07.2 Bulb design

See plan VO70-A1

07.2.1 Overview

The requirement of the canting keel is to carry the bulb; very limited sideforce will be produced by the keel. As such the bulb has no requirement to act as an endplate to reduce tip vortex ;(i.e. induced drag). Bulb optimisation is to reduce drag and lower the centre of gravity; such study can be done without thinking about the fin.

07.2.2 Computational Fluid Dynamic

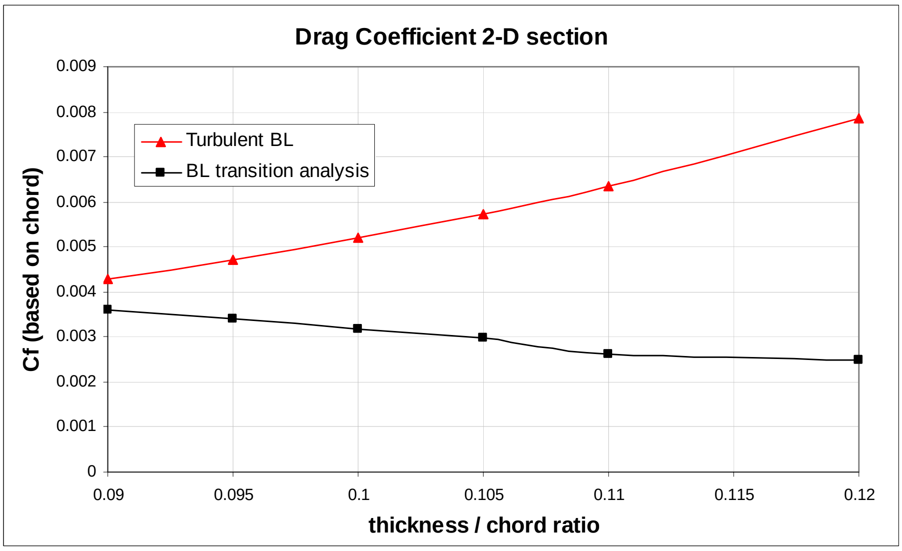
A CFD RANSE study of bulb shapes was not found time-efficient and result-efficient due to several raisons.

As seen during the last America's cup, bulb design is still a very open part and not well understood part of yacht design. This can be explained by the difficulty to predict the behaviour of the boundary layer (transition and separation) over such 3d body, and the limited investigation done by aeronautic company on similar bodies. Not much technical papers have been done on such bodies, except tip tanks and underwater missile. They most often deal with completely turbulent flow and no angle of attack. Optimum body geometry can be found for complete turbulent flow. But large gain can be getting in increasing the laminar extent on the body. This lack of knowledge can be shown looking at the bulb design process of the winner of the last America's cup Alinghi; they assumed laminar boundary layer flow over a finite length from experimental work. There is also lack of knowledge of turbulence behaviour differences between water and air.

Using today Turbulence algorithm in the design of bulb can give wrong results. It is common knowledge that RANSE solver will give the advantage to longer bulb.

Example

A good example of the problem can be shown with a study of the best thickness chord ratio for a 2d foil. The next graph (*Graph 07.2.a*) shows the results of the analysis using two different configuration, the first one the one assuming the boundary layer to be completely



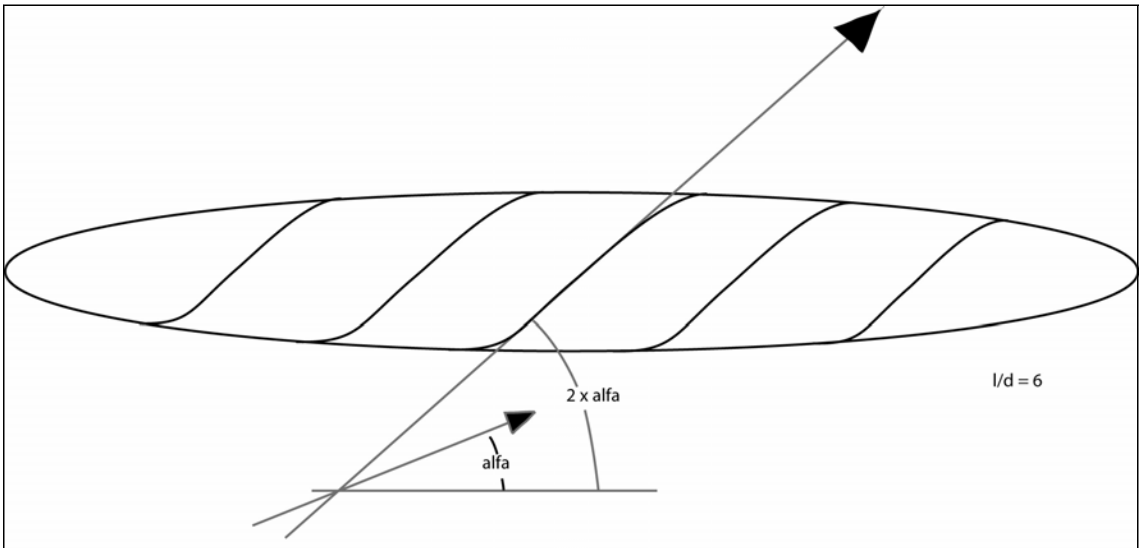
Graph 07.2.a

turbulent (as in RANSE), the second is where transition is modelled using the e^N method. This analysis represents a bulb cross section, therefore the cross section revolved around its axis must have the same volume, and an increase in t/c ratio gives a decrease in length hence in Reynolds number. The graph clearly shows that the conclusion are opposite, increasing t/c in the turbulent case increases area (wetted length/chord is higher) and pressure drag is increased, and therefore increases overall drag. In the case where laminar to turbulent transition is modelled, increasing t/c decreases drag, it is due to the fact that as t/c increases, pressure gradient on the fore body increases and therefore laminar flow can be sustained for a longer length.

07.2.3 Preliminary Bulb shape

07.2.3.1 Assumptions, main aspects

To study bulb drag, it is important to deal with angle of attack; as shown next the flow on a revolved body at an angle alpha to the flow has streamlines over its body of 2*alpha (Diagram 07.2.a). However leeway or pitch angle on such boats will be assumed to be small enough to be neglected.



Flow patern observed directly at the surface of a streamline body of revolution
Reproduced from Hoerner [7]

Diagram 07.2.a

The difficulty is the value of Reynolds number (around 5e6) which put us in the middle of transitional range (between laminar and turbulent). This is where large gain can be made. Decrease in drag as much as 50% can be achieved. These high decreases are achievable with body designed for specific Rn and with high D/L which is not applicable to the main role of the bulb; produce righting moment deep down. Drag predictions are problematic, however, as transition is difficult to predict.

Laminar flow on the forward part requires good pressure gradients hence a maximum thickness at the back and an important D/L.

Another area of research is to make the bulb asymmetric in the horizontal plane to move its cog down. Such aspect has to be though through due to the fact that the bulb will produce lift (directed upward) and be a nuisance when the keel is canted. It will increase base drag too.

Bulb has different requirement that affects its design, they are lower the CoG, act as an endplate, and limit its resistance. In fixed keel yacht a complex compromise has to be found, as shown is the last America’s cup the choices are difficult. A canting keel boat has the advantage that the keel is not the main source of lift hence the bulb doesn’t have to be optimised to be effective as endplate.

The importance of placing the CoG of the bulb deep down is expected to be an important factor, on fixed keel yacht a 1cm gain in CoG will give a gain of 1*sin(heel) of righting moment (righting moment can be expected to be proportional to GM for narrow boats). On a canting keel yacht a 1cm gain in CoG will give a gain of

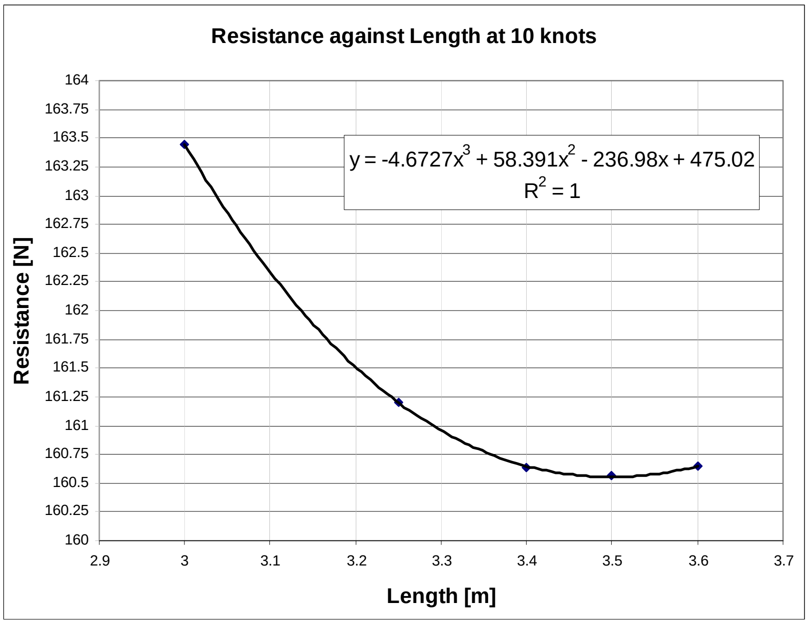
1*sin(heel+CKangle) of righting moment; for a heel angle of 15 the gain in RM is higher by 316 %.

07.2.3.2 Length

The design process is started using statistical regression data of streamline rotationally symmetric bodies. Such data can be found in Hoerner [8], above the critical Reynolds number (around 10^5), an interpolation of the data point is found to give a total drag based on wetted area of:

$$C_D=C_F*(1 + 1.5\,(d_B/l_B)^{3/2} + 7(d_B/l_B)^3) \qquad \textit{eq.7.2.1}$$

Where 1.5 $(d_B/l_B)^{3/2}$ is the average increment in dynamic pressure along the sides (super-velocity), and $7(d_B/l_B)^3$ the drag component due to flow separation. d_B/l_B is the diameter length ratio. Bulb l_B/d_B could be optimised quickly using the formula above, but C_F is related to l_B as well (increasing R_N reduces C_F at high R_N). Therefore the optimum length for lower drag in symmetric flow is calculated by designing a series of bulb of the same volume, and measuring area, length and diameter. Such process gave the following graph where an optimum can be seen at a l_B = 3.51 m. The next step was to optimise the cross section.



07.2.3.3 Cross section shape

Cross section shape has two main influence on bulb performance, (for now the approach will be limited to elliptic shapes), firstly changing the cross section from a circle to an ellipse will increase wetted area, therefore drag will increase, secondly as height/beam ratio is increased, VCG_{BULB} gets deeper therefore GZ increases, hence performance increases. This is another case where the Vpp is the tool that will enables the choices to be made.

07.2.3.3.1 Relation Centre of gravity – Increase of drag bulb (VPP study)

A VPP study was required to see the relation between changing the bulb shape to deepen the centre of gravity of the bulb and bulb drag. In general shaping the bulb to deepen its centre of gravity will increase the drag of the bulb. Obviously it is difficult to present a definitive answer since the bulb is required to operate at angle of attack and heel.

The relation needed is: what is the resistance increase that will match the performance increase due to the bulb being deeper, this will give the "envelope" for the bulb design. To simulate an added resistance, WinVPP is not very user friendly, the only way was to use propeller to do the job. (Input using the experimental method: towing tank but was found highly inefficient). This use corresponds well since it obeys a qV^2 for varying speed. The R.M.P was again used to simulate the race. The result is shown in graph 07b (pink curve). For example the increase in performance due to the bulb COG being 5 cm deeper is equal to a decrease of total resistance of 9.3N.

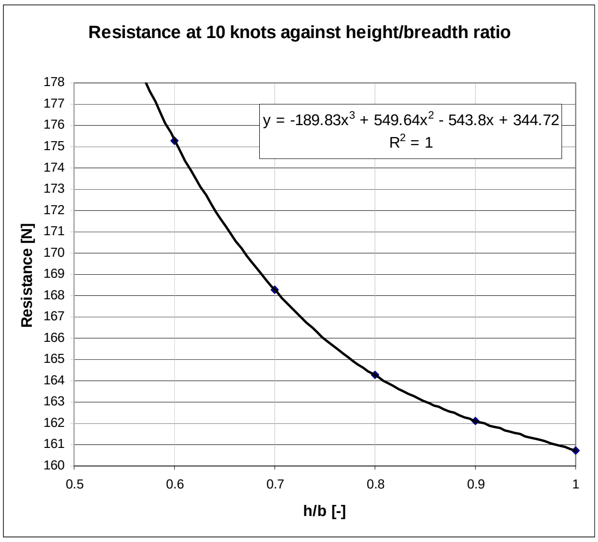
07.2.3.3.2 Height/Breadth ratio

Here again eq. 1 was used, the problem here is that the equation is for rotational bodies, no account is given for an elliptic cross section. A common approach is to use d_B/l_B as the average value of the sideview and topview ratios. The author does not find this approach correct since both the super-velocity term and separation term are term related to $(d_B/l_B)^X$, X being superior to 1. A better approach would be to average the drag.

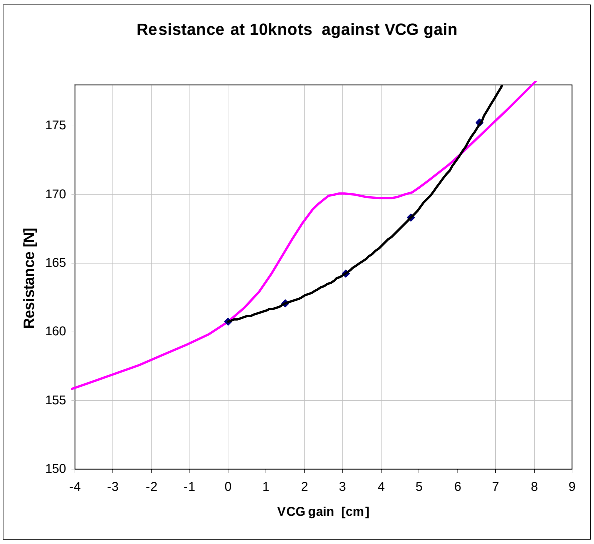
$$C_D=C_F*[(1 + 1.5\,(t_T/l_T)^{3/2} + 7(t_T/l_T)^3) + (1 + 1.5\,(t_S/l_S)^{3/2} + 7(t_S/l_S)^3)\,]\,/\,2 \qquad \textit{eq. 2}$$

Where subscript T and S represents top and sideview respectively

Obviously such equation is still an approximation, but it should give good results for the preliminary design.



Graph 07.2b



Graph 07.2c

Graph 07.2b shows the resistance at 10 knots of same volume bulb where the ratio height over breadth is varied. As expected a decrease of the ratio increases the drag. Graph 07.2c uses the value from graph 07.2b to find when it is beneficial to decrease h/b. As said above the pink curve represents the limit where it is not beneficial to increase the VCG_{BULB}. It can be seen that a gain of 6 cm is still good; it represents a drag of 172.5N and going back to 07.2a, an h/b of 0.63.

Such theory assumes a fix transition point, the thickness distribution does not influence the amount of laminar flow which his not correct. A large beam to height ratio will give a flat bottom and no favourable gradient to promote laminar flow. Therefore a limit has to be put. With the influence of length and h/b more accurate, a better design of the section could be done.

07.2.4 Computational tool aided design

Analogy between sphere/cylinder and bulb/airfoil section can be used to have a deeper look at bulb design. Potential flow theory states that the velocity increment (above free stream velocity) on the sphere is half that of the cylinder ($V_{SPHERE} = V_\infty * 3/2 \sin\theta$; $V_{CYLINDER} = V_\infty * 2 \sin\theta$). This is what is called 3D relieving effect. This relationship stays correct for the flow around a bulb (with circular cross section) and an airfoil section. It is expected that the velocity distribution (in an inviscid case: no boundary layer) around a bulb will be the same to that a foil section having half the thickness length ratio of the bulb. It can be deduced that the influence of the pressure and speed distribution on the transition point will be the same. This relieving effect will decrease the gradients, maintenance of laminar flow will be more difficult but separation will be less likely.

However, the similarity doesn't hold for the boundary layer, on the nose, the boundary layer will expand laterally therefore will be thinner, laminar flow will be easier to maintain. In the pressure recovery, the reverse occurs; the turbulent BL will increase and might separate.

Using CFX to see the influence of the boundary layer on the pressure distribution compared to inviscid case was beneficial, it showed that the in the recovery region the thickening of the BL was an important parameter. Navier Stokes solvers must run all laminar or all turbulent, so no check was done on the influence on the nose section.

The design process was to design an airfoil section that follows the above discussion and remarks. Using past experiment (see Lurie [9] and Carmichael [10]), the expansion of the boundary layer has a big influence on maintaining BL laminar. Therefore it was assumed that a laminar boundary layer could be sustained up to 60% of the bulb length with an acceptable shape. Therefore the section was designed to provide a fair favourable pressure gradient up to 60% of the chord. Then after transition a gentle recovery region was designed to take into account the fact that the boundary layer on the bulb will thicken quickly. The process is shown in diagram 07.2b

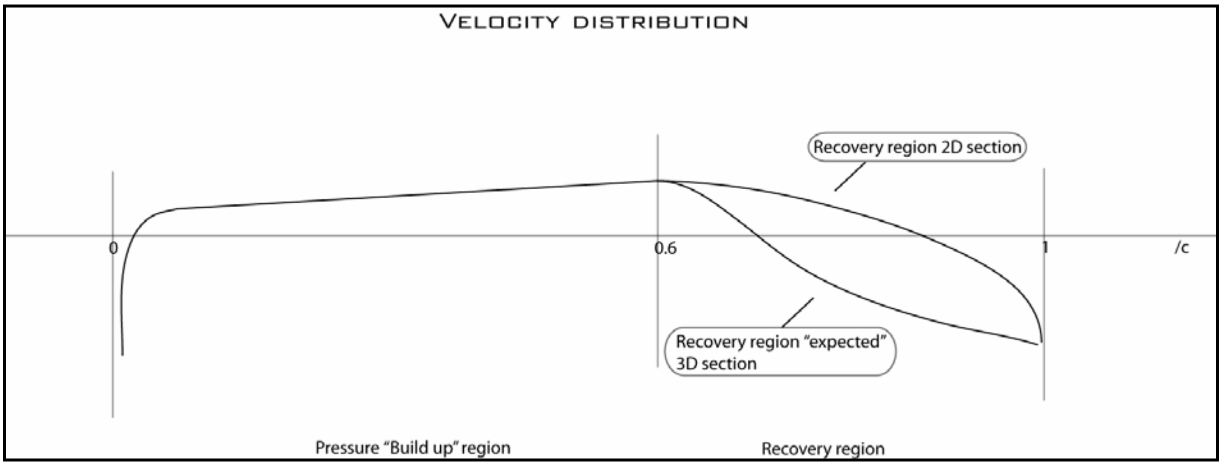


Diagram 07.2b

Pressure gradients on the 'pressure build up region" could have been made more important by decreasing nose radius, but it would cause high speeds on the nose when an important yaw angle is applied, hence a probable loss of laminar flow.

Finally the section was scaled by 2 in the thickness axis to give the bulb side-view shape. From the cross section discussion, a certain amount of elliptic cross section is beneficial (see above discussion), but transition development is not taken into account. Such theory assumes a fix transition point, the thickness distribution does not influence the amount of laminar flow which his not correct. A large beam to height ratio will give a flat bottom and no favourable gradient to promote laminar flow. Therefore a limit has to be put. The increase in laminar flow in an important factor since the decrease in drag is important. It the bottom surface is considered to be 25% of the total area a loss of laminar flow of 40% of this area will give an increase of drag of about 10%. A limit of height / breadth ratio of 0.85 was seen as reasonable.

07.2.5 Conclusion

For future work, it will obviously interesting to study laminar boundary layer development using an e^N approach or similar, but the next version of CFX will have an algorithm to deal with transition. CFD Ranse analysis is still very time consuming job, and therefore expensive. A 3D panel method with a method for boundary layer approach might be an interesting tool to develop since run times and therefore price will be far less.