Steering Towards Flow Optimization

FloEFD™ is an established part of the development process at Robert Bosch Automotive Steering GmbH

By Rolf Haegele, development engineer acoustics / simulation, Robert Bosch Automotive Steering GmbH.

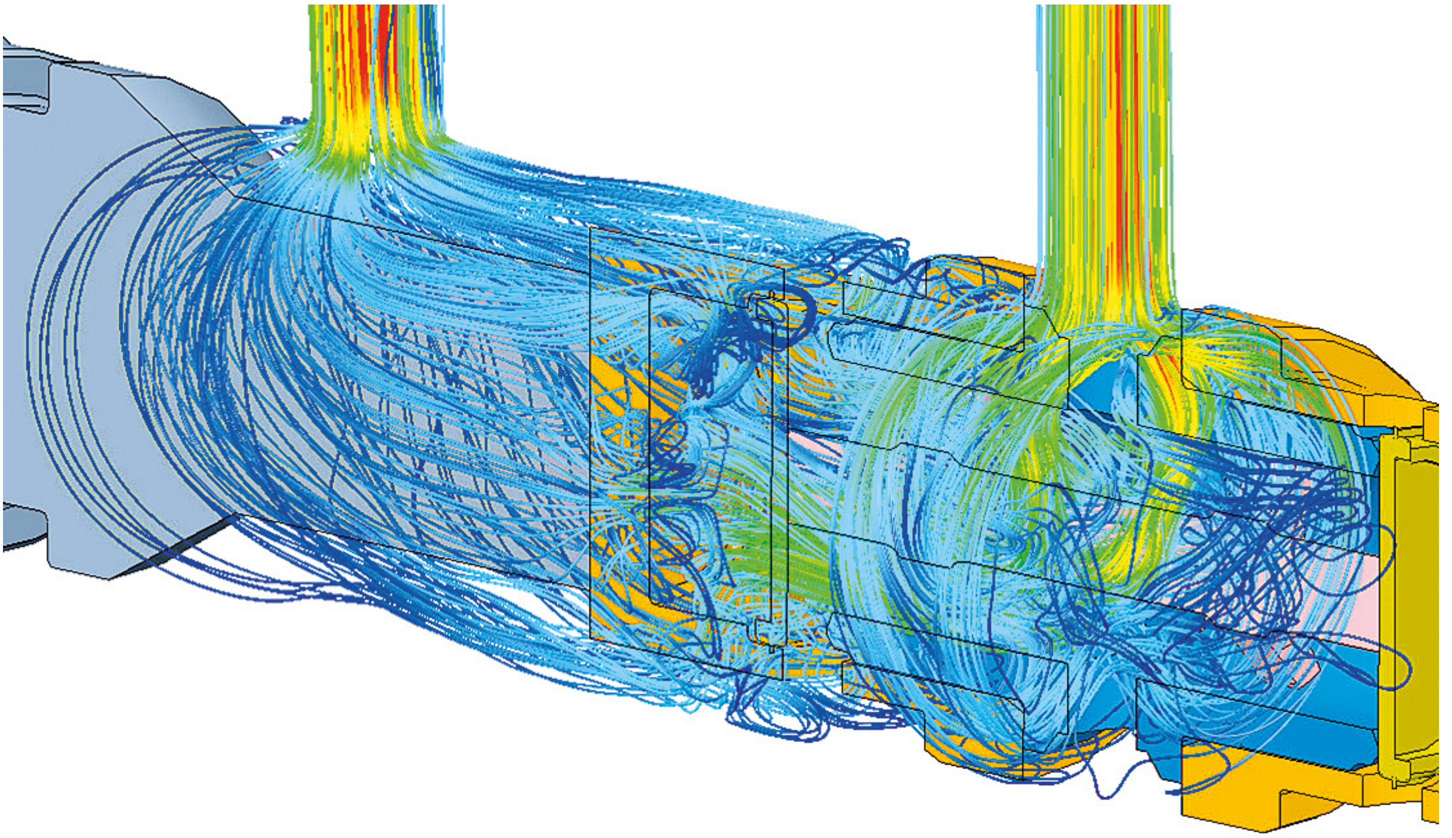

Figure 1. Flow Trajectories Inside the Valve.

The ever-shortening product cycles and decreasing development times in the automotive industry raise the need for up-to-date simulation tools equipped with reliable physical calculation methods. The use of Mentor Graphics' FloEFD Concurrent CFD software enables an evaluation of future automotive components at the earliest possible stage during the development cycle. This allows problem identification and correction when the concept is first evaluated at the feasibility stage of the project.

Steering assistance in commercial vehicles is performed by means of a hydraulic system circuit. The double valve (Figures 1 and 2) is used to supply the feed pump as a control valve. The double valve consists of one inlet and two outlets. The two outlets are opened by pressing against the corresponding spring force depending on the operating condition. Each outlet is opened by undershooting the environment pressure in the requesting partial circuit. A pin controls the distance and the partial circuit is supplied with hydraulic oil after that. To supply the drive with the required flow rate capacity, the pressure drop arising within the valve must be overcome. If the pressure drop is too high, there will be insufficient flow to the drive, and the system will not function correctly. In addition, a lower pressure drop reduces the power consumption of the hydraulic system, and thus the amount of energy required to steer the vehicle, contributing to the overall fuel savings and energy efficiency.

Hence the objective is to supply the required volume flow for each operating case, taking into account the given pressure conditions and keeping the pressure drop at required volume flow rates to a minimum. Simultaneously, cavitation effects have to be avoided. This is a critical consideration because the valve is opening by undercutting 0.95 bar below ambient (initial design shown in Figure 2). This pressure should be prevented from dropping

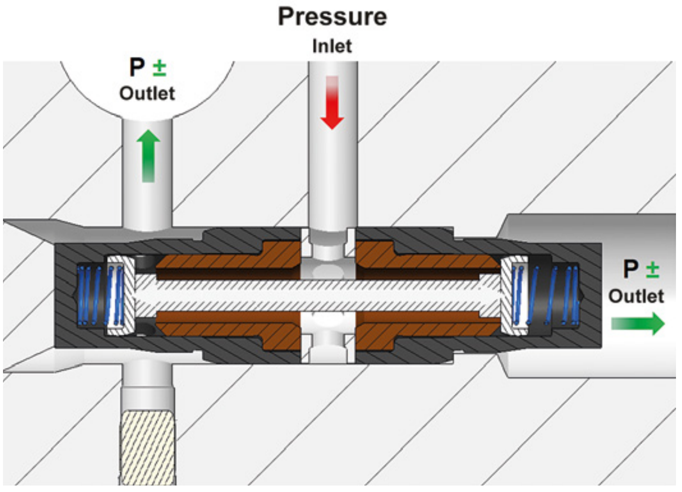

Figure 2. The initial design opens on the left by undershooting the environment pressure.

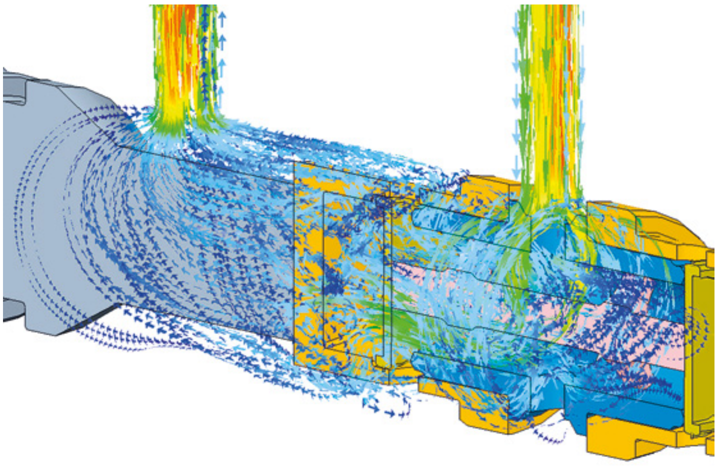

Figure 6. Flow Vectors Inside the Valve.

too low while being sufficiently negative to open the valve. At the same time, external factors constraining the design, such as available installation space and manufacturing capabilities have to be considered.

Several design variations for the double valve were investigated in FloEFD. Aside from the main geometry modifications, detailed changes to individual components and their effects were analyzed. For example, the pin designs shown in figures 3 and 4. The insights gained were incorporated at an early stage in the development of the product concept. The most efficient overall design based on the simulation results (Figures 5 and 6) was manufactured as a prototype and measured in a test setup. The measurements confirmed that the simulation results were accurate, reducing the number of physical prototypes to just one.

Using FloEFD for this application, the available flow rate was increased by approximately 300%, while the pressure drop was reduced by approximately 20% to approximately 0.8 bar below environment pressure. The time saving achieved compared to the conventional prototype-based development process was around five weeks for the application described above. By "frontloading" simulation – simulating each design iteration at the beginning of the development process – the development process is streamlined, and optimized to ensure that each design iteration leads to an improvement in performance.

For FloEFD simulation Bosch Automotive Steering uses native 3D CAD data directly within the PTC Creo Parametric environment. During the modeling process, the fluid space is automatically captured and the mesh is generated from just a few settings within the software. Today Bosch development engineers use the parametric study capability within the PTC Creo environment to quickly prepare FloEFD simulations that are both fast and reliable to run, eliminating the need and cost of integrating with other software, or face the problems associated with using CAD neutral files including loss of parametric information and feature history.

In this case, by frontloading the CFD simulations Bosch Automotive was able to optimize the design of the pin in detail, allowing it to be designed for use across a series of such valves in the future. In addition, with the simulation models being available for future analysis where the impact on the resulting weight and the quantity of material can be evaluated. Therefore cost optimizations have already been achieved at the product concept phase for the series.

"Using FloEFD within our PTC Creo environment has allowed us to front-load full CFD simulation into our design processes, cutting design times and making optimization possible from the very start of the development process. FloEFD has helped us meet today's requirement for short development cycles."

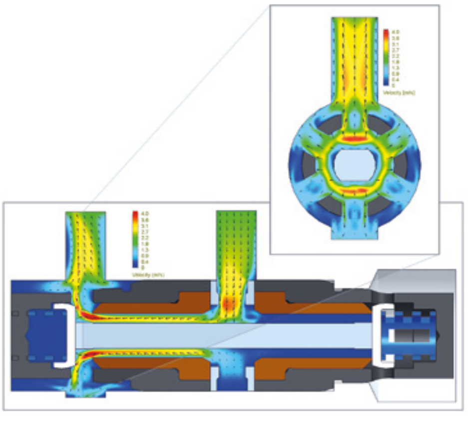

Figure 3. Design variation

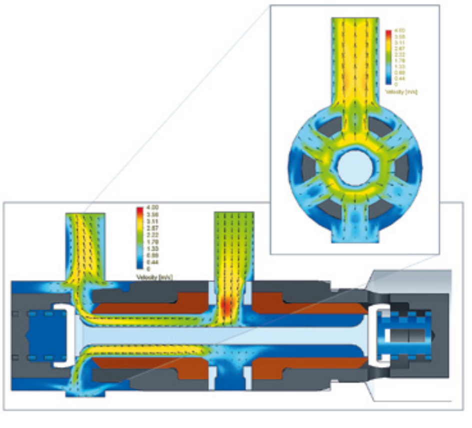

Figure 4. Design variation

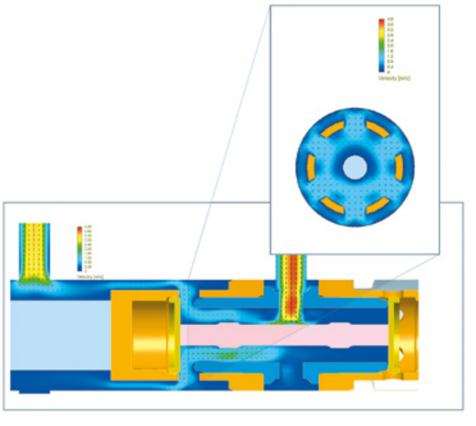

Figure 5. Design variation

Innovation isn't Optional

Mercury Racing® use FloEFD™ in the design of their lastest intercooler filter

By Prasad Tota, Application Engineer, Mentor Graphics

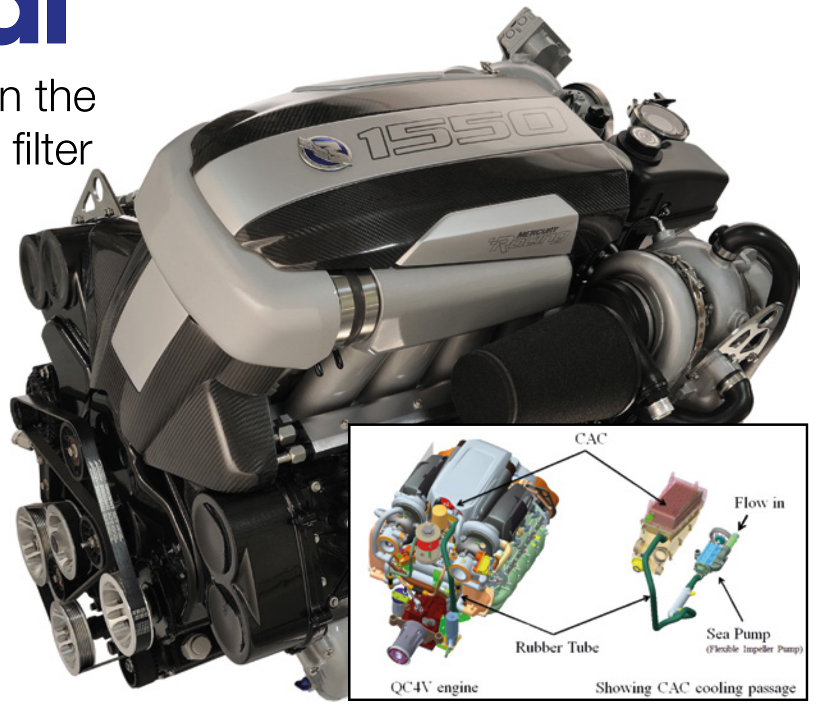

Figure 1. QC4V engine with compressed air cooler (CAC)

Mercury Racing® is known worldwide for its leadership in powerboat racing and production of high performance consumer and race marine products. Founded in the 1970's as a division of Mercury Marine®, Mercury Racing's philosophy of "innovation isn't optional" has served them well and led their customers to winning multiple championships including the Unlimited Offshore World Championship and Abu Dhabi Grand Prix Class 1 World Championship.

Their product line includes sterndrive and outboard engines, drive and propellers. We met up with Hiro Yukioka, Technical Specialist, at Mercury Racing and their latest project, a design study of an intercooler filter on a sterndrive engine- QC4V (figure 1) using FloEFD™ 3D CFD simulation software from Mentor Graphics.

The 9L V8 engine with an ouput of up to 1650hp, has two turbochargers. The engine uses a charge air cooler (CAC) to cool the compressed air from the turbo charger. The CAC uses seawater as a coolant and comes with some challenges owing to the debris it picks up, such as sand, sea shell etc. Mercury Marine has found from field experience that not all seawater boat filtration systems are capable of preventing this debris from accumulating in the CAC.

In the existing design, the size of the passage where seawater enters into the CAC is less than 0.033" (0.84 mm), figure 2. However, it would be a mistake to assume that all the debris that enters the CAC will exit the CAC with the heated water leaving the unit. Depending on the flow velocity, some of the debris entering the CAC can settle or accumulate in the unit. If the water speed inside CAC is too low then debris could settle inside it. At such low velocities the debris accumulation is also influenced by gravity i.e. weight of the particles. FloEFD simulation software was used to study the performance of the existing filtration system and to come up with an improved design.

FloEFD for Creo is a CAD-embedded general purpose CFD software designed for engineers, this focus makes the software easy to use by designers and engineers in an environment that they are already familiar with. The virtual test setup involves a CAD model with a flow inlet where the debris enters with the seawater, travels through a rubber tube into the CAC where some debris gets filtered and finally leaves from the flow outlet. It is important to note that the flow outlet is at a higher elevation than the inlet and hence the pump needs to deliver enough pressure for it to work against the adverse hydrostatic pressure.

At a flowrate of 60 litres/min the velocity inside the tube is about 3 m/s, but the velocity inside the CAC is less than 0.5 m/s. At such low velocities debris would settle inside the CAC. Hiro Yukioka had an idea to use the particle studies feature in FloEFD to virtually visualize if debris particles of a certain size would be carried by the seawater all the way to the outlet or remain in the unit. The particle study was conducted for debris size of 0.2 to 0. 5mm in diameter in increments of 0.1 mm. The particles were fed in at a mass flow rate of 0.01 kg/s which is less than 1% of the fluid mass flow rate. Activating the gravity field in the model accounted for particles settling under their own weight. The images in figure 3 show the particle trajectory colored by velocity magnitude.

Based on the findings, a sea strainer was created with wire mesh positioned around the inside of a cylindrical perforated part (Figure 4). The mesh element should have openings smaller than 0.3 mm and an off the shelf (OTS) wire cloth was chosen that met the criterion.

"If we wish to run a CFD simulation incorporating this new design the number of computational grid cells needed to refine the fine geometry of wire mesh is extremely high and impractical on a typical designer

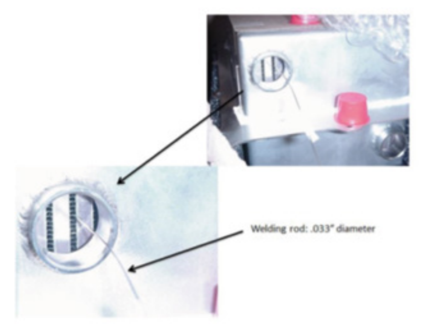

Figure 2. Fluid passage size at CAC entry

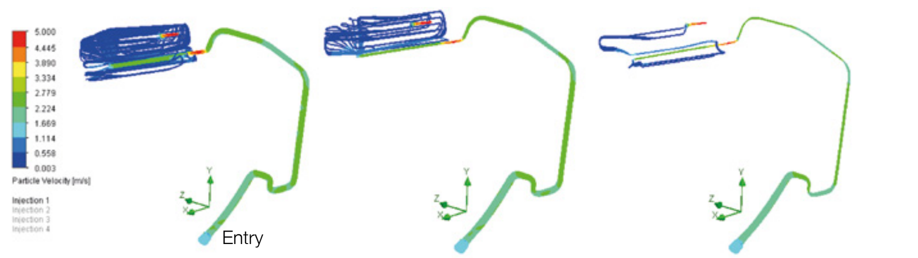

Figure 3. Virtual Debris test, Debris size from left to right (a) 0.2 mm (b) 0.3 mm (c) 0.5 mm

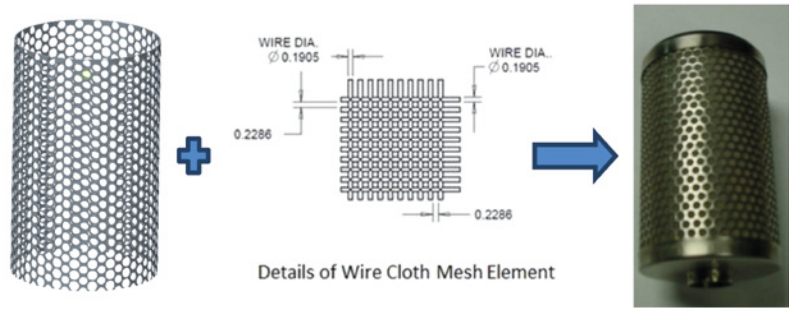

Figure 4. Sea strainer formed with a perforated part and wire mesh rolled on it

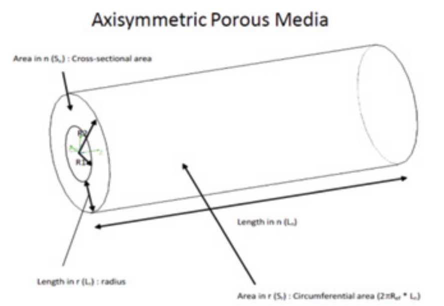

Figure 5. Axisymmetric Porous Media in FloEFD software

workstation. Fortunately FloEFD has a modeling technique where an object can be defined as a porous media which allows flow to go through the media with a pressure loss," said Hiro. A resistance curve was attached to the porous object to emulate the flow vs pressure drop characteristics of the actual device. For this particular geometry an axisymmetric porous media is ideal where the flow loss coefficient (K) can be defined normal to flow direction (r, radial) and along the axis (L, length) of cylinder (Fig.5).

The resistance characteristics of the wire mesh can be either obtained in physical testing or virtual tests set up in FloEFD. In this case a section of wire mesh was tested in a virtual wind tunnel set up within FloEFD to come up with a flow vs. resistance curve that was then attached to the cylindrical part in the overall model for CAC.

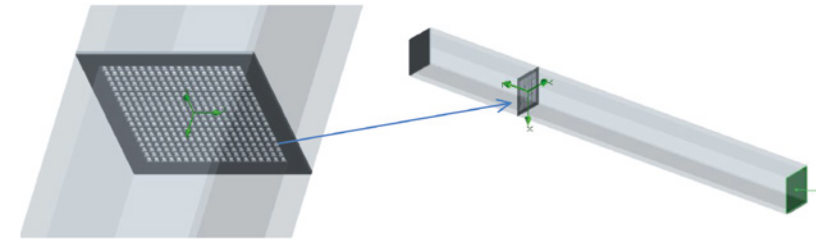

Figure 6. Virtual wind tunnel set up to characterize the wire mesh

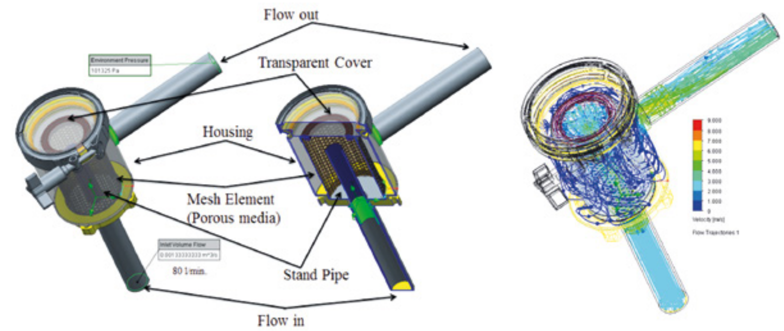

Figure 7. Cross section view of sea strainer and flow trajectories colored by speed (left to right).

The final FloEFD model with the wire mesh incorporated is shown in Figure. 7. The fluid flow simulations showed that the sea strainer results in a pressure drop of 20 kPa at a flowrate of 80 l/min.

The next step was to analyze the effect of debris accumulation on the pressure drop when a part of the overall height in cylindrical volume is completely covered with debris. This was easily tested with small modifications to the FloEFD model where a shell was added, blocking 50% of overall volume and using the parametric study feature in FloEFD this height was varied to 75% and 85%. The results show that there is minimal increase in pressure drop with debris accumulation. (Figure 8)

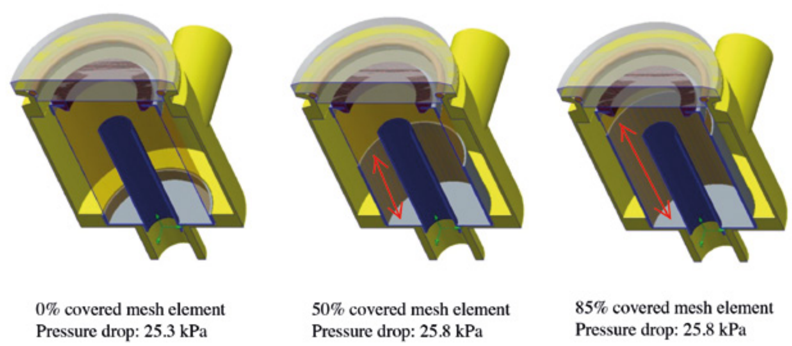

Figure 8. Debris accumulation effects on total pressure drop

A prototype was built to validate the CFD results using thorough hardware testing. Physical tests showed a pressure drop of 25-30 kPa for the sea strainer that is new (no blockage) to 90% blockage to mimic the effects of debris accumulation. These findings are in good agreement with FloEFD predictions of 25-26 kPa for a flowrate of 80 l/min where blockage was varied from 0% to 85%.

Conclusion

After testing the prototype on a test rig for several of Mercury Racing's customers, the redesigned CAC on the field in various conditions, the customer feedback was overwhelmingly positive. Performance was not compromised and the CAC filter was presented at the Miami Boat show in February 2015 and was very well received.

"Without the FloEFD software it would have been very difficult to develop this CAC filter in such a short time. The software is embedded within CAD environment and easy to use, which allowed us to test various ideas and design virtually without the need to create multiple prototypes and several days of physical test." said Hiro Yukioka.

Lastly I would like to express my gratitude to excellent customer support from Mentor Graphics. During this design activity I contacted them several times and every time I was impressed by their professionalism and great technical advice. FloEFD itself is an excellent product and, in my opinion, their support group adds significant value on this product." Hiro Yukioka

Reference

[1] http://www.mercuryracing.com/sterndrives/engines/1550-2/