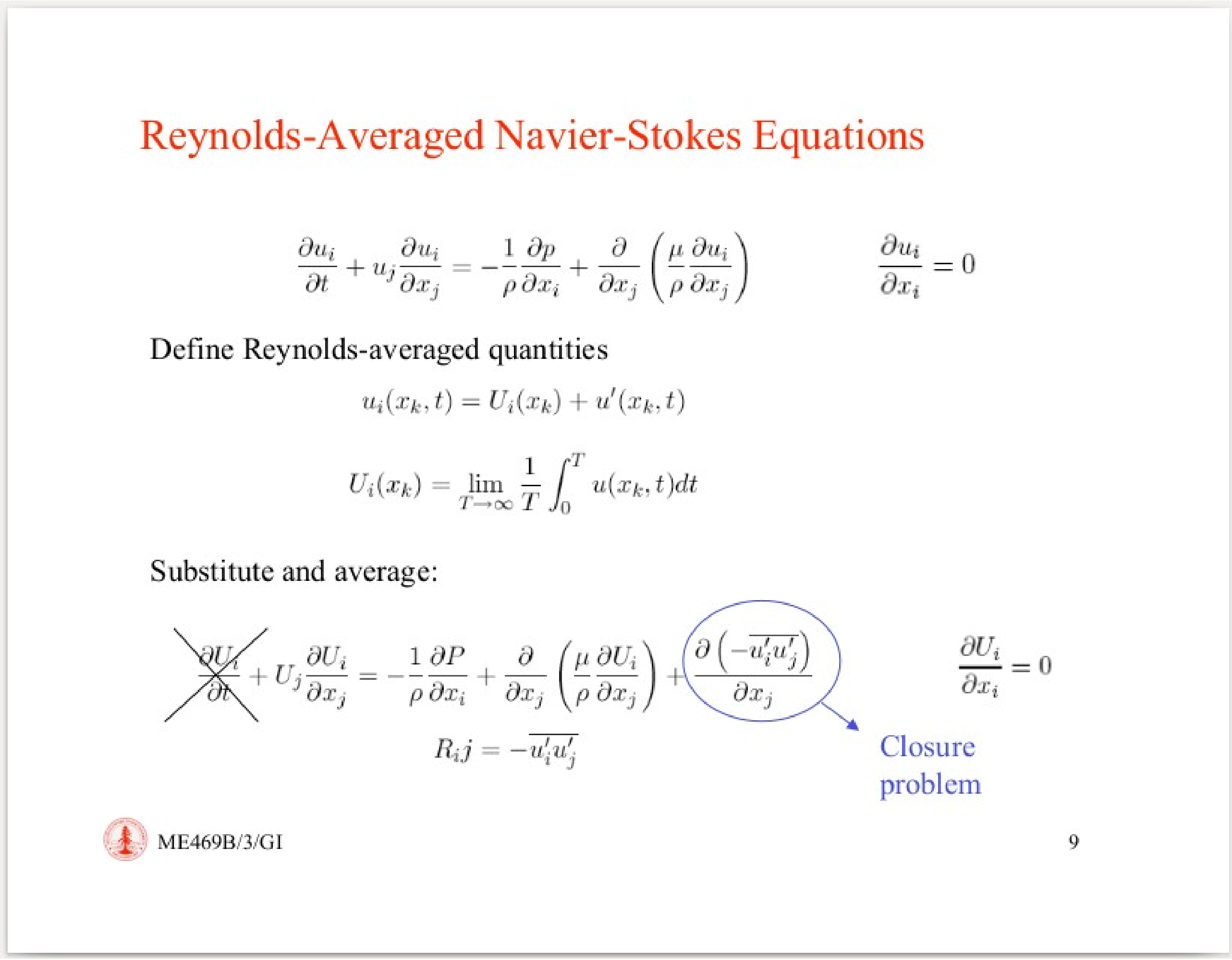
The effects of low and high frequencies disturbances for causing these steaks are theorized to have different from each other. The low frequencies disturbances cause the streaks due to their deep penetration property, while the high frequencies disturbances are heavily damped by the laminar shear layer, which is called as shear sheltering feature. This type of transition causes faster breakdown of the flow compared to the natural transition.

### 1.1.3 1.1.3. Separation-Induced transition

### 1.1.4 1.1.4. Wake-induced transition

## 1.2 Turbulence models

Several methods to calculate the flow behavior and laminar-turbulent transition are currently available. These include: (a) stability analysis for cases of low free flow turbulence, (b) low Reynolds number Reynolds-averaged Navier-Stokes (RANS) method.(c) direct numerical simulation (DNS), and (d) large eddy simulation (LES). In this project, we will run simulations over (a) flat-plates and (b) compressor-blade, using the RANS method, and then compare their results with (a) experimental data collected for the flat-plates and (b) data from DNS method respectively.




The Reynolds stresses are defined in the following ways:

1. Boussinesq Hypothesis.
2. Reynolds stress transport models.
3. Non-linear Eddy viscosity models (Algebraic Reynolds stress).
4. Models directly calculating the divergence of the Reynolds stresses.

The first- Boussinesq Hypothesis, and the third- Algebraic Reynolds stress, models are the stress models which are used in the BaseKOmega and transEARSM model respectively.

## Eddy viscosity models

Boussinesq relationship: $R_ij = -\overline{u_i' u_j'} = 2\frac{\mu_t}{\rho} S_{ij}$ with: $S_{ij} = \frac{1}{2}\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)$

$$\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\frac{(\mu + \mu_t)}{\rho}\frac{\partial U_i}{\partial x_j}\right]$$

$$R_{ij} = -\overline{\acute{u}_i \acute{u}_j} = 2\frac{\mu_t}{\rho} S_{ij}$$

where $S_{ij} = \frac{1}{2}\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)$ and $\mu_t = \frac{-1}{2}(\beta_1 + II_{\Omega}\beta_6) k\,\tau$

## 1.3 EARSM models compared to other RANS models

## 1.4 The extended EARSM Model with the transition model

The current model, in principal, consists of a modified k-Omega model with an algebraic transition model:

a. The pure k-Omega model has been modified by adding an extra tensor term called the an-isotropic stress term in the turbulent stress. The resultant model, after this stage, is known as Explicit Algebraic Reynolds Stress Model (EARSM).The extra term has a non-linear relation with the velocity, hence it has a higher computational demand.

b. The algebraic laminar-turbulent transition model used here is the latest version published by Dr Kubacki and Dr Dick in the reference [?]. This model considers an intermittency model which estimates the fraction of time a flow is turbulent opposed to laminar.

Governing Equations

averaged Navier-Stokes equation:

$$\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial(\bar{\rho}\tilde{u}_j)}{\partial x_j} = 0$$

$$\frac{\partial(\bar{\rho}\tilde{u}_i)}{\partial t} + \frac{\partial(\bar{\rho}\tilde{u}_i\tilde{u}_j)}{\partial x_j} + \frac{\partial(\bar{p})}{\partial x_i} = \frac{\partial}{\partial x_j}(\bar{\tau}_{ij} + \tau_{ij}^t)$$

$$\frac{\partial(\bar{\rho}\tilde{E})}{\partial t} + \frac{\partial}{\partial x_j}[(\bar{\rho}\tilde{E} + \bar{p})\tilde{u}_j] = \frac{\partial}{\partial x_j}[(\bar{\tau}_{ij} + \tau_{ij}^t)\tilde{u}_i] - \frac{\partial}{\partial x_j}(\bar{q}_j + q_j^t)$$

Here, the unknown variables are:

$\bar{\rho}$ - mean values of density
$\widetilde{u}_i$ - components of velocity vector
$\widetilde{E}$ - total specific energy
$\bar{p}$ - static pressure

The special symbols $\bar{\Phi}$ and $\widetilde{\Phi}$ means:

$\bar{\Phi}$ - indicates time average (Reynolds average)

$$\bar{\Phi}(\vec{x},t)=\lim_{T\to\infty}\frac{1}{T}\int_{t}^{t+T}\Phi(\vec{x},t)\,d\tau$$

$\widetilde{\Phi}$ - indicates density-weighted average (Favre average)

$$\widetilde{\Phi}=\frac{\overline{(\rho\Phi)}}{\bar{\rho}}$$

$\bar{\tau}_{ij}$ - mean stress tensor. For Newtonian fluids, the stress tensor components are:

$$\bar{\tau}_{ij}=\bar{\mu}\left(\frac{\partial\widetilde{u}_i}{\partial x_j}+\frac{\partial\widetilde{u}_j}{\partial x_i}-\frac{2}{3}\delta_{ij}\frac{\partial\widetilde{u}_k}{\partial x_k}\right)\quad,\quad\bar{\mu}=\mu_{ref}*\left(\frac{\partial\widetilde{u}_i}{\partial x_j}\right)^{\left(\frac{3}{4}\right)}$$

where $\bar{\mu}$ is the mean viscosity computed using the Rayleigh relation with constants $\bar{\mu}_{ref}$ , $\rho_{ref}$ , and $p_{ref}$ .

$\bar{q}_j$ and $q_j^t$ - components of the mean heat flux vector and the turbulent heat flux vector respectively.

$$\bar{q}_j=\frac{-\kappa}{\kappa-1}\frac{\bar{\mu}}{Pr}\frac{\partial}{\partial x_j}\left(\frac{\bar{p}}{\bar{\rho}}\right)\quad,\quad q_j^t=\frac{-\kappa}{\kappa-1}\frac{\bar{\mu}_t}{Pr_t}\frac{\partial}{\partial x_j}\left(\frac{\bar{p}}{\bar{\rho}}\right)$$

State equation of perfect gas

$$\bar{p}=(\kappa-1)\left[\bar{\rho}\widetilde{E}-\frac{1}{2}\bar{\rho}\widetilde{u}_j\widetilde{u}_j-\bar{\rho}k\right]$$

# 2. OpenFOAM implementation

## 2.1 Introduction to OpenFOAM

The implementation of the “EARSM+KD” algebraic transition model was carried out in OpenCFD distribution of OpenFOAM (v1812). OpenFOAM is an open source environment to perform the calculation for various fluid mechanics applications. The software is written in the C++ object-oriented programming language and includes a selection of multipurpose solvers and models designed for finite volume discretization.

The pre-processing of a case is done through text files located in a structured case directory. They serve as configuration files that provide the configuration for the solver. Once the calculation done by a chosen solver finishes, the results can be seen in external post-processing tools. However, the greatest advantage of using OpenFOAM software is the possibility of reprogramming existing tools, as well as incorporating user-defined functions and libraries in the basic program.

There are more than one ways of compiling and using an application, a solver or a library. The RANS model which we are writing will be compiled into a library.

The first and a popular way for implementing a model in OpenFOAM is to create a Make directory containing two files called ‘files’ and ‘options’ in the model directory. Files file contains name of the source code and location with user defined name of the library or an application (executable). The option file contains other dependency libraries which are required for compilation of the new model’s source code. This method is explained in OpenFOAM User Guide. Old version of OpenFOAM had a few additional steps for compiling and using a user-defined model which is explained on Hassan Kasem’s page. See Reference (?)

The second way of compiling a user defined RANS model is by copy-pasting a whole Class of all the turbulence model which is completely independent from other parts of OpenFOAM. This will define all default defined RANS model once again and then each case when run will use a model from user-defined model. In this method, there is no need of having a Make-directory containing ‘files’ and ‘option’ files in it. This method is defined by professor Jacek on the Chalmers University website. See Reference (?).

The calculation was performed using the simpleFoam solver using the Semi-Implicit Method for Pressure Linked Equations. Details on the development and implementation of custom tools in OpenFOAM have been included in Appendix A. The code for the implemented model can be found in Appendix C. A general description of the case configuration is provided in Appendix D.

## 2.2 Validation of the model

The validation of the implemented transitional model was carried out in two parts. First, it was tested for a series of three flat-plates with constant pressure gradient, namely flat-plates t3a, t3b, and t3am (read t3am as t3a_minus). The results were compared with experimental data from the ERCOFTAC database [???], as well as with simulation data obtained using the Kubacki-Dick transition model published by inz. Robert Tykocki. They served as a reference for checking the correct implementation of the model, as well as to determine its numerical properties. The model was then tested over V103A transitional compressor blade case. The results were compared with Direct Numerical Simulation (DNS) data obtained by Zaki and Durbin [???] and similarly to cases of flat plates with results obtained using the k-ω turbulence model. All the simulations performed in the project, the flat-plates and the compressor blade cascades are two-dimensional and in-compressible.

## 2.3 Transitional flat plate cases

The experimental ERCOFTAC database is one of the most detailed database of cases related to the interaction between turbulence generation and boundary layer transition characteristics.

The ERCOFTAC experimental setup is as follows, in general:

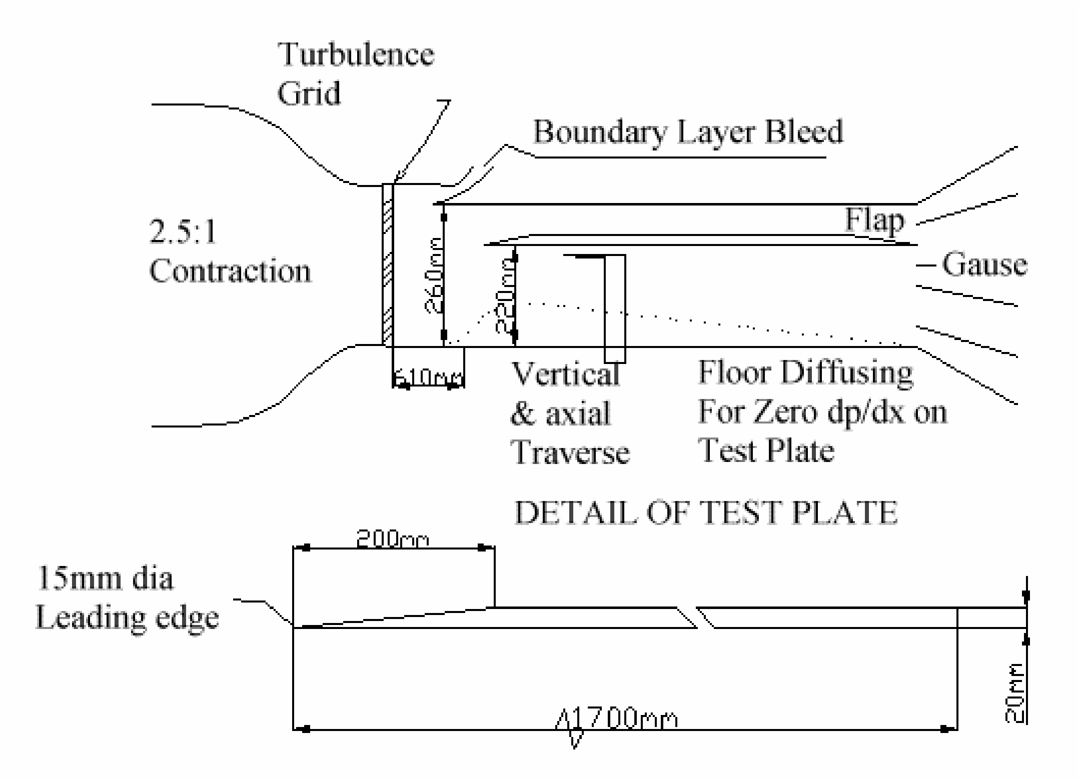



Figure 2.1: Setup of the transitional flat plate experiments

The detailed description of the case set up is presented in [????]. For an overview, the 1.7 m long plate is suspended and inclined at an angle of 0.5 degrees to ensure that the flow at the leading edge is 0.200 m long. The boundary layer in the upper wall of the main channel is separated by a bleed to minimize the effect of the upper part of the domain on the formation of the boundary layer on the upper surface of the plate. To ensure a zero pressure gradient along the test surface, the flat plate was fitted with a fin on the back edge, and the bottom wall tilted slightly. In addition, pressure gauges were mounted inside the working section to control the pressure drop. These measures also contributed to maintaining attachment on the trailing edge. The flow is generated by a centrifugal fan followed by a honeycomb grid to reduce turbulence. Turbulence generating meshes are mounted in front of the flat plate to obtain highly homogeneous and isotropic turbulent flows in the range of 0.5-7% turbulence intensity and 5-30 mm on a turbulent length scale.

X The measurements were obtained using hot wire anemometer systems and included both turbulent free flow properties and boundary layer velocity profiles. The collected data includes velocity profiles, three normal stress components and two shear stress components as wall shear stress which allows you to calculate the skin's friction coefficient.

The test case presented in this project are:

| NAME | Upstream Velocity (m/s) | Upstream Turbulence Intensity (%) | pressure Gradient |
|---|---|---|---|
| T3A | 5.4 | 3.0 | Zero |
| T3B | 9.4 | 6.0 | Zero |
| T3A- | 19.8 | 0.9 | Zero |

$ Notice that all these three cases have zero-pressure gradient. T3A and T3B have higher turbulent intensity but low inlet velocity while T3A- has low turbulent Intensity and high inlet velocity. The lower the background turbulent intensity of a flow is, the slower its process of natural transition will be.

### 2.3.1 Geometry and Mesh

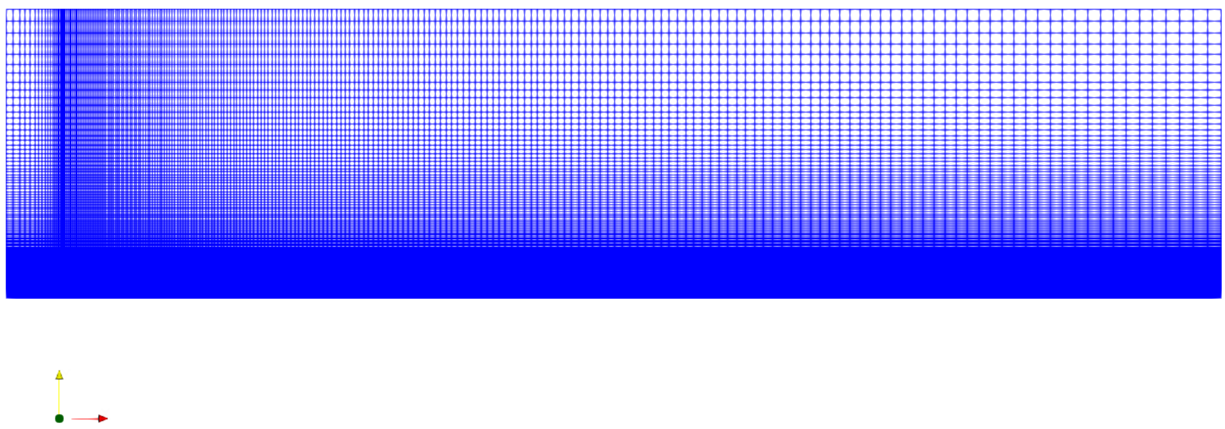


Figure. 2.2: Computation domain for flat plate, number of cells are different for 3 different cases.

### 2.3.2 Case Setup

$ The input parameters for all simulations with different RANS models depends on the requirements of the RANS model used in a particular case. In this project, both RANS models used are extended versions of k-omega model. These models needs initial conditions for the parameters k, nut, omega, p, and U. The following table summarizes the initial boundary conditions for the three flat plate cases:

| | k | nut | omega | p | U |
|---|---|---|---|---|---|
| Inlet | fixedValue | calculated | fixedValue | zeroGradient | fixedValue |
| outlet | zeroGradient | calculated | zeroGradient | fixedValue | zeroGradient |
| plate | noSlip | calculated | omegaWallFunction | zeroGradient | noslip |
| above | slip | calculated | zeroGradient | zeroGradient | slip |
| top | slip | calculated | zeroGradient | zeroGradient | slip |
| frontAndBack | empty | empty | empty | empty | empty |

In general, at inlet, for the turbulent Kinetic energy, specific dissipation rate, and velocity Dirichlet boundary conditions are used. Pressure is set to be at zero-gradient at the inlet for smooth transition. A corresponding boundary conditions are set for appropriate parameters at the outlet.

# 3. Report of Results

For every simulation, a turbulent-intensity-decay graph confirms a similar mimicking of presence of turbulent intensity in the vicinity of transition region. The experimental distribution of turbulent intensity is collected for the flat-plate cases from the ERCOFTAC website and collected for compressor-blades by reverse-extracting values from the report published by Rodi [?????].

Turbulent Intensity distribution is controlled by turbulent-intensity(k) and turbulent-dissipation-rate ( /omega) at the inlet of a flow. These parameters need an educated guess. For first interpolation, the following formulas are used:

## Turbulence kinetic energy

The turbulence kinetic energy, $k$, is the kinetic energy per unit mass of the turbulent fluctuations $u'_i$ in a turbulent flow. The SI unit of k is $J/kg = m^2/s^2$.

$$k \stackrel{\text{def}}{=} \frac{1}{2}\overline{u'_i u'_i} = \frac{1}{2}\left(\overline{u'^2_x} + \overline{u'^2_y} + \overline{u'^2_z}\right) = \frac{3}{2}\overline{u'^2}$$

# Turbulence intensity

Contents [hide]



## Definition

The turbulence intensity, also often refered to as turbulence level, is defined as:

$$I \equiv \frac{u'}{U},$$

where $u'$ is the root-mean-square of the turbulent velocity fluctuations and $U$ is the mean velocity (Reynolds averaged).

If the turbulent energy, $k$, is known $u'$ can be computed as:

$$u' \equiv \sqrt{\frac{1}{3}\left(u'^2_x + u'^2_y + u'^2_z\right)} = \sqrt{\frac{2}{3}k}$$

$U$ can be computed from the three mean velocity components $U_x$, $U_y$ and $U_z$ as:

$$U \equiv \sqrt{U_x^2 + U_y^2 + U_z^2}$$

An online tool for this process is available at reference [?????]
Next, the skin friction is calculated after extracting wall-shear-stress from the simulation.

## Skin friction coefficient [ edit ]

### Definition [ edit ]

$$C_f = \frac{\tau_w}{\frac{1}{2}\rho v^2}$$

where:

- $C_f$ is a skin friction coefficient.
- $\rho$ is the density of a fluid.
- $v$ is the free stream speed, which is the fluid speed far from the body's surface.
- $\tau_w$ is a skin shear stress on a surface.
- $\frac{1}{2}\rho v^2$ is the dynamic pressure of a free stream.

The skin friction coefficient is a dimensionless skin shear stress which is nondimensionalized by the dynamic pressure of a free stream.

All simulations, all flat-plate cases and compressor-blade cases, assumes the fluid density as 1.0. This is the default value in OpenFOAM.

## 3.1 Flat Plate Cases

### 3.1.1 t3a

P. Experimental data from the ERCOFTAC website suggests that t3a-case shows a rapid bypass transition. For this case, the turbulent-intensity at the leading edge of the flat-plate is 3.0% and velocity 5.4 m/s, which shows moderately high level of free-stream turbulence. The turbulent intensity decay graph is controlled with the the following two parameters:

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       volScalarField;
    location    "0";
    object      k;
}

dimensions      [0 2 -2 0 0 0 0];

internalField   uniform 0.05;


// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //
FoamFile
{
    version     2.0;
    format      ascii;
    class       volScalarField;
    location    "0";
    object      omega;
}
dimensions      [0 0 -1 0 0 0 0];
internalField   uniform 435;


// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //
```

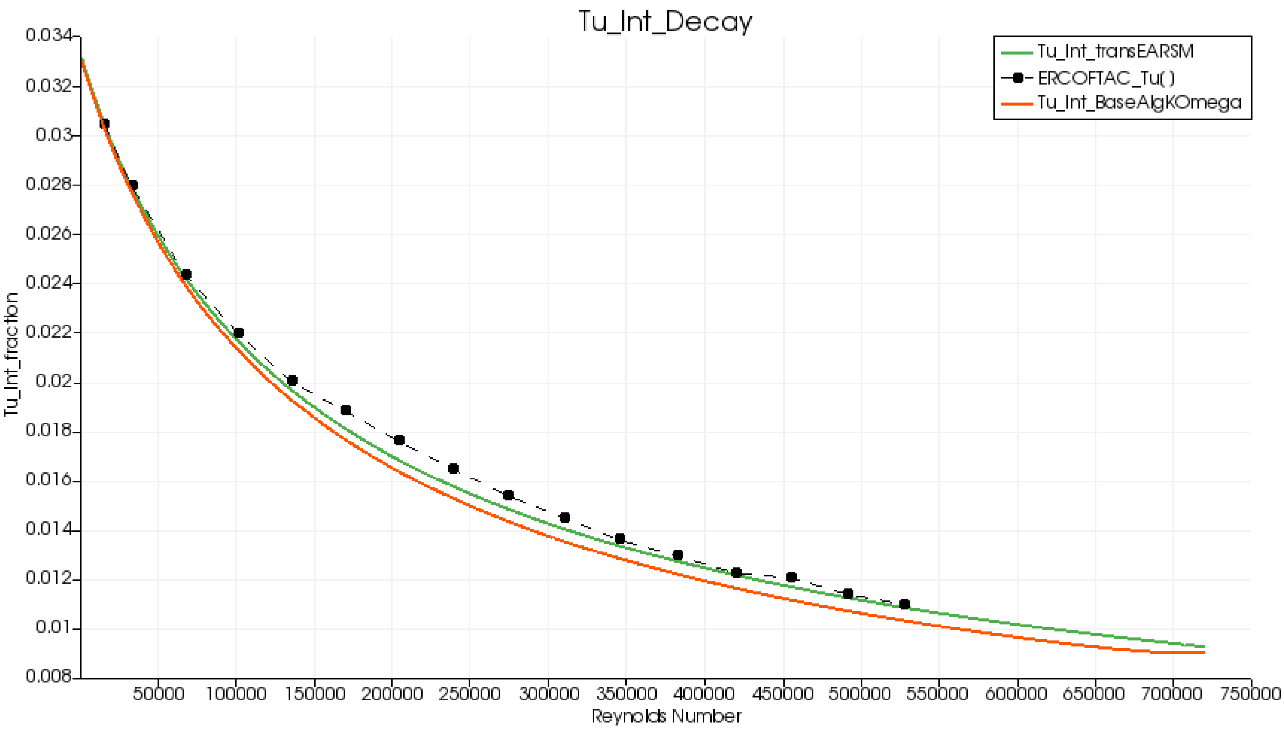



Fig.3.1 __t3a__Distribution of turbulent Intensity at an estimated boundary layer of flat-plate.

The above graphs, Fig. 3.1 ,shows that the simulation is conducted under the similar condition as that of the ERCOFTAC experiment. Next, the Fig. 3.2 (both a, and b) shows distribution of skin-friction over the surface of flat-plate t3a case. The figures Fig.3.2.a and Fig.3.2.b represents the same result, but the difference is in their abscissa. Over their abscissas, Fig.3.2.a takes a linear distance, while Fig.3.2.b takes Reynolds number with upstream of the flat-plate.

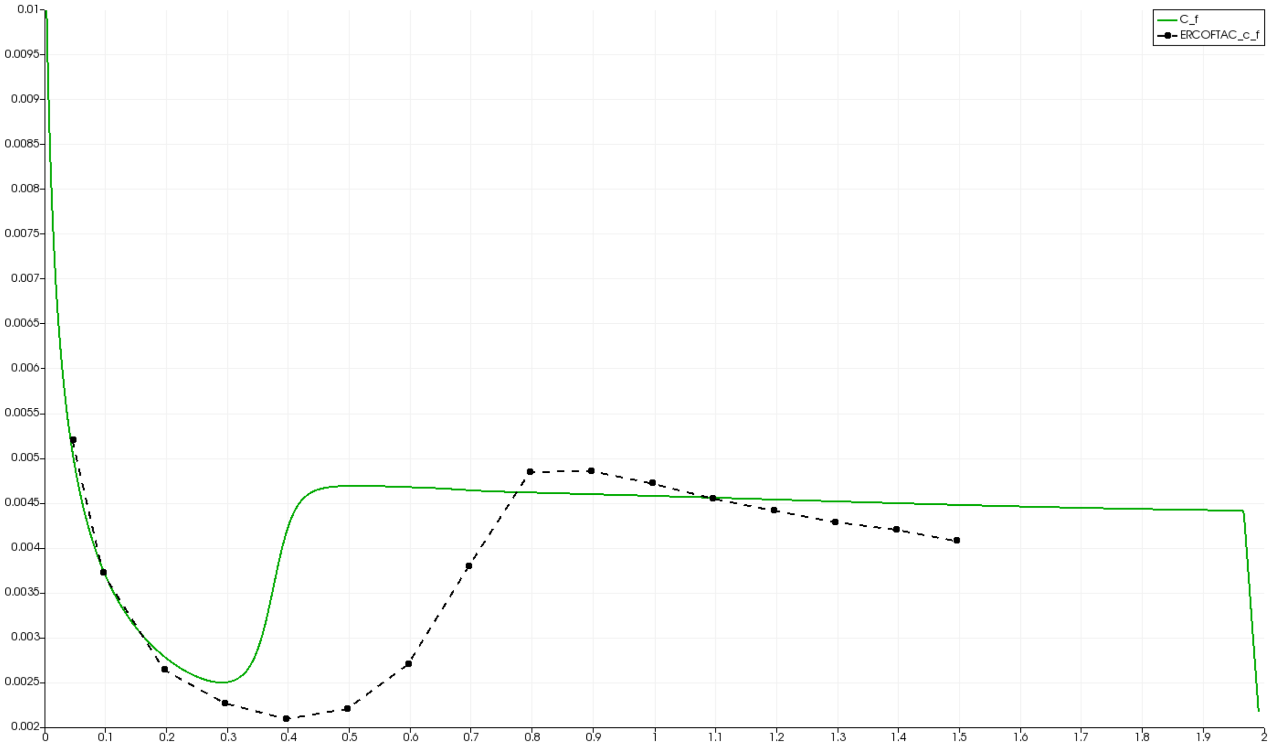



Fig.3.2.a __t3a__Distribution of Skin-friction Coefficient over t3a flat-plate over linear distance over the plate

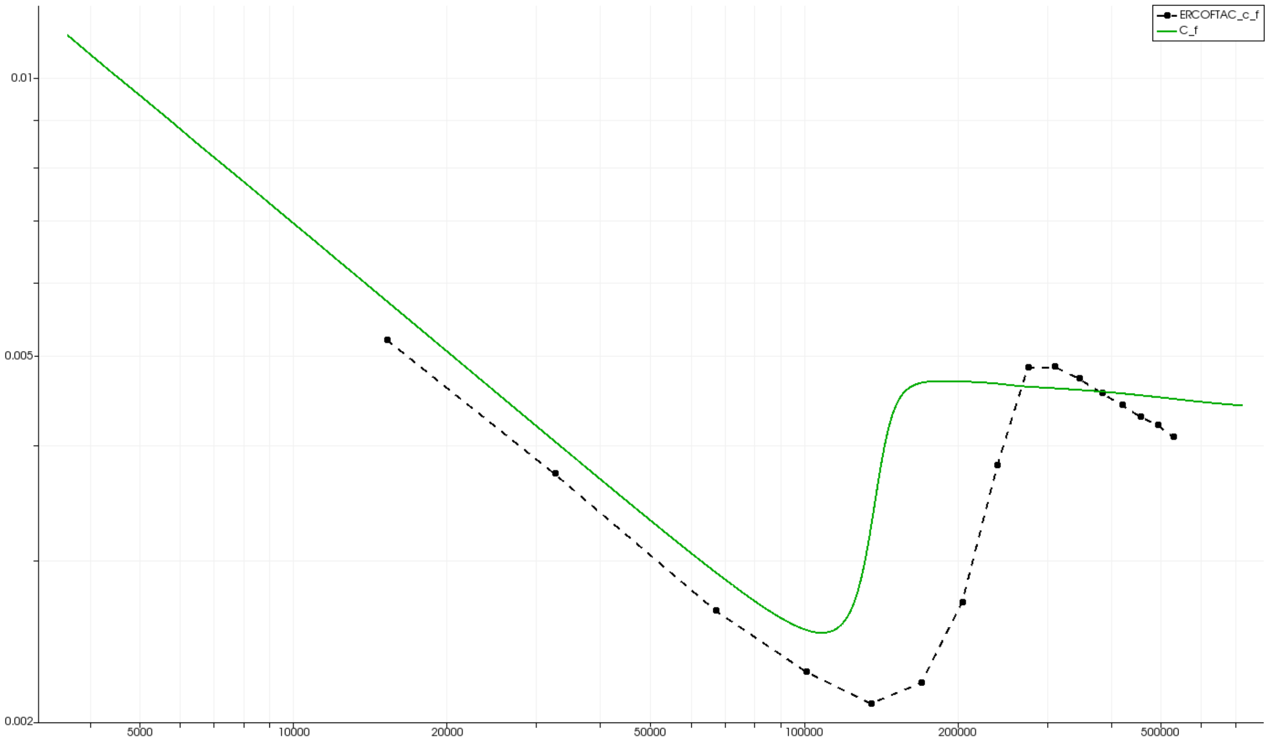



Fig.3.2.b__t3a__Distribution of Skin-friction Coefficient over t3a flat-plate over Reynolds number

In the above graph, the ERCOFTAC data shows that the laminar to turbulent flow transition starts on the 0.4m on the flat-plate in t3a case. The result from simulations shows that the BaseKOmega model predicts an early transition. Secondly, this model also predicts wall-shear-stress to remain constant after transition has happened, which is unlike the ERCOFTAC data. A better result from simulation should be where the wall-shear-stress coefficient follows the ERCOFTAC data and asymptotically converges with it. <write also about EARSMTrans model>

### 3.1.2 t3b flat-plate

Similar to the t3a case, the initial conditions for the t3b case is also bypass-transition.

```
     NAME                Upstream          Upstream           pressure
                         Velocity         Turbulence          Gradient
                           (m/s)         Intensity (%)

     T3A                   5.4               3.0                  Zero
     T3B                   9.4               6.0                  Zero
```

Comparing the initial conditions for the t3b-case against the t3a-case, we notice that it has higher inlet velocity and turbulent intensity at the tip of plate. The following initial conditions for turbulent intensity and specific dissipation rate (\omega) is considered for matching the following decay graph to match the ERCOFTAC conditions.

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       volScalarField;
    location    "0";
    object      k;
}
dimensions      [0 2 -2 0 0 0 0];
internalField   uniform 0.8;

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

FoamFile
{
    version     2.0;
    format      ascii;
    class       volScalarField;
    location    "0";
    object      omega;
}
dimensions      [0 0 -1 0 0 0 0];
internalField   uniform 504.1;

// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //
```

A comparison of turbulent intensity of two models with that of the ERCOFTAC data is presented in Fig. Fig.3.3.


Fig3.3 _t3b_Distribution of turbulent Intensity at an estimated boundary layer of flat-plate.

The next graph, Fig.3.4 (a, and b), shows comparison of skin-friction coefficients against that from the ERCOFTAC data. In this case, the ERCOFTAC data shows that these conditions cause a faster transition from laminar to turbulent flow in its boundary layer. This transition takes place at 0.1m over the flat-plate in the t3b-case while it happened on 0.4m in the t3a-case. By comparing skin-friction coefficient of the two models against the ERCOFTAC data, we can see that both the models predicts a laminar boundary layer in the beginning but undergoes rapid transition to a turbulent flow. The algebraic model, transEARSM, is able to capture the laminar part in more detail. After the models predict the boundary layer has turned turbulent, they both matches the skin-friction coefficient asymptotically in the later part.

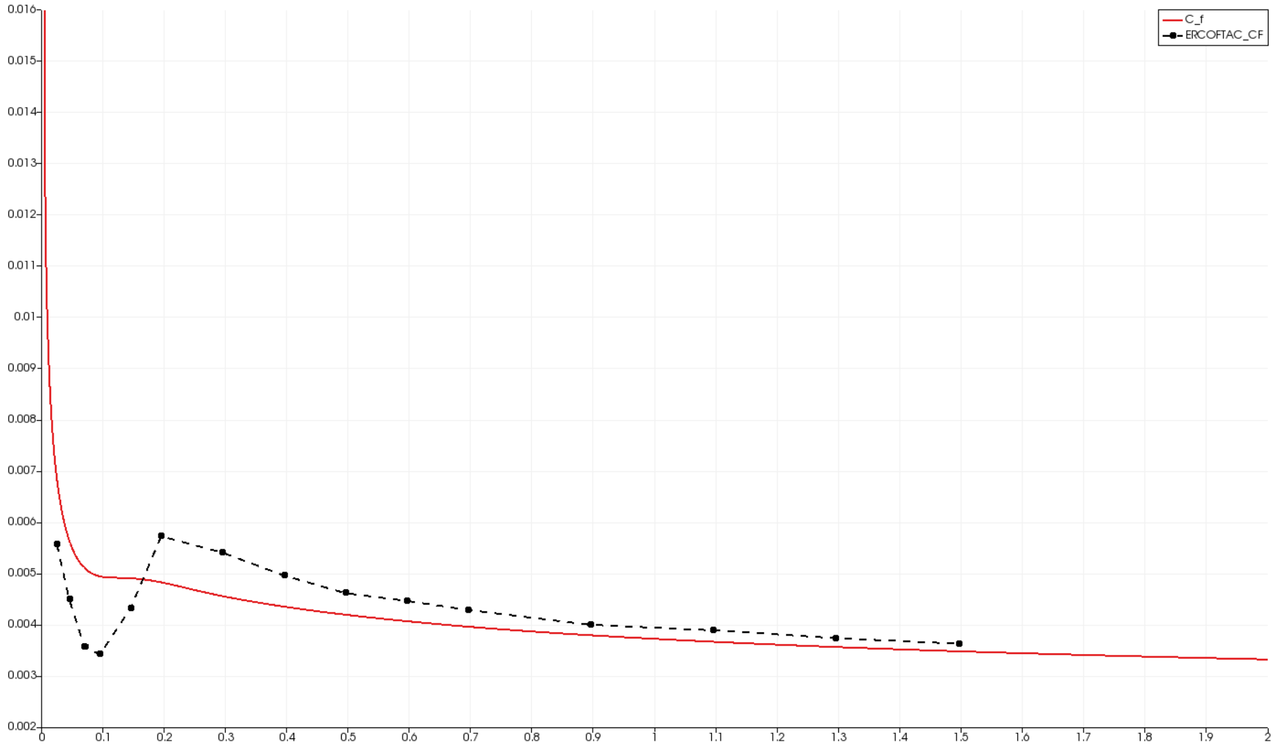



Fig3.4.a _t3b_Distribution of Skin-friction Coefficient over t3a flat-plate over linear distance over the plate

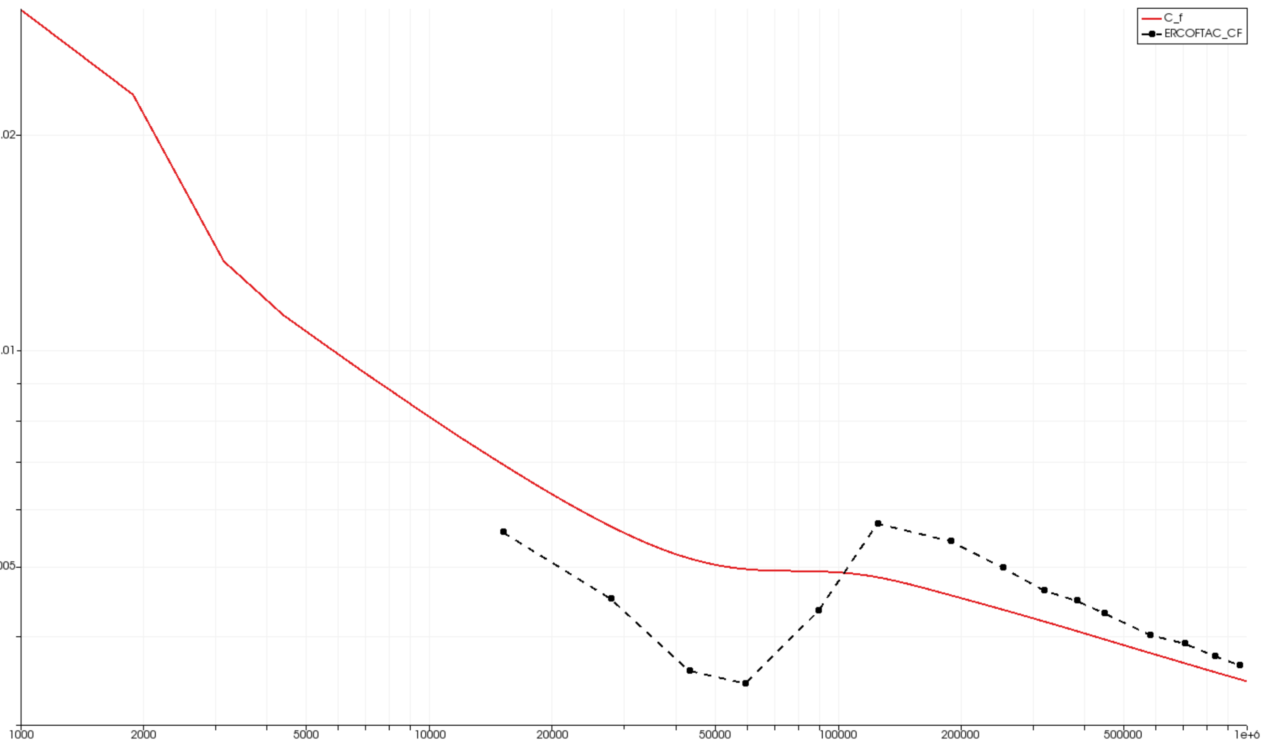



Fig3.4.b _t3b_Distribution of Skin-friction Coefficient over t3a flat-plate over Reynolds number

### 3.1.3 t3a- flat-plate

Unlike to the previous two cases, t3a and t3b, t3a- exhibits very different behavior. In this case, turbulent intensity is substantially lower and the initial velocity at tip of the late is higher. The ERCOFTAC experiment shows that this case has a natural transition of boundary layer.

| NAME | Upstream Velocity (m/s) | Upstream Turbulence Intensity (%) | pressure Gradient |
|---|---|---|---|
| T3A | 5.4 | 3.0 | Zero |
| T3B | 9.4 | 6.0 | Zero |
| T3A- | 19.8 | 0.9 | Zero |

The following initial conditions for turbulent intensity and specific dissipation rate (\omega) is considered for matching the following decay graph to match the ERCOFTAC conditions.

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       volScalarField;
    location    "0";
    object      k;
}
dimensions      [0 2 -2 0 0 0 0];
internalField   uniform 0.06;

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

FoamFile
{
    version     2.0;
    format      ascii;
    class       volScalarField;
    location    "0";
    object      omega;
}
dimensions      [0 0 -1 0 0 0 0];
internalField   uniform 418.7;

// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //
```

A comparison of turbulent intensity of two models with that of the ERCOFTAC data is presented in Fig. Fig.3.5.

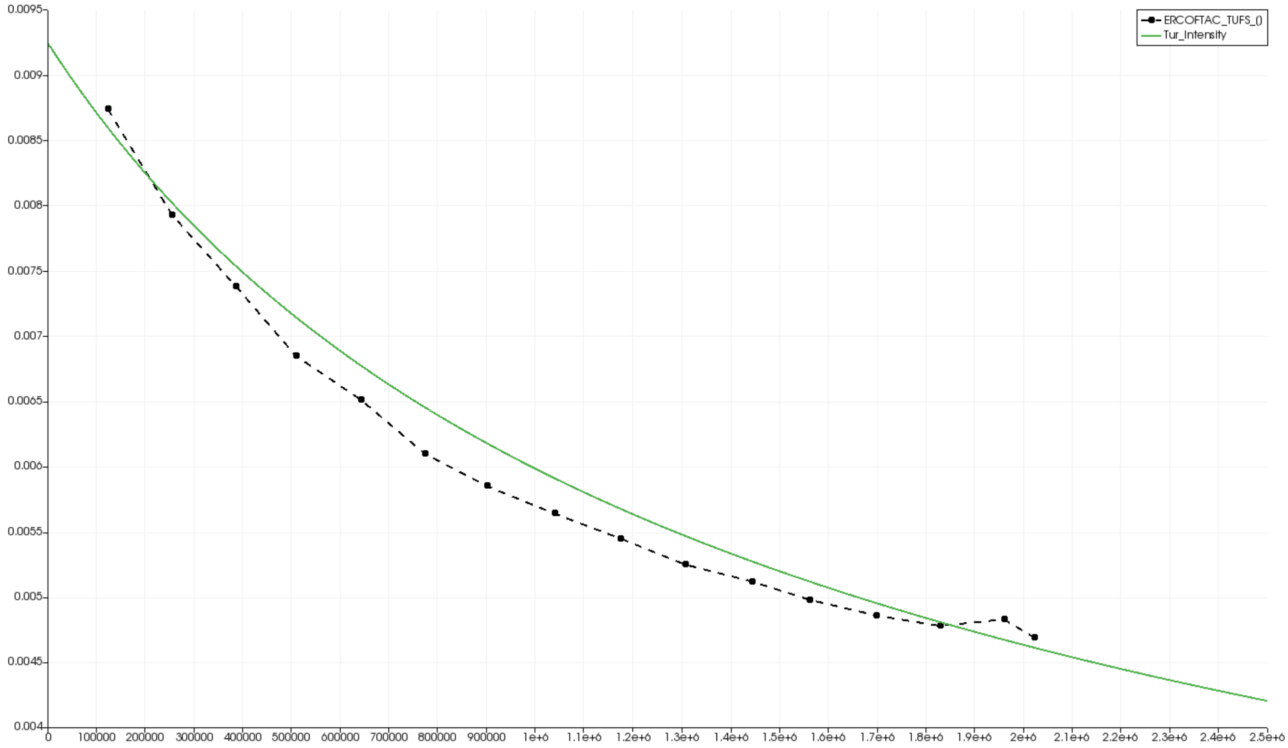



Fig3.5_t3am_Distribution of turbulent Intensity at an estimated boundary layer of flat-plate.

The next graph, Fig.3.6 (a, and b), shows comparison of skin-friction coefficients from the two models against that from the ERCOFTAC data. In this case, the ERCOFTAC data shows that the natural transition takes place at the rear end of the flat-plate in t3am case. Both the models show early onset of transition but the transEARSM model is able to better capture the laminar part and predicts a closure location of transaction. Given that the BaseKOmega model is developed to capture the bypass and separation-induced transition and not natural transition, its result is acceptable. This case exhibits that the transEARSM model is more accurate than the BaseKOmega model.

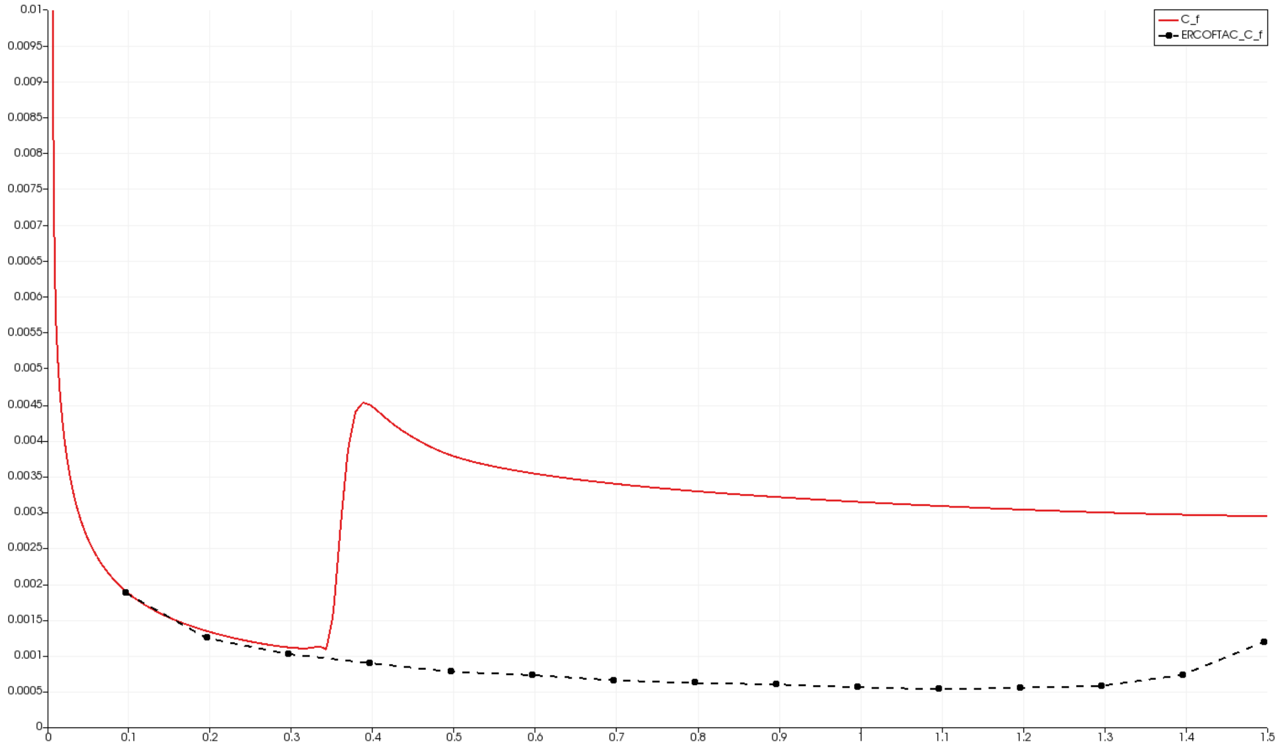



Fig3.6.a _t3am_Distribution of Skin-friction Coefficient over t3a flat-plate over linear distance over the plate

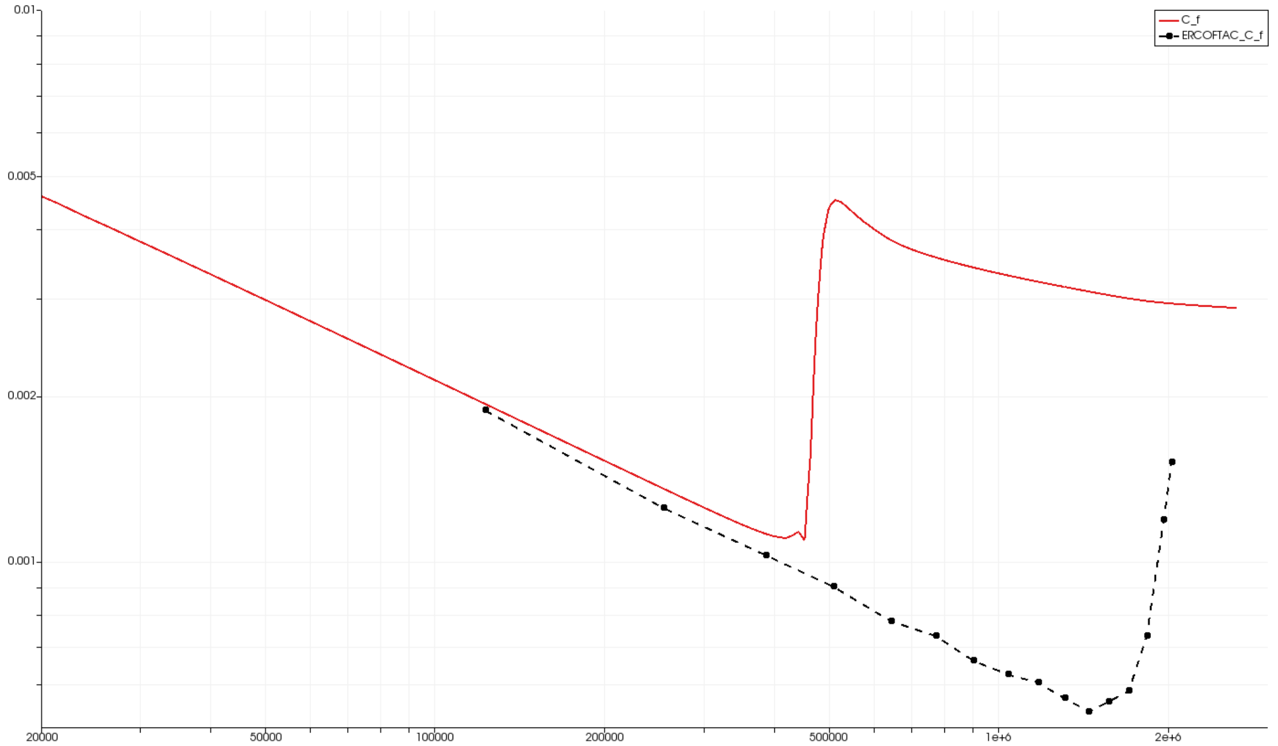



Fig3.6.b _t3b_Distribution of Skin-friction Coefficient over t3a flat-plate over Reynolds number

## 3.2 Compressor Blade Cascades

The following two cases of compressor blade are evolved from the reference [???] where Direct Numerical simulation of transition in a compressor cascade was conducted to study the influence of the free-stream turbulence.

The experiment was conducted in the University of the Armed forces in Munich, (Hilgenfeld & Pfitzner 2004). Fig.3.7 presents a schematic diagram of the mesh used in the following simulation where the two models, transEARSM and BaseKOmega, are tested. This mesh uses a heavily modified version of NACA 65 airfoil. The linear configuration of the blade in the cascade was called V103 in that experiment, hence the name in this project is the same. The experimental data is published in the report published my Dr Zaki and it needs to be extracted in digital form for further use in this project.

Notably, in the mesh used in OpenFOAM, Fig.3.8, only one channel from the Fig.3.7 is used. Over this mesh, a linear-periodic boundary conditions in the y-direction is imposed. The mesh in OpenFOAM was imported from a fluent Mesh provided by Dr Kubacki, using an inbuilt tool OpenFOAM, called “fluent3DMeshToFoam”. An important step to extract this fluent mesh into OpenFOAM case was to resolve internal-wall defined in Fluent. The Fluent software provides an option to its user where they can define a wall as type “internal” which makes the wall as non-existent for the flow. This “internal-type wall” can be used to define geometry and to define mesh but it will not interfere with the flow. This feature is not defined in OpenFOAM and hence needs to be resolved. A work-around for this is to split the “internal-wall” into two different walls (so the the mesh is split into two parts in the vicinity of the wall) and then stitch this detached-mesh together. The two inbuilt function of OpenFOAM used  for this work-around are “splitMesh” and “stitchMesh”:

```
splitMesh <name_of_internal> <name_spllitWallNew_1> <name_splitWallNew_2>

stitchMesh -overwrite -perfect <master_patch> <slave_patch>
```

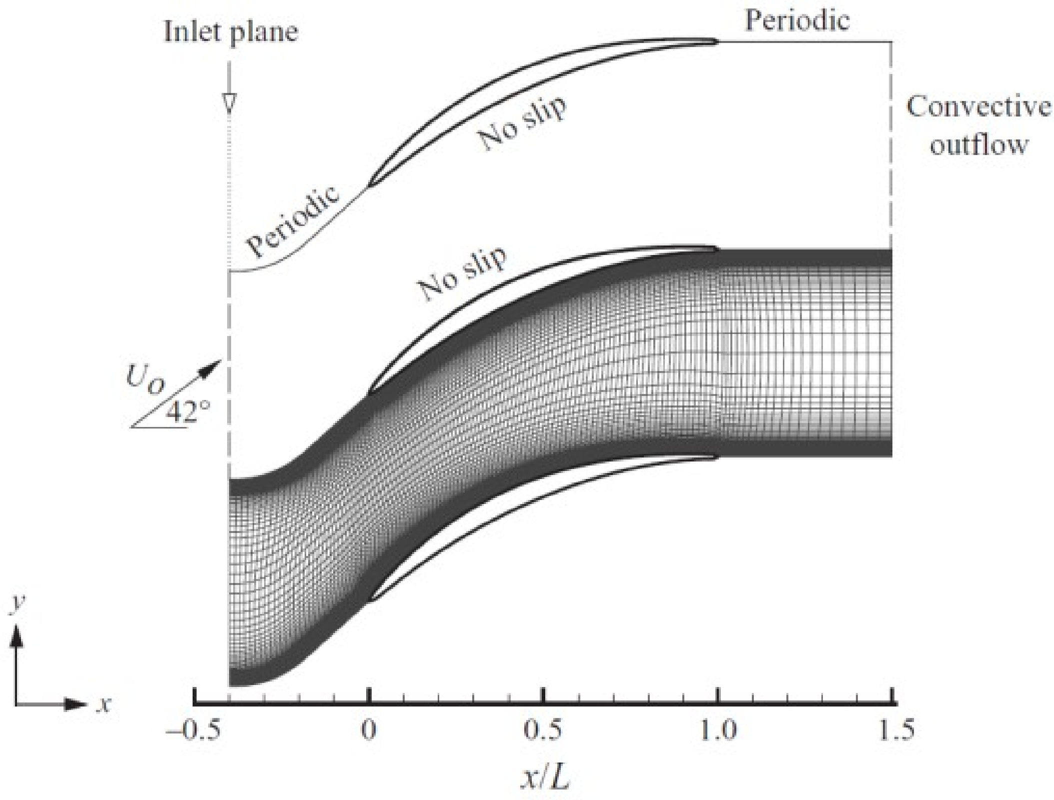



Fig.3.7__V103 Compressor Blade Cascade

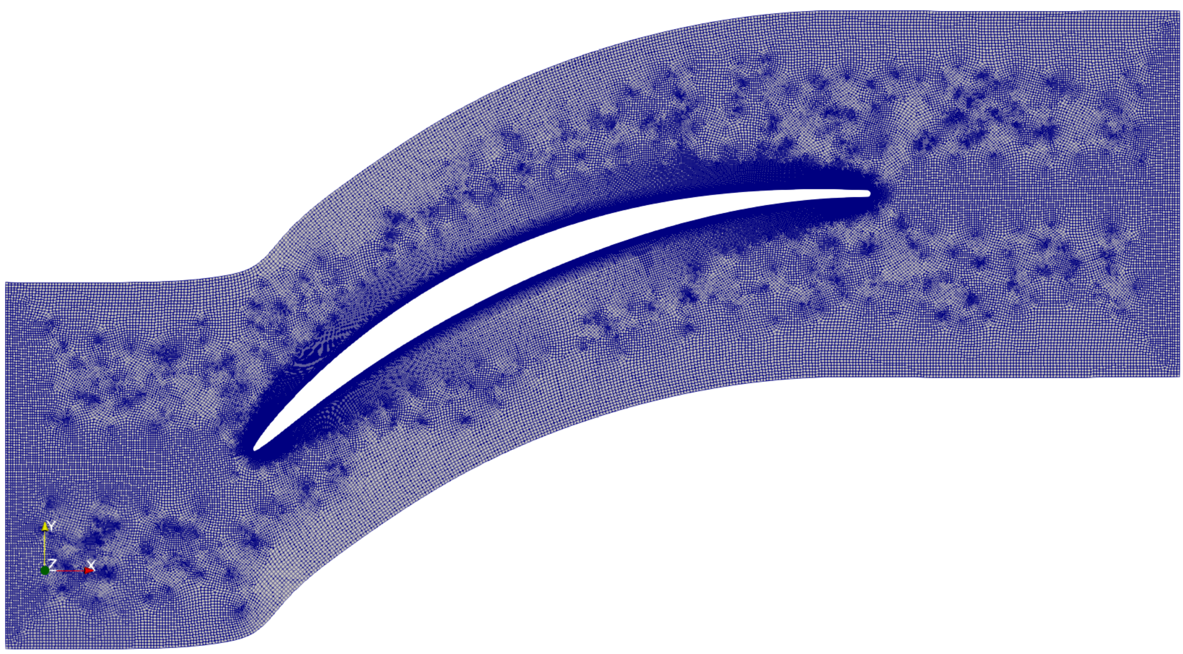


Fig.3.8__mesh for V103 Compressor Blade Cascade, used in OpenFOAM

The compressor blade provides a more realistic case than flat-plates, over which transition mechanism is tested. On the suction surface (concave-side) of the blade, it exhibits more detailed and complicated mechanism of transition of flow since Klebanoff instabilities are accompanied by Kelvin-Helmholtz instabilities. On this surface, a boundary layer separation appears which generates more shear stress which in turn cause the laminar-to-turbulent transition.

### 3.2.1 V103A_325 Compressor Blade Cascade

The first case in this project corresponds to “T1-case” from the publication of Dr Zaki ref.[???], where the turbulent intensity is 3.25% at the inlet. Fig.3.9 reflects how the current models are configured to run at turbulent intensity and specific dissipation rate against the data extracted from the DNS. All these data correspond to the mid-pitch of two compressor-blades.

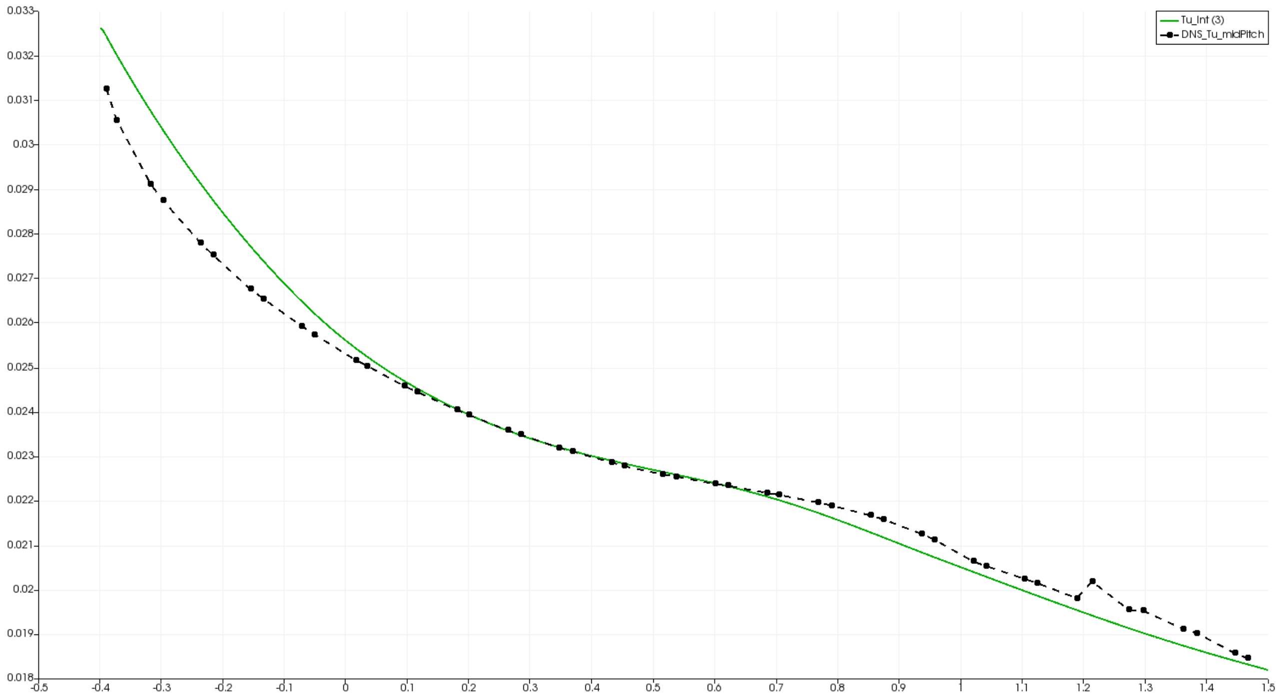



Fig.3.9__Turbulent Intensity decay at mid-pitch of two compressor blades.

The next figure, Fig.3.10, shows the comparison of skin-friction-coefficient on the suction surface of the compressor blade. The DNS’s data shows that there is a separation bubble on the suction surface where the skin-friction-coefficient drops below zero because the flow is reversed in this bubble. Unlike the DNS’s data, none of the two models is able to capture this separation bubble.

The physics of this separation-induced transition is addressed in the P_sep parameter of the BaseKOmega model. This term directly contributes in the production of turbulence in separation region. The effect of this term can be seen in the contour plot of the turbulent kinetic energy (k) in the Fig.3.11.

The next model, transEARSM

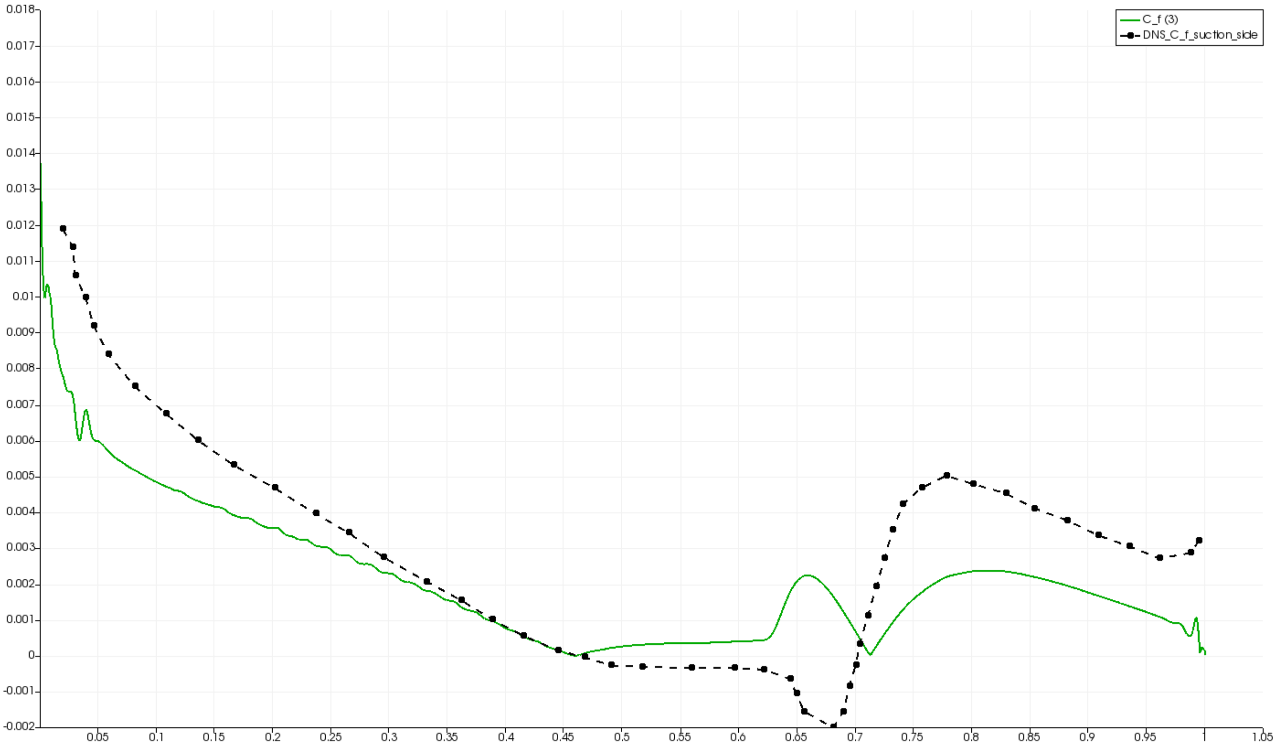



Fig.3.10:: V103_325:: C_F on the Suction side

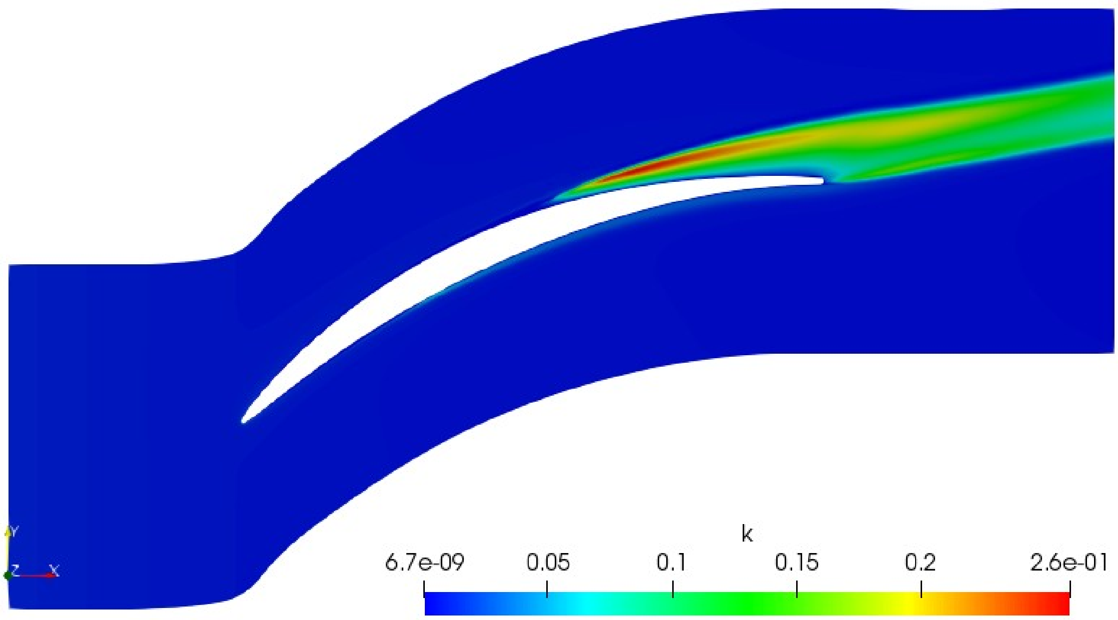



Fig.3.11:: V103_325:: turbulent kinetic energy field (BaseKOmega)

X The process of separation-induced transition is clearly visible in Figure 3.8 which shows the distribution of the skin friction coefficient on the suction surface of the compressor blade - transition is preceded by a drop of the skin friction coefficient to negative values. This mechanism of transition is modeled using the P_.sep term whose contribution to the production of turbulence is visible in Figure 3.9 which displays the turbulence kinetic energy field of the transition model simulation with an enlarged view of the separation region placed on the suction side of the compressor blade. As visible, the k-ω turbulence model fails to capture separation - the skin fricition coefficient's values remain positive throughout the length of the blade. This is also visible in Figure 3.10 which shows the turbulence kinetic energy field - a fully turbulent boundary layer is modeled on both sides of the compressor blade.

### 3.2.2 V103A_650 Compressor Blade Cascade

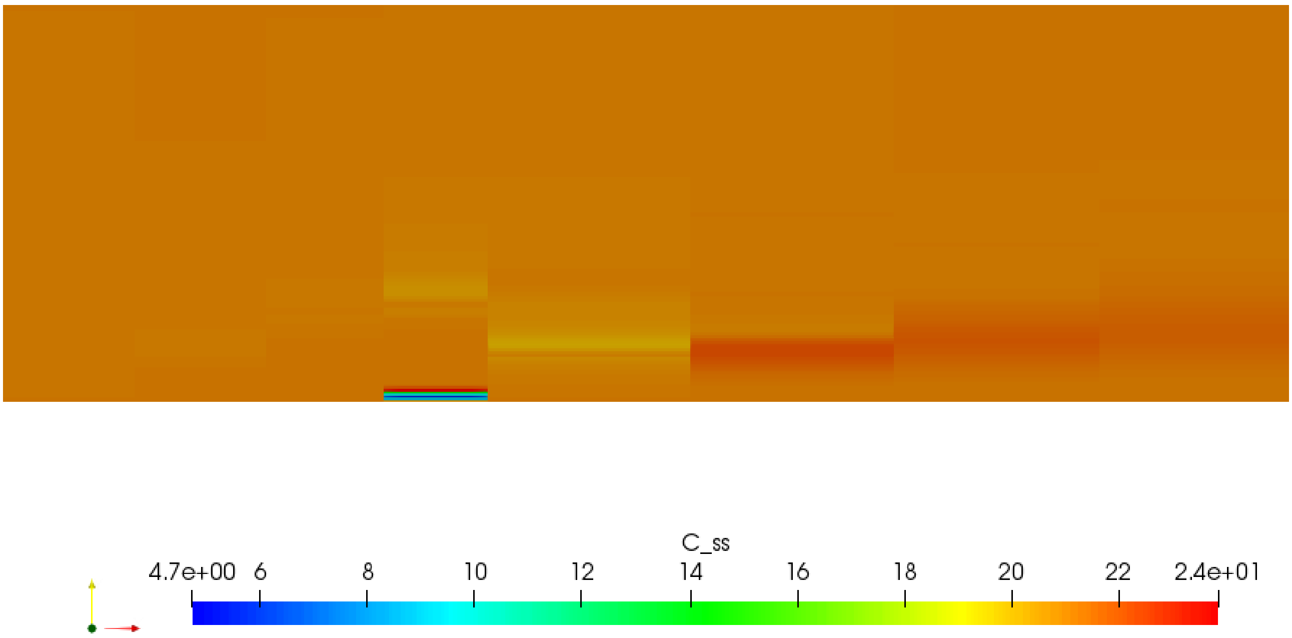
Ldkfh

# 4. Model Parameters

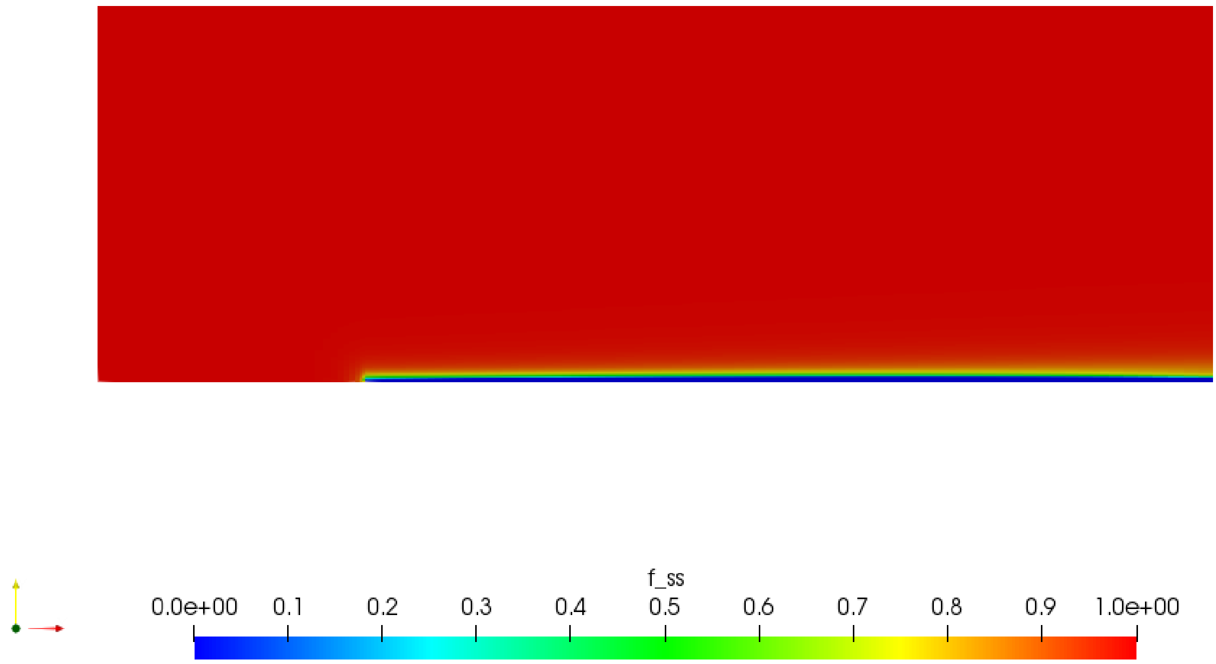
The BaseKOmega model is an extension of the standard K-Omega model. It uses the standard K-Omega model and adds Kubacki-Dick model (KD model) for laminar to turbulent boundary layer transition. This KD model has some parameters which are of special interest as they are critical for the transition modeling. Contour Plot of such parameters are attached in the following section. For studying effect of these parameters, only t3a flat-plate case and V103_325 compressor blade cascade case are considered.

## 4.1 t3a flat-plate:: Contour Plots of some parameters of the BaseAlgKOmega model

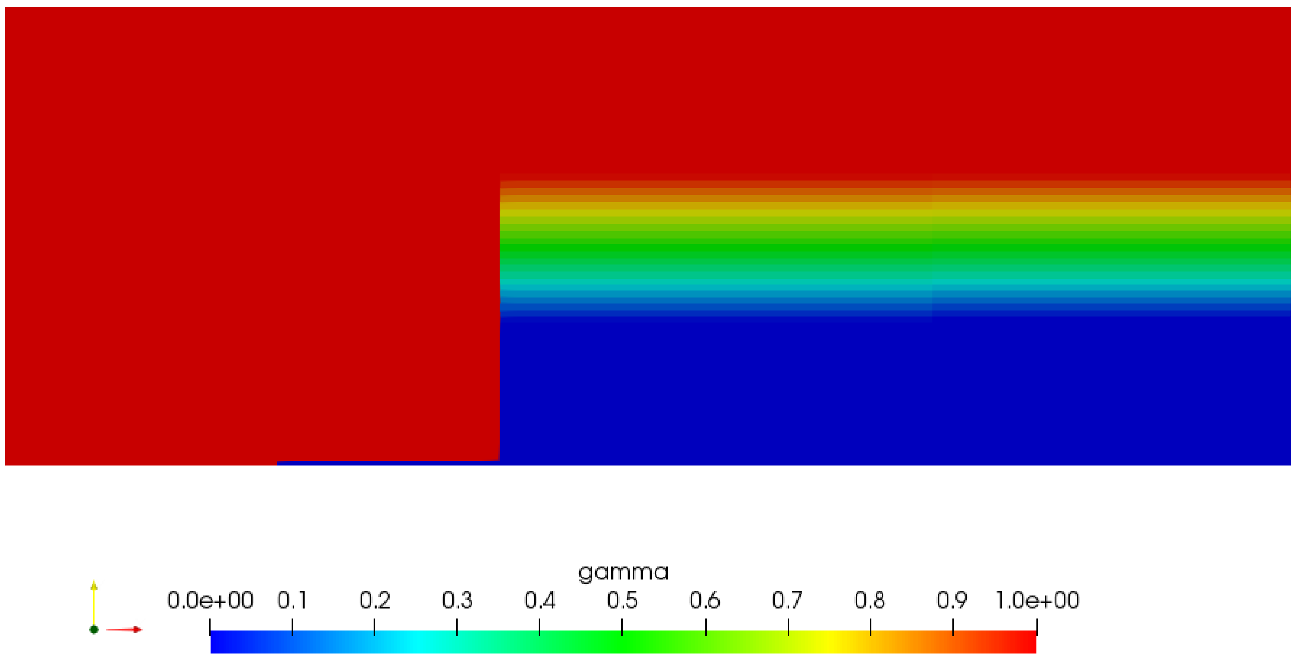
### 4.1.1 C_ss



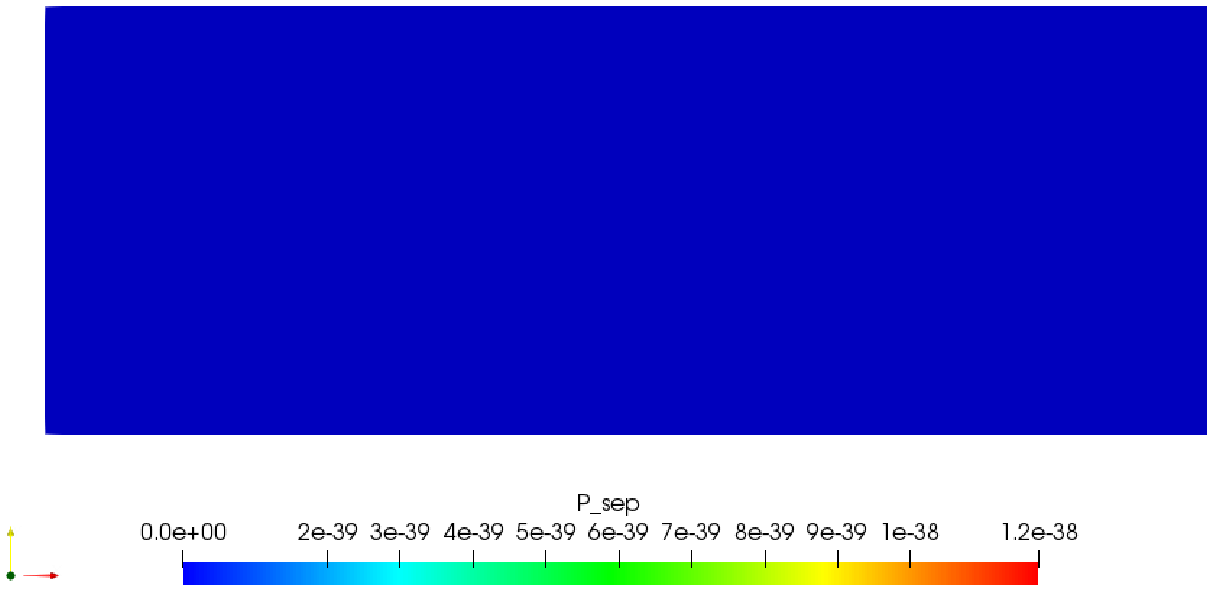
### 4.1.2 f_ss



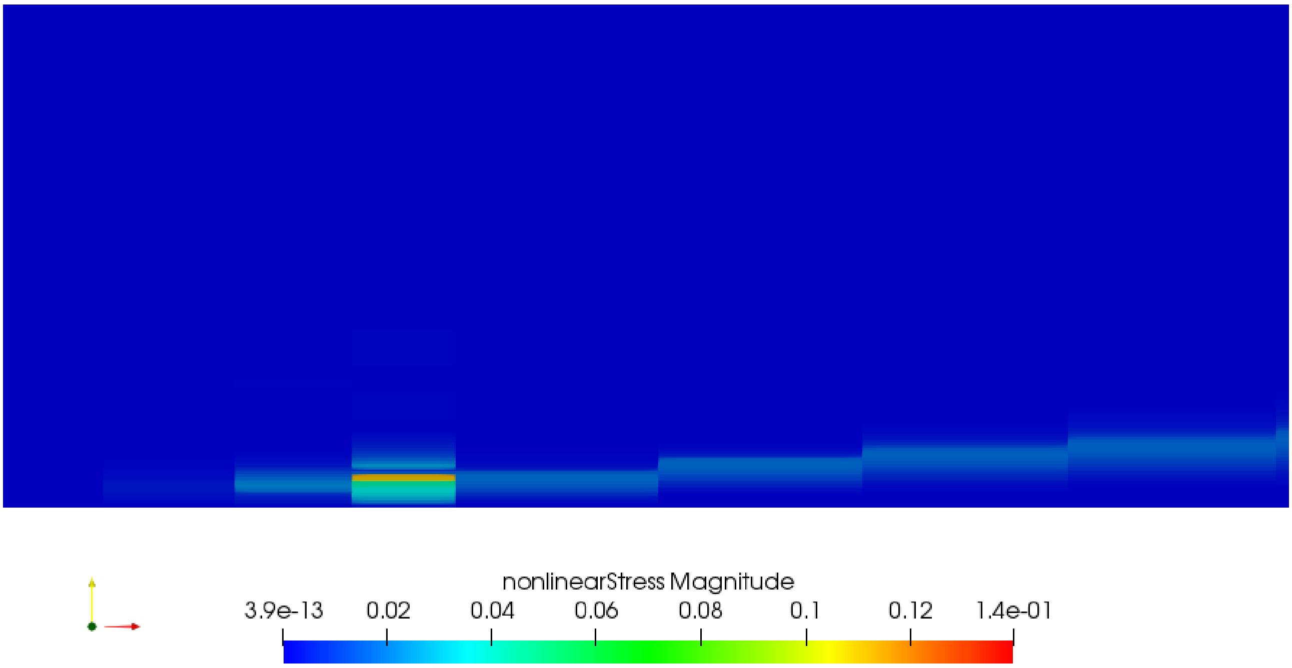

### 4.1.3 gamma

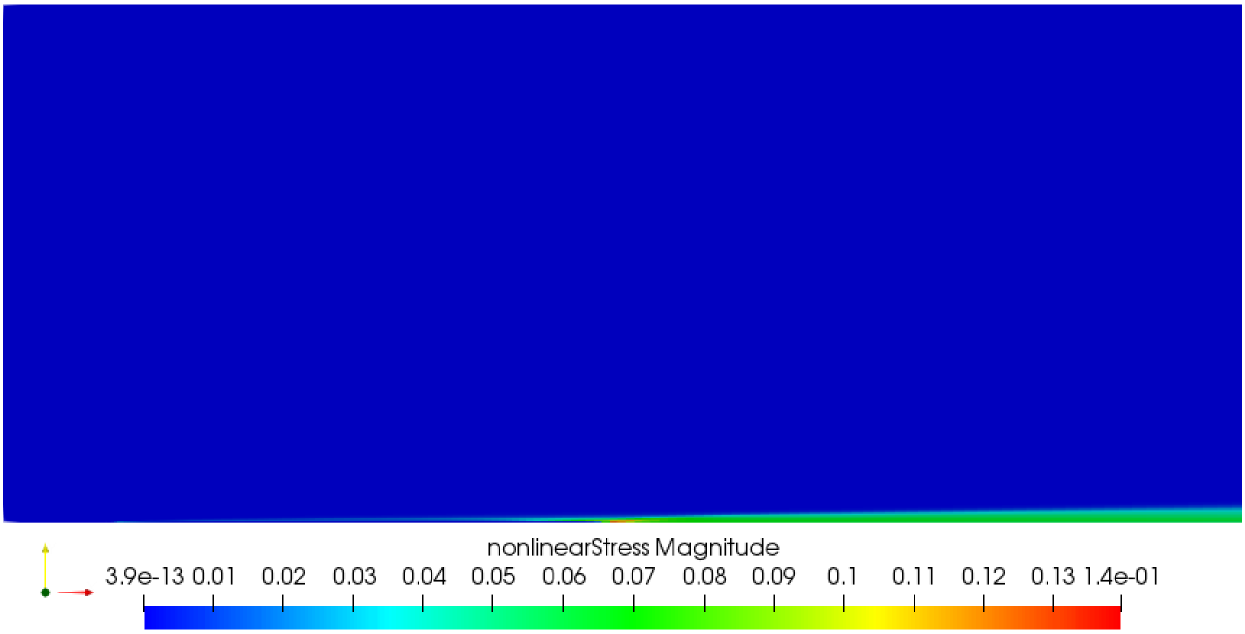


### 4.1.4 P_sep

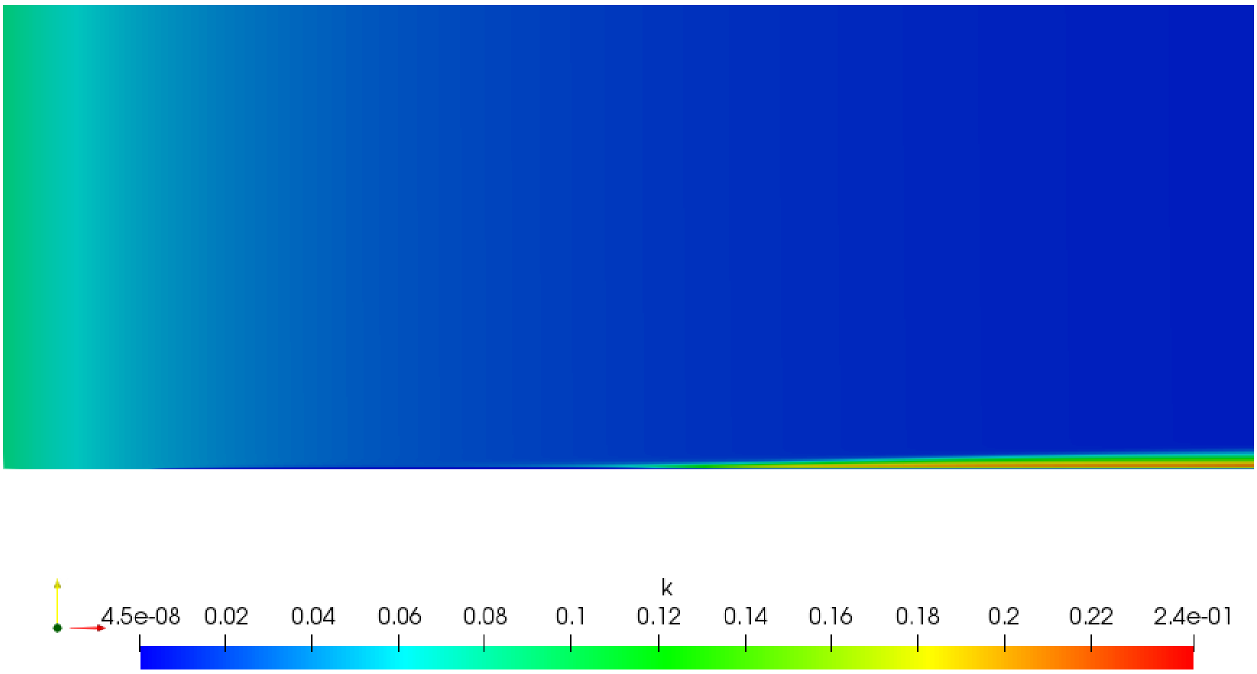

P_sep
0.0e+00
2e-39
3e-39
4e-39
5e-39
6e-39
7e-39
8e-39
9e-39
1e-38
1.2e-38

## 4.2 t3a flat-plate:: Contour Plots of the extra non-linear term of EARSM



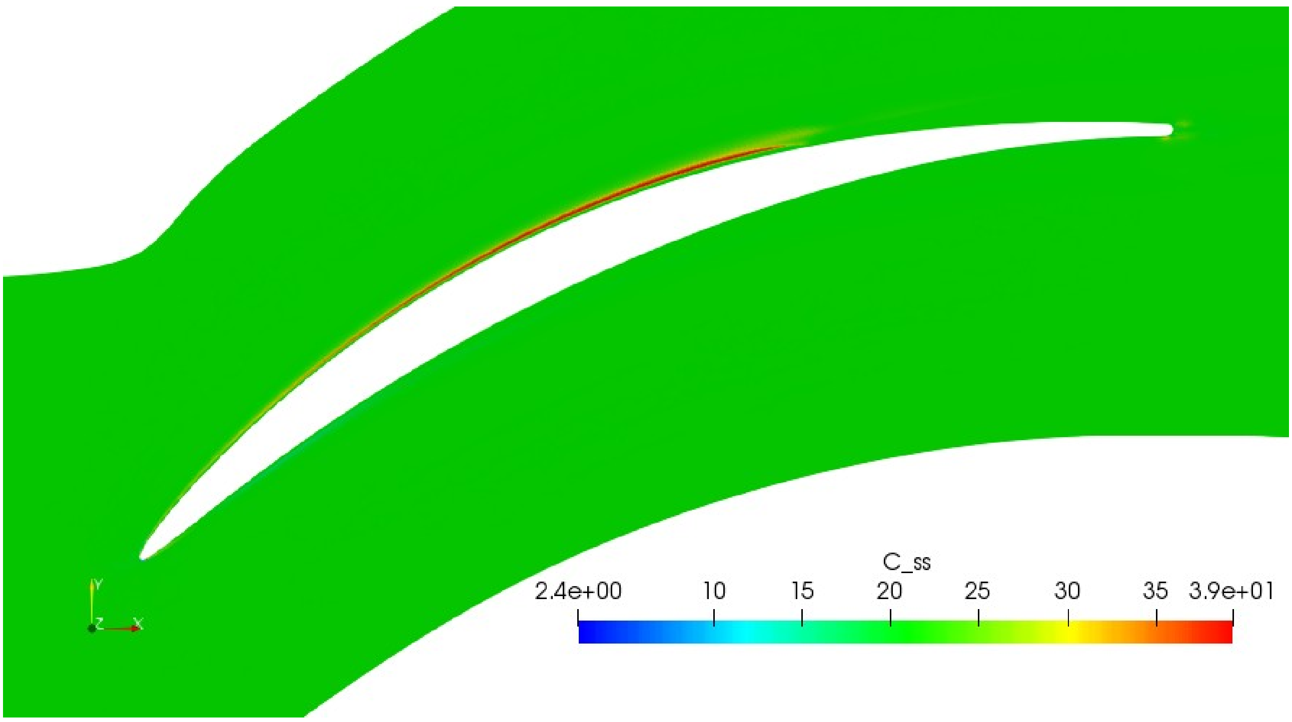




Notably, the contour plot of the non-linear-stress term (a_ij) of the EARSM model looks very similar to that of turbulent intensity (k):

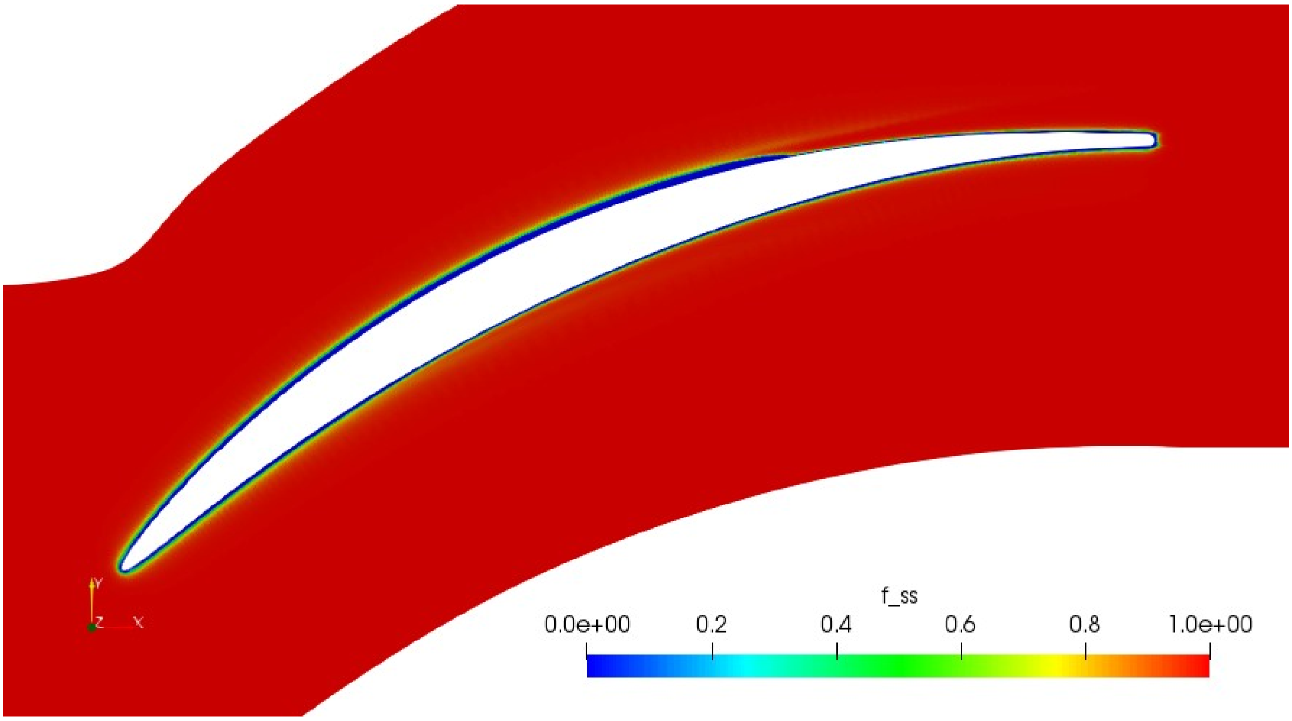



## 4.3 V103_325:: Contour Plots of some parameters of the BaseAlgKOmega model

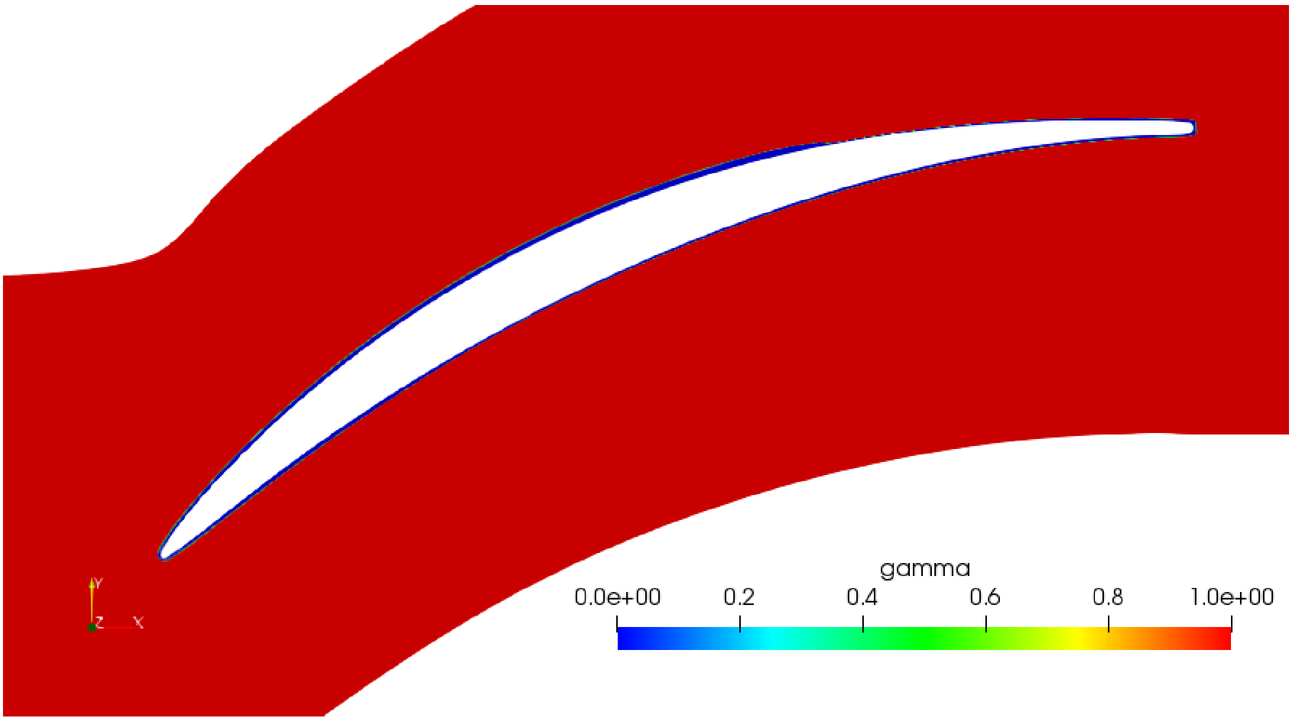
### 4.3.1 C_ss



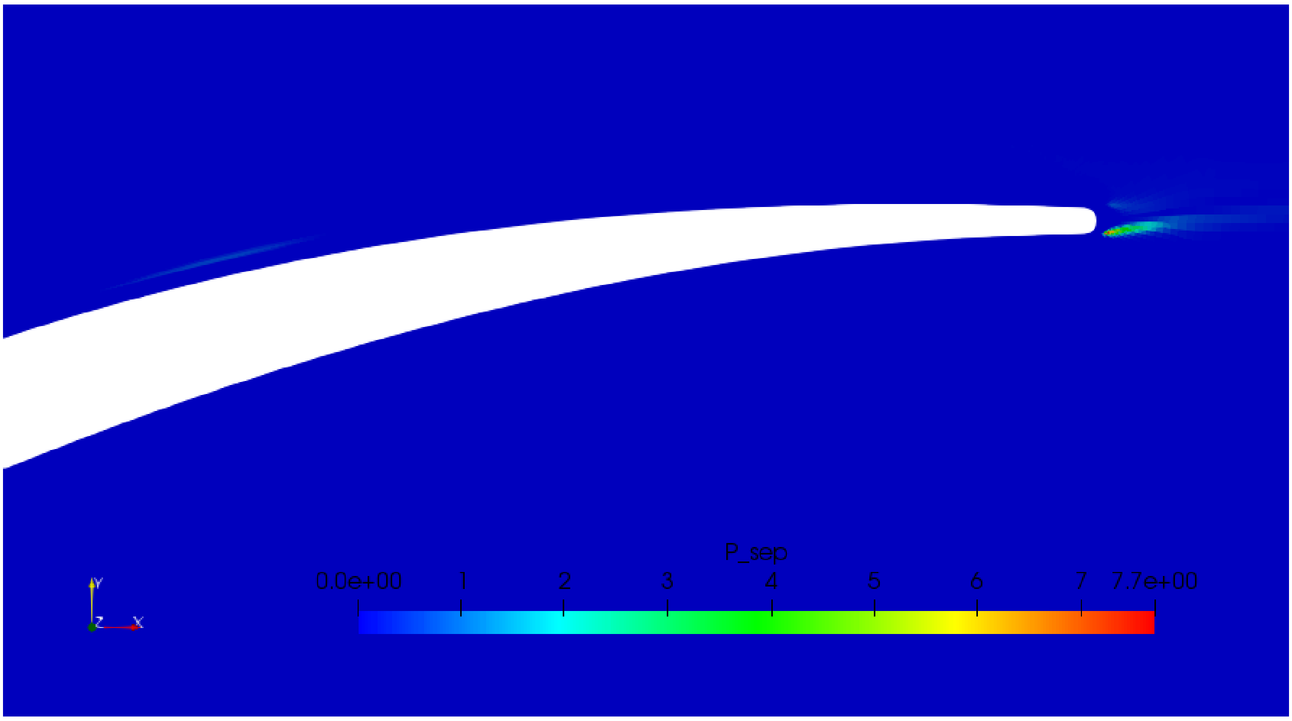
### 4.3.2 f_ss




### 4.3.3 Gamma

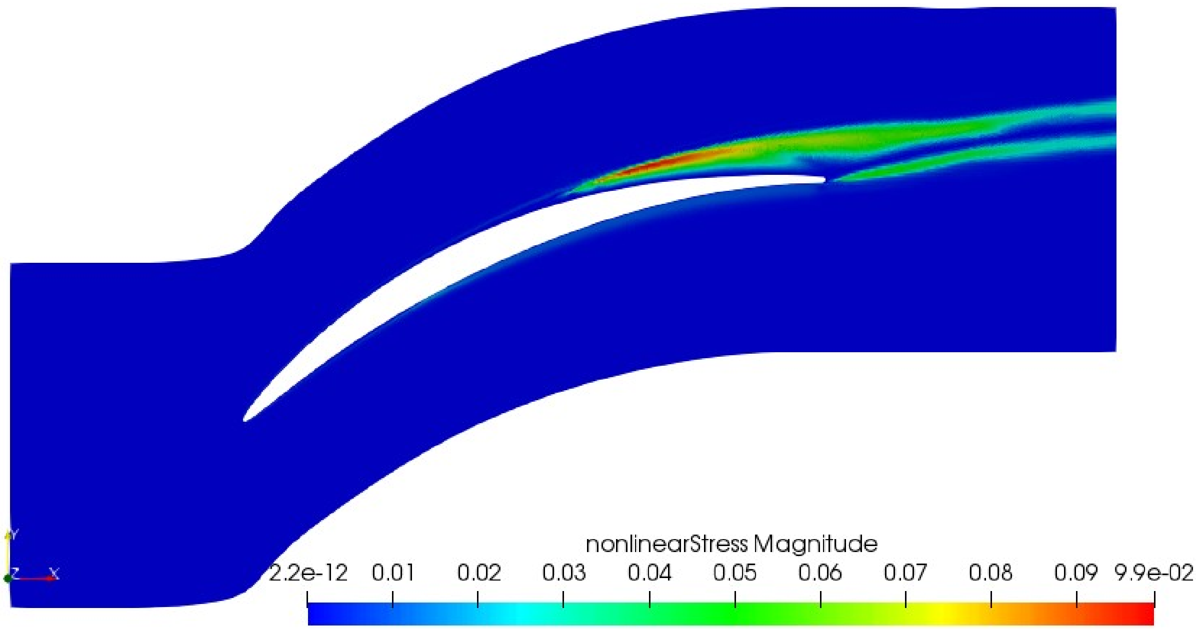


### 4.3.4 P_sep



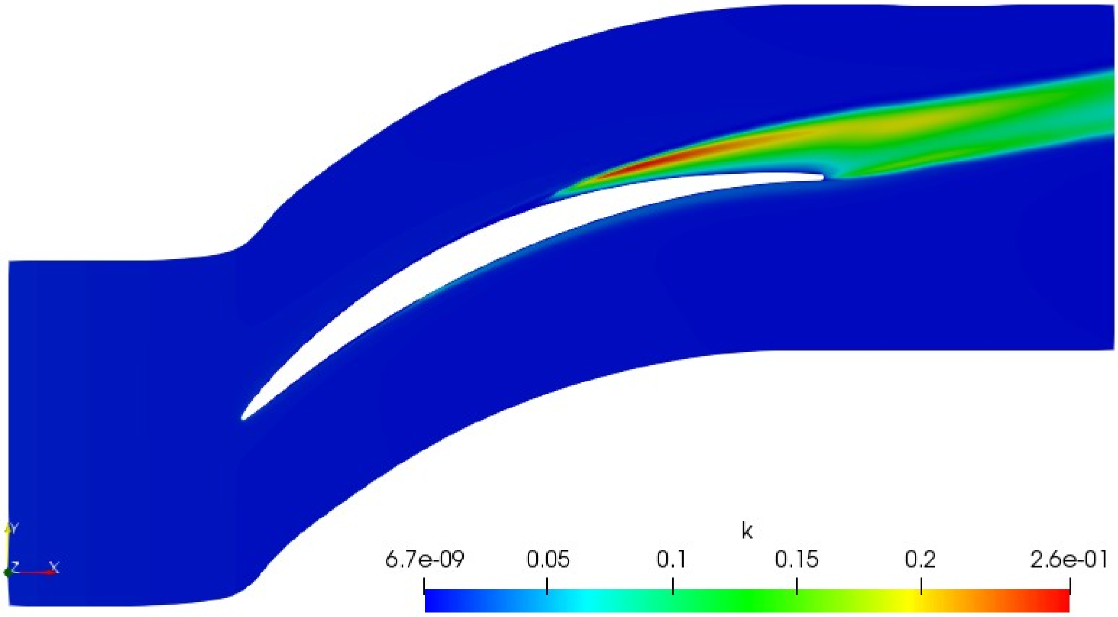

## 4.4 V103_325:: Contour Plots of the non-linear-stress term of the transEARSM model

### 4.4.1 Non-linear stress term (a_ij)



### 4.4.2 Turbulent intensity (k)

Again, the contour-plot of the nonLinearStress-term of the transEARSM model can be compared with the following contour-plot of turbulent intensity (k).




## 5. Conclusion

The new algebraic model, transEARSM, is more calculative expensive but it exhibits better prediction of transition, specially in the cases of natural transition.

The above simulation of different cases of flat plates (T3A, T3A-, T3B) assumes a 2D case which has only one cell in the third dimension. Though this assumption makes the simulation less expensive for computation, it still effects the accuracy of the result obtained from it especially while comparing the simulation output with experimental result.

The above simulations are conducted within technical limitations of computing system of the author. Each case converges necessities can be improved by making the residual roof more demanding.

## Appedix

# 6. Post-processing

To watch convergence of residuals live on the monitor:

1. Use a standard OF function called “residuals”. To use it, add one more command <#includeFunc residuals> in the functions dictionary in the case controlDict file. For example, in the pitzDaily/controlDict

```
functions
{
        #includeFunc streamline
        #includeFunc residuals
}
```

Then while running the solver, simpleFoam, save the output in a log file as simpleFoam > log &

and then hit the following command from the terminal:

foamMonitor -l -r 1 postProcessing/residuals/0/residuals.dat &

This method is explained on a YouTube video by CFD Tutorials but it doesn’t work on OF1812+

2. While running OF, it is a good practice to save intermediate run-time directories as they are helpful to verify weather or not solution is converged.

3. If user want to calculate a particular value, for example Wall Shear Stress, or yPlutees value, for domain of a case after running the solver then he/she can follow this command from the terminal:

simpleFoam -postProcess -func yPlus

simpleFoam -postProcess -func wallShearStress

This command will lead to calculation of yPlus and wallShearStress for the whole domain but only for the saved time-directories. The result can be viewed in post-processing tool like paraView.

# 7. Tips and Tricks

While programming:

1.

While running a case:

1. It is a good practice to keep runTimeModifiable option yes/true/1 in the controlDict while running a case.
runTimeModifiable true;
It also helps to quit simulation of a case at any desired point when it is running in background and its Process ID (PID) number is unknown.

Ubuntu has a command <top> to list all the application and processes running in the computer-system. OpenFOAM facilitates its own command <ps> and <jobs> to list only those processes which are relevant to it.

2. It is recommended to start a simulation with relaxation factor = ~1 (not exactly 1 but rather 0.99 or something) . After a reasonable number of iterations, the relaxation factor should be lowered as required.
If the simpleFoam solver crashes while changing the relaxation factor (case/controlDict file) then the user can use the < | tee -a > command to start writing the output in the old log file (process is called appending) instead of overwriting it.

SimpleFoam | tee -a log

3. If

4. If at any time a solver spits out an error saying
#0 Foam::error::printStack(Foam::Ostream&) at ??:?

#1 Foam::sigFpe::sigHandler(int) at ??:?
....
....
then it means that the solver is encountering a 0/0 calculation while running for the case.

# 8. References


1. Three popular EARSM models published on the NASA Turbulence Modeling Resources website:

https://turbmodels.larc.nasa.gov/easmko.html

2. Cross wires calibration:

https://people.eng.unimelb.edu.au/imarusic/proceedings/20/636%20Paper.pdf

3. To plot residuals,

https://www.cfd-online.com/Forums/openfoam-community-contributions/64146-tutorial-how-plot-residuals.html

4. EROCFTAC bulletin 2016- reports of various experiments

https://www.ercoftac.org/downloads/bulletin-docs/ercoftac_bulletin_106.pdf

5. Calculation of friction coeffecient for flat plate:

https://www.mne.psu.edu/cimbala/me32o C_f0web_Spring_2015/pdf/Flat_plate_turbulent_BL.pdf

6.

…….,……………………………………………………………………………

……………………Numbered Bibliography………………………………

……………………………………………………………………………………

1. ERCOFTAC data for flat-plates simulation: http://cfd.mace.manchester.ac.uk/ercoftac/database/cases/case20/Case_data/
2. Zaki T., Wissink J., Rodi W., Durbin P., Direct Numerical simulation of transition in a compressor cascade: the influence of free-stram turbulence. Journal of Fluid Mechanics
3. ERCOFTAC case setup: http://cfd.mace.manchester.ac.uk/cgi-bin/cfddb/prcase.cgi?20&EXP&database/cases/case20/Case_data&database/cases/case20&cas20_head.html&cas20_desc.html&cas20_meth.html&cas20_data.html&cas20_refs.html&cas20_rsol.html
4. Online tool to calculate intial k, omega: https://www.cfd-online.com/Tools/turbulence.php