RANSE Calculation of Laminar-to-Turbulent Transition-Flow around Sailing Yacht Appendages

Christoph Böhm, R&D-Centre Univ. Applied Sciences Kiel, Yacht Research Unit, Germany
Kai Graf, Institute of Naval Architecture, University of Applied Sciences Kiel (UAS), Germany

ABSTRACT
A new method to simulate laminar-turbulent transition has been used to study flow around sailing yacht appendages. Based on an empirical correlation, the method allows to predict transitional flow in a fully 3D-environment using unstructured grids and massive parallelization. Reynolds number of the flow around the appendages of sailing yachts is of an order where transitional flow plays an important role. The method thus serves as a highly valuable tool for realistic optimisation of yacht appendages.

The paper describes briefly Menter’s $γ$-$Re_Θ$-transition-model and shows its validity for the given purpose. 2D-analysis of NACA-profiles has been chosen to validate the transition model. The resulting lift and drag coefficients from the RANSE calculations have been compared with experimental data and results from the 2D-boundary layer code Xfoil, showing reasonable agreement. The method then has been used for a bulb length optimisation of an appendage configuration of an ORC GP42 level racer. Here the differences between a fully turbulent and a laminar turbulent optimisation are shown and discussed.

NOTATION

(1+k) form factor
$A_{Ref}$ reference surface
c_D drag coefficient
c_f,Hughes friction line according to Hughes
D drag force
$D_k$ destruction term of k equation
$\tilde{D}_k$ modified destruction term of k equation
$D_θ$ destruction term of $θ$ equation
$E_{k1}$ destruction term of transition sources
$E_{k2}$ destruction term of relaminarization sources
$g_i$ component of mass normalized body force
$k$ turbulent kinetic energy

$K$ flow acceleration coefficient
$L$ reference length
$p$ pressure
$P_k$ production term of k equation
$\tilde{P}_k$ modified production term of k equation
$P_θ$ production term of transition sources
$\tilde{P}_θ$ production term of relaminarization sources
$P_{θ,t}$ source term to make $Re_{θ,t}$ match $Re_{θ,t}$
$P_u$ production term of $u$ equation
$Re$ Reynolds number $LU_0/ν$
$Re_c$ critical Reynolds number
$Re_θ$ momentum thickness Reynolds number $ΘU_0/ν$
$Re_θ,c$ critical momentum thickness Reynolds number
$Re_θ,t$ transition onset momentum thickness Reynolds number $ΘU_0/ν$
$Re_θ,t$ local transition onset momentum thickness Reynolds number
$R_T$ viscosity ratio
$Re_y$ Reynolds number based on wall distance
$Re_v$ vorticity Reynolds number
$S_θ$ strain rate tensor
$S$ absolute value of strain rate, wetted surface
$Tu$ turbulence intensity
$U_∞$ freestream velocity
$u_i$ local velocity component
$U_0$ velocity outside of boundary layer
$y$ wall distance
$y'$ dimensionless wall distance
$γ$ Intermittency
$δ$ boundary layer thickness
$δ_*$ displacement thickness
$δ_0$ Kronecker delta
$ɛ$ turbulence dissipation rate
$θ$ boundary layer momentum thickness
$λ_0$ pressure gradient coefficient
$μ$ = $ν$ $ρ$
$μ_λ$ = $ν_λ$ $ρ$
$μ_τ$ friction velocity
$ν$ kinematic viscosity
ν<sub>T</sub> Turbulent viscosity  
ρ Density  
σ<sub>i</sub> constants in diffusion coefficients  
τ wall shear stress  
τ<sub>ij</sub> Reynolds stress tensor  
ω specific turbulence dissipation rate  
Ω Absolute value of vorticity  
Ω<sub>ij</sub> vorticity tensor

INTRODUCTION
RANSE simulations of flow around sailing yacht appendage systems are nowadays common practice in the design stage of high performance sailing yachts. They are used by many flow scientists and yacht designers to optimize the shape of yacht keels, ballast bulbs, rudders and wings. Modern RANSE codes offer great flexibility taking into account viscosity, turbulence and free surfaces, providing results that can compete with towing tank and wind tunnel test results.

A common procedure in the analysis of flow around appendages is to assume fully turbulent flow allowing to use conventional turbulence models. However the Reynolds number of typical yacht appendages is of an order of magnitude where large laminar regions can be expected. In fact designers usually try to extend the laminar region of the appendage set by suitable form modelling to decrease viscous resistance. Consequently flow analysis not taking into account laminar-to-turbulent transition may lead to erroneous design results.

Late developments of turbulence models address this problem. The most promising of these enhanced turbulence models, the γ-Re<sub>θ</sub>-Turbulence model which is implemented in the commercial RANSE solver CFX, uses a correlation-based transition model to predict production and dissipation of turbulent kinetic energy and wall shear stress, MENTER (2004). The correlation used in the transition model is based on experiments to correctly describe the transition location. The principal idea behind this method is to compare free stream turbulence intensity with the local Reynolds number based on boundary layer momentum thickness. Integral quantities needed for this transition criteria are approximated by local variables to avoid numerical problems within a RANSE code.

This transition model is used in a research project at YRU-Kiel to investigate the laminar-to-turbulent transition flow around the appendages of an ORC 42 level racer. The achieved results are compared with results for fully turbulent flow. The ORC 42 has been used as a benchmark since it features a small fin-keel and a medium length ballast bulb, where laminar flow can be expected on about 50% of it’s wetted surface. Goal of this investigation is to show whether the conventional assumption of fully turbulent flow is appropriate or if it may lead to wrong determination of optimal geometric properties of the ballast bulb.

The following chapters give an introduction to the theory of both transitional flow and γ-Re<sub>θ</sub>-Turbulence model and finally two examples which demonstrate the capability of the method to predict laminar-to-turbulent transition.

TRANSITIONAL FLOW
Also the complex mechanism of transition from laminar to turbulent flow, as well as turbulence itself is still not fully researched, it is known, that the transition from laminar to turbulent flow is caused by viscous instabilities, which are developing in the shear layer. These instabilities always increase downstream and depend only on the Reynolds number. At the beginning of the transition process turbulent and laminar regions are mixed in the boundary layer. The degree of this mixing is defined as the intermittency factor γ. Figure 1 shows a scheme of the transition process obtained from experiments on a flat plate.

![Fig. 1: By-Pass-Transition on a flat plate](image)

In case of low levels of turbulence intensity in the free stream, a laminar boundary layer is developing downstream of the leading edge of the flat plate. This laminar region extends until x=x(Re<sub>θ</sub>) is reached, where the first instabilities occur as so called Tollmien-Schlichting (T-S) waves. Shortly afterwards, the mainly 2D T-S waves start to develop 3D fluctuations of speed and pressure, which open out into longitudinal eddies, so called Λ-vortices. The breakdown of these vortices at locations of high vorticity results in the loss of the natural frequency of the T-S waves and the developing of high-frequency vortices. These vortices break down into even smaller vortices, leading to the growth and forming up of
wedge-shaped turbulent spots. Turbulent spots move downstream with the surrounding laminar boundary layer and lead to an alternation between laminar and turbulent flow, the intermittency as mentioned above. The unification of the turbulent spots finally leads to a fully turbulent boundary layer after exceeding the transition Reynolds number \( \text{Re}_t \).

Transition to turbulence is mainly influenced by:

- Local Reynolds number
- Free stream turbulence intensity
- Pressure gradient
- Surface roughness
- Separation

For common yacht appendages the transition length is usually much shorter than the overall length of the appendage element. Consequently accurate prediction of transition onset is usually done without taking into account the details of the transition process. They rather predict the critical Reynolds number \( \text{Re}_n \) and the transition Reynolds number \( \text{Re}_t \) using empirical methods.

Most of these empirical methods are based on the assumption that transition onset is linked to the Reynolds number based on the momentum thickness of the boundary layer. Empirical equations for the transition onset momentum thickness Reynolds number are given by many researchers. They take into account parameters describing details of the flow as pressure gradient and freestream turbulence intensity.

The general problem of implementing one of these methods within RANSE codes is that RANSE simulations use local information only for solving governing equations. Derivation of integral values, which are needed for determining the momentum thickness, is troublesome and should be avoided.

Menters method to circumvent these problems is based on a couple of ideas that allow him to calculate transition from local variables only. His approach covers many of the different aspects of transition: A new empirical formulation for the transition onset momentum thickness Reynolds number in the freestream is given.

- A transport equation is used to diffuse the freestream transition onset momentum thickness Reynolds number into the boundary layer.
- The momentum thickness Reynolds number is correlated with the vorticity Reynolds number which can be calculated from local variables
- An intermittency function is calculated from a transport equation, the source term of this equation depending on the transition onset momentum thickness Reynolds number
- The intermittency function is used to manipulate production and dissipation of turbulent kinetic energy.

The following chapter outlines the basic equations of this model and its integration into common RANSE methods.

FLOW SIMULATION METHOD

The theory given here is a very brief outline of Menters method. For details see the paper of MENTER (2004).

Governing Equations

RANSE solver use a volume based method to solve the time-averaged Navier-Stokes equations in a computational domain around the investigated body. The RANS equation evolves from time averaging mass and momentum conservation for a continuous flow. In the following method it is assumed that the Reynolds stress evolving from time averaging is modelled using the eddy viscosity hypothesis. Assuming incompressible flow this yields, see FERZINGER (1991):

\[
\frac{\partial u_i}{\partial t} + \frac{\partial \left( u_i u_j \right)}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{1}{\rho} \left( \frac{\partial \left( \rho u_i u_j \right)}{\partial x_j} - \frac{\partial \left( \rho u_j u_i \right)}{\partial x_j} + g_i \right) + \frac{\partial}{\partial x_j} \left( \left( \nu + \nu_t \right) \frac{\partial u_i}{\partial x_j} \right)
\]

\[
\frac{\partial u_i}{\partial x_i} = 0
\]

Turbulence Model

The turbulence model used in the presented approach calculates the turbulent viscosity \( \nu_t \) from the turbulent kinetic energy \( k \) and the specific turbulent dissipation \( \omega \):

\[
\nu_t = \frac{\sigma_t k}{\max(\sigma_t, a_t \Omega F)}
\]

The turbulent kinetic energy \( k \) and the specific turbulent dissipation rate \( \omega \) are calculated using the \( \gamma \)-Re\( \Omega \)-Turbulence model shown below.

\[
\frac{\partial \left( \rho k \right)}{\partial t} + \frac{\partial \left( \rho u_i k \right)}{\partial x_i} = \tilde{P}_k - \tilde{D}_k + \frac{\partial}{\partial x_j} \left( \mu + \sigma_t \mu_t \right) \frac{\partial k}{\partial x_j}
\]
\[
\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\rho \mathbf{u}, \omega)}{\partial x_i} = \alpha \frac{P_t}{\nu} - D_\omega + C_d \omega \\
+ \frac{\partial}{\partial x_i} \left( (\mu + \sigma_i \mu_i) \frac{\partial \omega}{\partial x_i} \right) \tag{5}
\]

The \(\gamma\)-Re\(_{\theta_0}\)-Turbulence model is equivalent to the standard SST-Model besides having modified production and destruction terms \((\tilde{P}_i, \tilde{D}_i)\) in the turbulent kinetic energy equation. \(\tilde{P}_i\) and \(\tilde{D}_i\) are calculated from production and destruction of turbulence in fully turbulent regimes, \(P_i\) and \(D_i\), and an intermittency function \(\gamma\) which controls production and destruction of turbulent kinetic energy depending on the transition onset momentum thickness Reynolds number:

\[
\tilde{P}_i = \gamma_{\theta_0} P_i \tag{6}
\]

\[
\tilde{D}_i = \min(\max(\gamma_{\theta_0}, 0.1), 1.0) D_i \tag{7}
\]

**Transition-Model**

The transition model assumes that the transition onset momentum thickness Reynolds number outside the boundary layer can be calculated by an empirical formula. Menter suggests a new empirical correlation for Re\(_{\theta_0}\), which relates it to the freestream turbulence intensity \(Tu\) and two empirical correlation factors \(\lambda_0\) and \(K\). The correlation is defined as:

\[
\text{Re}_{\theta_0} = 803.73 [Tu + 0.6067]^{-0.027} F(\lambda_0, K) \tag{8}
\]

\[
\lambda_0 = \left( \frac{\Theta^2}{\nu} \right) \frac{dU}{ds} \tag{9}
\]

\[
K = \left( \frac{\nu}{U^2} \right) \frac{dU}{ds} \tag{10}
\]

The function for \(\lambda_0\) shows strong similarities to a pressure gradient method proposed by Thwaites, whereas the second parameter \(K\) is the so called acceleration parameter similar to the method proposed by Mayle, for details see WHITE (1991). The acceleration parameter \(K\) is used to characterize the accelerations at the beginning of the transition process. Function \(F(\lambda_0, K)\) is used to decide if a polynomial function depending on \(\lambda_0\) or \(K\) is used to determine Re\(_{\theta_0}\).

A local value of the transition onset momentum thickness Reynolds number is calculated from a transport equation, assuming that the freestream value of Re\(_{\theta_0}\), calculated from (8), diffuses into the boundary layer:

\[
\frac{\partial(\rho \hat{\text{Re}}_{\theta_0})}{\partial t} + \frac{\partial(\rho \mathbf{u}, \hat{\text{Re}}_{\theta_0})}{\partial x_i} = P_{\theta_0} + \frac{\partial}{\partial x_i} \left( \sigma_{\theta_0} (\mu + \mu_i) \frac{\partial \hat{\text{Re}}_{\theta_0}}{\partial x_i} \right) \tag{11}
\]

Here the source term \(P_{\theta_0}\), ensures that \(\hat{\text{Re}}_{\theta_0}\), matches the local value of Re\(_{\theta_0}\) outside the boundary layer, the latter one obtained from (8). The source term \(P_{\theta_0}\) contains a blending function, which turns off the source term in the boundary layer.

The transition onset momentum thickness Reynolds number in the boundary layer, \(\hat{\text{Re}}_{\theta_0}\), is related to the critical momentum thickness Reynolds number Re\(_{\theta,c}\), which is the Reynolds number where first instabilities in the laminar boundary layer occur. Unfortunately this relationship is proprietary and a hidden procedure within Menter’s method:

\[
\text{Re}_{\theta,c} = f(\hat{\text{Re}}_{\theta_0}) \tag{12}
\]

The critical momentum thickness Reynolds number Re\(_{\theta,c}\) is now forwarded to the intermittency transport equation, which controls the production and destruction of turbulence in the turbulence model:

\[
\frac{\partial(\rho \gamma)}{\partial t} + \frac{\partial(\rho \mathbf{u}, \gamma)}{\partial x_i} = P_{\gamma} - E_{\gamma} + P_{\gamma} - E_{\gamma} + \frac{\partial}{\partial x_i} \left( \mu + \frac{\mu_i}{\sigma_i} \right) \frac{\partial \gamma}{\partial x_i} \tag{13}
\]

In this transport equation, \(P_{\gamma}\) and \(E_{\gamma}\) depict the transition sources necessary to start the production of turbulence whereas \(P_{\gamma}\) and \(E_{\gamma}\) are the destruction or relaminarization sources.

\(P_{\gamma}\) is controlled by an onset function which depends on the ratio of the local momentum thickness Reynolds number Re\(_{\theta}\) and the critical momentum thickness Reynolds number Re\(_{\theta,c}\). To avoid the calculation of the non-local momentum thickness Reynolds number Re\(_{\theta}\) and the local vorticity Reynolds number Re\(_{\psi}\):

\[
\text{Re}_{\Theta} = \frac{\text{Re}_{\theta,\text{min}}}{2.193} \tag{14}
\]

Re\(_{\psi}\) is based on local values only and can therefore easily determined at each computational node:

\[
\text{Re}_{\psi} = \frac{\Omega^2}{\mu} \tag{15}
\]
APPLICATIONS

In the following the method of Menter is validated by comparing it with respective experimental and numerical results from other sources. It is then used to investigate the flow around the appendage set of an ORC GP 42 level racer. While the first investigation assumes planar flow, the appendage flow simulation is fully 3D. The original empirical methods to predict transition from the momentum thickness Reynolds number are 2D by nature. However due to the substitution of integral values with local variables fully 3D flow can be investigated with Menters method. In addition, since the local transition onset momentum thickness Reynolds number in the boundary layer, $\tilde{Re}_{\theta}$, is calculated from diffusion of the freestream transition onset momentum thickness Reynolds number, in-homogeneity of turbulence intensity in the freestream can be taken into account. This allows to take account of interaction of appendage elements like bulb and blade or even blade and rudder.

2D TEST CASE: NACA 642-015 PROFILE

A profile of the well researched NACA 6-series was chosen for analysis to verify the capability of the $\gamma$-$Re_{\theta}$-Turbulence model to predict transition. The results of the profile analysis have been compared with experimental data collected from Abbott/Doenhoff, ABBOTT (1959) and numerical results generated by the well known profile code XFOIL of DRELA (2001).

The simulation has been performed using a hexahedral mesh with C-grid topology and approx. 180000 computational nodes. As an important pre-conditions to achieve accurate results using the $\gamma$-$Re_{\theta}$-Turbulence model the dimensionless wall distance has to be restricted to $Y+ \leq 1$. Thus the wall nearest node of the grid has to be placed at a distance of approximately 0.003 mm from the surface to achieve correct results for a Reynolds number of $3*10^6$. Consequently great care has to be taken in the design of the profile surface, which has to be very smooth to allow proper gridding of the cells in the vicinity of it.

Fig. 2 shows the hexahedral mesh, the inset showing grid resolution at the profile leading edge.

A sweep of angle of attacks from $0^\circ$ to $10^\circ$ have been tested at constant inflow speed. Turbulence intensity in the freestream has been set to $0.85\%$, which corresponds to an $N_{crit}$-Value of 3.012 in the Xfoil settings. The exact Turbulence level at which the experiments were conducted in the wind tunnel is unknown, however Abbott/Doenhoff mentioned it to be in the order of a few hundredths of one percent.

Fig. 2: Grid around NACA 642-015 profile

Fig 3 shows the drag coefficient $c_D$ over angle of attack AoA for RANSE calculation (both laminar-turbulent and fully turbulent), XFOIL calculations and experimental data. Drag coefficient $c_D$ is calculated by dividing the drag force obtained from the computation with dynamic pressure and the planform area.

$$c_D = \frac{D}{0.5 \cdot \rho \cdot U^2 \cdot A_{ref}} \quad (16)$$

Fig. 3: Drag coefficient $c_D$ for NACA 642-015

Generally the RANSE results show rather good compliance with both numerical and experimental comparative data, in particular close to the non-lifting condition AoA=0$. This is a good indicator for proper laminar to turbulent transition prediction. Here the results from the new transition model almost coincides with the experimental data at angles from 0 to 2 degrees. Compared to the experimental results the beam of the drag bucket is narrower for the RANSE results, however comparing the
RANSE results with those from XFOIL, the trend is inversed. Additional experimental results are needed here to evaluate the merit of RANSE compared to XFOIL results.

For wider angles of attack, the curves for the laminar-turbulent RANSE calculations show a different slope than the comparative data and quickly approach the results from the fully turbulent RANSE calculation. In this region the need for further research in order to determine possible flaws in the calculation setup and to calibrate the model is obvious. It is also worth considering that the quality of the experimental and numerical comparative data itself is unknown.

Fig. 4 compares transition points on top and bottom of the surface resulting from numerical calculations with CFX and XFOIL. It can be nicely seen that the curve slopes are very similar. Given the fact that XFOIL predicts narrower drag bucket than the RANSE calculations, the horizontal shift of the top transition points are very plausible.

In general the agreement between the different methods compared here seems to be reasonable. This holds particularly for a situation, where design input from an optimisation has to be generated. Comparing variants of topologically similar geometries like ballast bulbs of different volume distribution or length will give quite reasonable trends as long as the simulation set-up is not changed.

Fig. 5 shows porcupine-plots representing the wall shear stress on the profile. The upper plot depicts a profile in fully turbulent flow, visible by a smoothly decreasing distribution of wall shear stress in flow direction. In the lower plot the result for a laminar-turbulent transition simulation is shown.

Here the wall shear stress progression shows the typical characteristic of laminar to turbulent transition: a sharp increase close to the stagnation point followed by a distinct shear stress decrease over almost 60% of the chord length. The transition to turbulent flow can be identified at the location where shear stress starts to increase again indicating turbulent flow in the backward region.

**BULB LENGTH VARIATION**

As a practical application a length variation of a keel bulb configuration for an ORC GP 42 level class racer will be presented here. When trying to predict the optimum length for a ballast bulb of given buoyancy the following problem occurs:

Bulbs optimised using fully turbulent CFD codes are known to become very long and slender, whilst bulbs which are optimised using methods capable to take into account laminar turbulent transition (like wind tunnel testing) tend to be too bulky. The reason for this is, that for fully turbulent flow a prolongation of the bulb increases frictional resistance only slightly as can be derived from Fig. 5. For a bulb with a laminar region at the forward part any prolongation increases only the region where turbulent flow prevails, resulting in an over-proportional increase of resistance. Of course the bulb resistance in fully turbulent flow is generally higher than in laminar turbulent transition flow.

For comparison the bulb length investigation is carried out for laminar turbulent transition flow as well as for fully turbulent flow. This allows to estimate the error margin, giving a measure of the merits bound to the use of the presented method.
Geometry

No complete design of a GP 42 racer has been available at the YRU Kiel. Consequently the blade and the benchmark bulb have been developed according to the ORC GP42 level class rules with dimensions likely to be used in reality. From estimation of blade dimensions and weight a benchmark bulb with the following characteristic properties has been derived:

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length L [m]</td>
<td>2.400</td>
</tr>
<tr>
<td>Wetted Surface S [m²]</td>
<td>2.215</td>
</tr>
<tr>
<td>Vertical Center of Gravity VCB [m]</td>
<td>0.154</td>
</tr>
<tr>
<td>Long. Center of Gravity LCB [m]</td>
<td>-0.323</td>
</tr>
<tr>
<td>Volume Vol [m³]</td>
<td>0.170</td>
</tr>
<tr>
<td>Height H [m]</td>
<td>0.326</td>
</tr>
<tr>
<td>Width B [m]</td>
<td>0.482</td>
</tr>
<tr>
<td>B/H Ratio SQR [m]</td>
<td>1.480</td>
</tr>
<tr>
<td>Maximum Draft MD [m]</td>
<td>2.300</td>
</tr>
</tbody>
</table>

The bulb variants are derived from the benchmark geometry by affine distortion of the bulb length, height and width. During the study the bulb volume and its squish ratio (B/H-ratio) as well as the maximum draft remains constant. The following variants have been developed:

<table>
<thead>
<tr>
<th>Variant</th>
<th>L [m]</th>
<th>S [m²]</th>
<th>VCB [m]</th>
<th>H [m]</th>
<th>B [m]</th>
<th>Vol [m³]</th>
</tr>
</thead>
<tbody>
<tr>
<td>CSYS 18_V1</td>
<td>1.900</td>
<td>1.988</td>
<td>0.173</td>
<td>0.367</td>
<td>0.543</td>
<td>0.170</td>
</tr>
<tr>
<td>CSYS 18_V2</td>
<td>2.150</td>
<td>2.104</td>
<td>0.164</td>
<td>0.345</td>
<td>0.510</td>
<td>0.170</td>
</tr>
<tr>
<td>CSYS 18_V3</td>
<td>2.650</td>
<td>2.323</td>
<td>0.147</td>
<td>0.310</td>
<td>0.459</td>
<td>0.170</td>
</tr>
<tr>
<td>CSYS 18_V4</td>
<td>2.900</td>
<td>2.427</td>
<td>0.140</td>
<td>0.297</td>
<td>0.439</td>
<td>0.170</td>
</tr>
<tr>
<td>CSYS 18_V5</td>
<td>3.150</td>
<td>2.527</td>
<td>0.135</td>
<td>0.285</td>
<td>0.422</td>
<td>0.170</td>
</tr>
</tbody>
</table>

The bulbs come completely without chines and have an asymmetric profile to keep the centre of gravity low. This results in a rather flat bottom part.

Simulation Setup

The simulation has been set up in a boxed environment similar to those encountered in wind tunnels, with the following differences: The box walls are treated as frictionless free slip walls and the appendage configuration is tested in full scale, thus eliminating any problems with Reynolds similarity. The boundary conditions applied are a constant inflow speed of 5.144 m/s on the inlet, which is corresponding to a speed of 10 knots. On the outlet von Neumann condition (derivation of flow forces perpendicular to the boundary is zero) is applied. The bulb is investigated in downwind condition only, the free stream turbulence intensity has been set to 0.85% as during the profile investigation.

All geometries have been simulated using two different gridding schemes: A hexa-tetra grid uses hexahedral cells in the vicinity of the keel to accurately resolve the boundary layer, while a tetrahedral grid topology has been used for the far field in order to restrict the necessary number of grid cells. In a tetra-prism grid the boundary layer flow is resolved using flat prisms while the rest of the flow domain is discretized using tetrahedral cells. The first method consists of an inner box around blade and bulb, which is meshed with a multi-block structured hexahedral mesh. The far field (outer box) is meshed with a relatively coarse tetra mesh to save computational time. The hexahedral and the tetrahedral grida are finally connected with a node-matching interface via pyramid cells.

The second meshing approach is a fine unstructured tetrahedral grid with prism layers at the investigated surfaces for a refined resolution in the
boundary layer. This method has the advantage that it can be easily adapted to any geometry, but comes with the disadvantages that the task of resolving the correct boundary layer height is not as easily performed as with the hexahedral mesh and the computational effort is bigger.

<table>
<thead>
<tr>
<th>Elements</th>
<th>Hexa-Tetra-Hybrid Grid</th>
<th>Tetra-Prism Grid</th>
<th>Hexa-Tetra-Hybrid Grid</th>
<th>Tetra-Prism Grid</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hexahedral</td>
<td>2784428</td>
<td>-</td>
<td>1512944</td>
<td>-</td>
</tr>
<tr>
<td>Tetra</td>
<td>828109</td>
<td>992155</td>
<td>1421511</td>
<td>992155</td>
</tr>
<tr>
<td>Prism</td>
<td>-</td>
<td>4350902</td>
<td>-</td>
<td>869978</td>
</tr>
<tr>
<td>Pyramids</td>
<td>50656</td>
<td>158</td>
<td>41152</td>
<td>66</td>
</tr>
<tr>
<td>Σ Elements</td>
<td>3663193</td>
<td>5342315</td>
<td>2975607</td>
<td>1862199</td>
</tr>
<tr>
<td>Σ Nodes</td>
<td>2909960</td>
<td>2358636</td>
<td>1745740</td>
<td>618495</td>
</tr>
</tbody>
</table>

Tab. 3: Grid parameters

Both grid variants have been developed with approximately the same grid resolution, see Tab. 3. The tetra-hexa grid has a computational load which is around half a million nodes greater than with the tetra-prism grid. This mainly owed to the fact that the tetra-prism grids allows for more aimed node distribution and can therefore save some nodes, whilst hexahedral grids always come with some constraints that cause the grid size to increase.

The grids for the fully turbulent simulations are generally around 40% (hybrid) – 70% (tetra-prism) smaller than the laminar-turbulent grids. This is due to high resolution needed in the boundary layer for the γ-Reθ-Transition model to work correctly. The differences between the prism elements in fully turbulent and laminar unstructured grids depict this very well, see Tab. 3. Naturally this also enlarges the differences in node numbers between the two grid approaches developed for fully turbulent flow.

It has to be mentioned that the meshing effort for laminar-turbulent calculations is huge, since the grid generator has to capture the intersections between bulb and blade with the same high resolution as with the profile investigation. Therefore it is even more important than for the 2D test case to invest great care in developing smooth and clean surfaces.

**Results Hexa-Tetra Grid**

The computational results achieved with the hexa-tetra grid suffer from a most unexpected drawback: a review of the form factors shows that the pressure drag of the bulb is heavily underestimated. The form factor is defined as follows:

$$ (1 + k) = \frac{c_D}{c_{F,Hughes}} \quad (17) $$

where $c_D$ is defined as in formula 14, but normalized using the wetted surface instead of the planform area. The coefficient $c_{F,Hughes}$ resembles the friction line as introduced by Hughes.

$$ c_{F,Hughes} = \frac{0.063}{(\log(Re) - 2)^2} \quad (18) $$

A length variation is usually an interplay between the viscous drag, mainly depending on the wetted surface, and the pressure drag, depending on the form of the investigated body. A massive pressure underestimation leads to a very blunt bulb, this being the body with the least wetted surface. Obviously these results are useless as they do not resemble reality.

![Fig. 8: Unstructured tetra grid with prism layer on bulb and blade surfaces](image)

![Fig. 9: Results of fully turbulent length variation with hexa-tetra-hybrid grid](image)
Fig. 9 and Fig. 10 show the results for the fully turbulent and the laminar- turbulent test run with the hexahedral grid. All results have been modified in a way that they depict the difference to the benchmark geometry in percent. Besides total drag the curves for the resistance components viscous and pressure drag, as well as the development of the bulb vertical centre of gravity are also given to show the different trends.

In case of the fully turbulent length variation an optimum was found, at a length of 2.15m, this being 0.25m smaller than the benchmark bulb. This resulted in a bulb which was bulkier than one would expect for a fully turbulent calculation. The calculations taking laminar-turbulent transition into account did not lead to an optimum at all. Here the result was that any shortening of length leads to decrease of resistance.

A thorough investigation of the simulation lead to the conclusion that the problems have to be grid related. Whether the problems arose from the mixing of tetra and hexa grid, from the blocking structure or from the node distribution could not yet be resolved. Currently we assume the pressure underestimation to be related to distortion of the hexa grid caused by the use of an unconventional blocking strategy.

Results Unstructured Tetra-Prism Grid

The results received from the simulations with the second meshing approach, using the tetra-prism grid, are by far more encouraging. The problems of massive pressure underestimation which occurred with the first grid did not appear here and the ratio of pressure and viscous drag looks very realistic.

Fig. 11 shows results for fully turbulent calculations using the tetra-prism grid. Here an optimum of total drag was acquired for a bulb length of 3.15m, which is +0.75m longer than the benchmark bulb. The optimum for the fully turbulent calculation is rather flat, however it fits well with theory and common knowledge that this kind of simulation does tend to produce long and needle-like bulbs.

The results of the simulations conducted with the new laminar-turbulent-transition model are shown in Fig. 12. As to be expected, the pressure drag increases with decreasing bulb length due to the bulkier shape of the body. Simultaneously the viscous drag decreases because the wetted surface of the body gets smaller. The optimum distribution of pressure and viscous drag is reached for a bulb length of 2.15m. This corresponds to a delta of -0.25m to the benchmark. The optimum bulb length for a bulb in laminar turbulent flow is about 1 m shorter than for a bulb in fully turbulent flow.
Fig. 12: Influence of Length Variation with respect to Benchmark Geometry CSYS18_BM (Laminar-Turbulent-Transition Model)

Fig. 13 shows longitudinal location of transition onset for the bulbs. The transition locations are measured at bottom, top and side of the bulb and normalized with the bulb length. An increase of bulb length decreases the longitudinal position of the transition location quite smoothly towards the bulb nose. The sole exception here is bulb variant V3 which shows a unexpected peak on the bulb bottom.

By comparing the results for the bulb top, bottom and side, one can see that the changes are most distinctive for the bulb top. This seems to be plausible since the top of the bulb geometry is intersecting with the blade. Corner vertices as well as transition on the blade may cause the flow on the bulb top to transit earlier to turbulence thus overriding the natural transition process.

Fig. 14 and Fig. 15 depict wall shear stress for the benchmark bulb for fully turbulent flow and for laminar-turbulent transitional flow. Fig. 14 shows shear stress increasing from zero at the nose to maximum somewhere behind mid section. Afterwards the wall shear stress decreases towards the end.

Fig. 15 shows a similar contour plot for laminar-turbulent transition flow around benchmark bulb. Shear stress is characterized by a sudden increase of wall shear stress on the nose, followed by a swift decrease to a very low level.

Fig. 13: Transition Points on bulb calculated with the γ-Re₉₀-Transition model

Fig. 14: Contour plot of wall shear stress on benchmark bulb (fully turbulent)

Fig. 15: Contour plot of wall shear stress on benchmark bulb (laminar-turbulent)
This low level of shear stress is maintained until the critical Reynolds number $Re_c$ is reached. From here the wall shear stress increases to almost the same value as with the fully turbulent calculation. It then follows the same pattern as described for the fully turbulent calculation and decreases towards the beaver-tail end of the bulb.

Fig. 16, Fig. 17 and Fig. 18 show turbulent kinetic energy on the surface of the benchmark bulb, the V2 variant (very short bulb) and the V4 variant (very long bulb) respectively. The images unveil the development of laminar and turbulent zones for the three selected bulb lengths. Here blue colour regions indicate laminar flow, whilst regions of turbulent flow are coloured red. The bulbs shown here are the benchmark bulb, the optimum-length bulb for laminar turbulent transition flow (V2) and the optimum-length bulb for fully turbulent flow (V4).

Flow at the bulb-blade junction of the benchmark bulb transits relatively early to turbulent. By looking at the transition course at blade and bulb, it seems that instabilities of the flow on the bulb force the flow on the blade to transit to turbulence forward of the natural transition location.

The optimum bulb for laminar-turbulent transition flow (Fig.17) possesses several differences compared to the benchmark bulb. Noticeable is the rather late transition of laminar to turbulent flow on the entire bulb surface. The laminar-turbulent transition at the upper part of the bulb surface is probably triggered by the flow instabilities on the blade. However, late transition at the bulb side may be a consequence of higher curvature and thus negative pressure gradients compared to the benchmark.

Fig. 16: Contour plot of turbulent kinetic energy on the benchmark bulb indicating laminar (blue) turbulent (red) flow

Fig. 17: Turbulent kinetic energy on the V2 bulb

Fig. 18: Turbulent kinetic energy on the V4 bulb

Fig. 19 shows turbulent kinetic energy at bulb bottoms for the entire range of investigated bulb lengths. By comparison the different lengths of laminar and turbulent regions can be detected easily.

It can be generally stated that the flow on the top and bottom side of the bulb transits earlier to turbulent flow than at the bulb sides. Furthermore, the contour of the transit regions varies from a more or less even sectional
distribution around the shortest bulb (V1) towards a wedge shape turbulent peak at bottom of the longest bulb (V4.)

This phenomena is surely owed to the geometry of the bulb which was designed rather flat with a B/H ratio of 1.48 in order to keep the centre of gravity low. Additionally the bulb bottoms are flatter than the bulb tops. This also enhances hydrostatic stability, but is not suited to stabilize the laminar flow.

![Contour plot of kinetic energy on bulb bottom](image1)

**Fig. 19: Contour plot of kinetic energy on bulb bottom**

The most obvious evidence for this theory is the smallest bulb (Variant V1), of which the bottom and top side show highest curvature. This introduces a laminar region of almost equal length around the entire body.

A cut through the blade profile shown in the Fig. 20 demonstrates that the ϑ-Re$_{Θ}$-Transition model is capable of catching flow phenomena otherwise only seen in costly experimental investigation. One can see that due to an increase of pressure, which results in a loss of kinetic energy (due to friction), the laminar boundary layer is no longer able to maintain attached flow at the wall and consequently separates. This triggers the boundary layer to transit to turbulence, thus gaining additional momentum which diffuses back into the now turbulent boundary layer allowing it to reattach to the wall.

![Boundary layer profile at transition location](image2)

**Fig. 20: Boundary layer profile at transition location**

**Conclusion**

In this paper a new laminar-turbulent transition model has been tested for its ability to predict laminar-to-turbulent transition flow around yacht appendages. The method consists of an empirical approach for the transition onset momentum thickness Reynolds number and correlations of integral values and local variables to be solvable within a RANSE code.

The method has first been verified by comparing results of a 2D-profile investigation with respective investigations using a boundary layer method (XFOIL) as well as experimental data. The drag coefficients from the different data sources have been compared and the agreement between them was found to be generally sufficient.

The second test case, a length variation of a typical ORC42 ballast bulb, successfully extends the calculation to a fully 3D environment. The length variation shows that the assumption of fully turbulent flow does not only lead to a significant overestimation of the flow forces, but – of even more importance – also may lead to wrong determination of optimal geometric properties of the ballast bulb.

It could be shown that the optimum low-drag bulb length is significantly longer assuming fully turbulent flow than the optimum bulb length determined taking laminar regions on the bulb surface into account.

It has to be pointed out that this paper investigates optimum bulb lengths with respect to resistance minimization only. To maximize boat speed the impact of bulb length on hydrostatic stability has to be taken into account.
REFERENCES


