Proceedings in Manufacturing Systems, Volume 8, Issue 2, 2013 ISSN 2067-9238

ENGINEERING DESIGN AND ANALYSIS OF NOZZLE ASSEMBLY FOR WATER SPORTS TRAINING DEVICE

Georgi TODOROV[1]*, Tsvetozar IVANOV[2], Konstantin KAMBEROV[3], Boyko ALADJEMOV[4]

1) Prof., PhD Eng., Dean, Faculty of Industrial Technologies, Technical University, Sofia, Bulgaria
2) Eng., Research Engineer, Laboratory “CAD/CAM/CAE in Industry”, Technical University, Sofia, Bulgaria
3) PhD Eng., Research Engineer, Laboratory “CAD/CAM/CAE in Industry”, Technical University, Sofia, Bulgaria
4) Eng., Research Engineer, Laboratory “CAD/CAM/CAE in Industry”, Technical University, Sofia, Bulgaria

Abstract: *Virtual engineering usually is applied as to reduce time/money spent for product development stage, but also is a powerful basis to explore details that might otherwise remain undetected. New products development process is demonstrated based on virtual engineering techniques application, forming new, coupled approach. Alternating between CAD enhanced design and numerical techniques (CFD analysis) are in the core of applied approach to design flowboarding training device. This approach enables to implement manufacturing constraints in each step of design development as to produce structure that not only suits technical requirements, but also is cost effective. Simulated physics is also challenging as it concerns two phases (water and air) fluids mixture. The study is focused on the major component of examined structure – nozzle assembly. It should provide sufficient fluid flow at certain pressure losses while correspond to certain requirements for produced water layer. Detailed product design is evolved through initial conception study and characterization to final design exploration, using contemporary tools, applied over common virtual prototype. Major advantages of this approach are mixing (alternating) manufacturing constraints requirements with technical requirements that results in a perspective design, decreasing required time for its development.*

Key words: *CAD, Engineering analysis, CFD, wave-forming generator, nozzle.*

1. INTRODUCTION

During the last fifty years, sport has become a global phenomenon. Today, the sport is everywhere, and its presence is a sign of belonging to global unified civilization. The global flows that pattern world sports have several dimensions, including the technology dimension (the flow between countries of the machinery and equipment produced by corporations and government agencies). It increases the demands on engineering services as well [1].

Flowboarding is a hybrid boardsport, which has been evolving since the 1980’s. The sport combines the progression, maneuvers, and skills from the core action sports of skateboarding, surfing, bodyboarding, and snowboarding and to a lesser extent, wakeboarding and skimboarding. Flowboarders ride on artificial waves that are called "sheet waves" (stationary waves, that does not move forward, and the movement is derived from water flowing over a stationary surface). Powerful pumps project a 60mm layer of water at speeds ranging from 20 MPH to 30 MPH. The water flows up and over surfaces designed to replicate the shape of ocean waves.

Flowboarders get their speed from the energy of the water flowing at them, and can perform basic to sophisticated turns and tricks within a relatively small area.

Product design is a complex decision-making process requiring intense interaction between designers and the designed product. Virtual engineering is an emerging technology that is defined as integrating geometric models and related engineering tools such as analysis, simulation, optimization, and decision making tools, etc., within a computer-generated environment that facilitates multidisciplinary collaborative product development. It provides a means of creating a replica of a physical system in a computer-generated virtual environment. The objective is to allow designers to observe how a system reacts to changes in design and operation without the need to create a physical prototype. Engineers can make appropriate decisions by interacting with the system naturally and exploring details that might otherwise remain undetected [2].

Virtual engineering has a direct application in such case of new product development as flowboarding training device design. In fact, it is needed to integrate virtual reality, simulation and calculations that provide enough data for design development, corresponding to technical requirements as water layer thickness and flow velocity.

Current study demonstrates a successive application of virtual engineering approach for a novel product development that requires combining of manufacturing and performance constraints into design concept. It examines a welded structure design that is required to provide nec-

---

* Technical University – Sofia, 8 “Kl. Ohridski” Blvd.,
Sofia, BULGARIA
Tel.: +359 2 965 25 74;
Fax: +359 2 965 25 74.
E-mail addresses: gdt@tu-sofia.bg (G.Todorov),
tsivanov@3clab.com (Ts. Ivanov),
kkamberov@3clab.com (K. Kamberov),
baladjemov@3clab.com (B. Aladjemov)

essary outgoing fluid flow parameters (fluid flow at certain pressure loss). Major challenge is to provide technologically proved solution at minimal cost, in a field that is complex – Computational Fluid Dynamics (CFD) simulations of phases mixture. A novel product mostly suits an improved, adaptive approach, based on virtual engineering techniques. Applied step-by-step approach introduces conception development, based on consecutive design evolution with alternating simulations and technological expertise (manufacturing constraints) to reach suitable solution.

2. COUPLED APPROACH

Presented product development is tightly connected to CFD simulations. Thus, the major approach specific is related to coupled simulations and design/manufacturing constraints expertise steps. Detailed scheme of approach steps is shown on Fig. 1.

First step – S1 – is to define technical specification, or to determine input parameters for next stages of product development. General conception of the product consists of next several components – as it is shown on Fig. 2: Contained in the tank water is pumped through nozzle assembly. This assembly is in the focus of current research as it forms a thick water layer over the ramp and reaches its upper part. Ramp itself is assumed as predefined as design and is not in the focus of current research. Excess water falls through the positioned on upper side of the system grid and enters again in the tank. Entire system is closed and water recirculates.

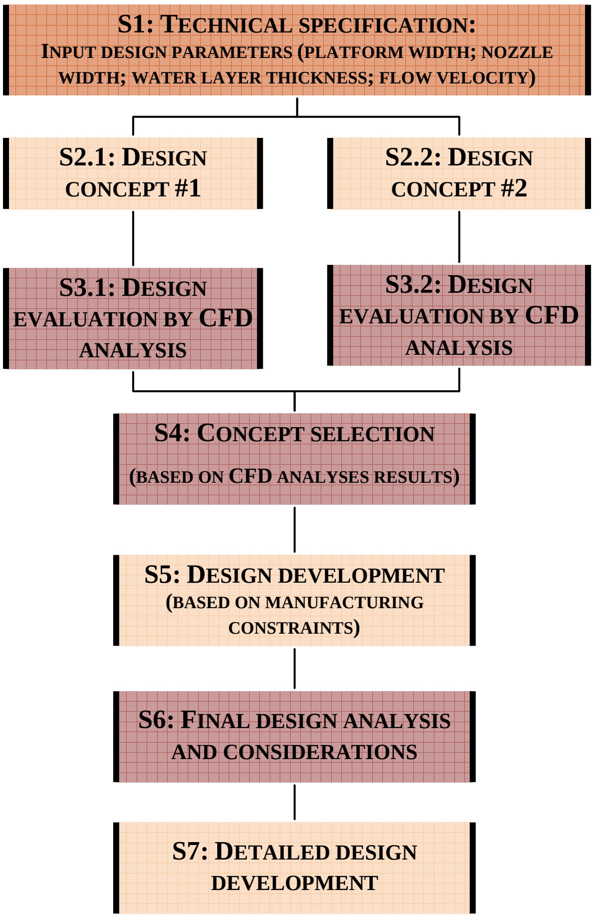


Fig. 1. Approach steps.

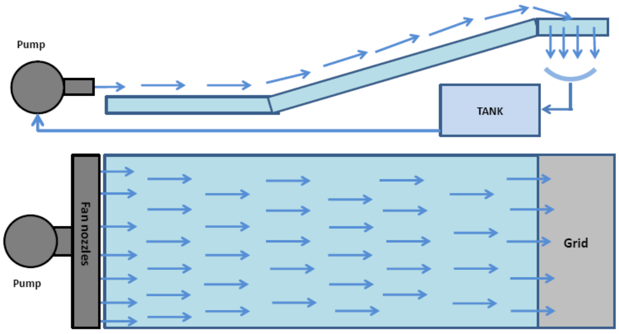


Fig. 2. Overall design conception.

Next, input design parameters are defined:

- pump (supplies 1.5 m^3/s fluid flow);
- single platform width and longitudinal shape;
- nozzle width (4.5 m);
- water layer thickness (40 mm);
- flow velocity (30 km/h).

These parameters definition is based on separate study, which is not subjected in current demonstrative research.

Major component that is to be demonstrated in this study is nozzle assembly design development.

Next two steps are separated in parallel branches as they explore concept variants. S2 concerns of development of designs – 1 and 2. Major target is to evaluate whether to use single (#1) or multiple (#2) nozzles. Step 3 consists of evaluation of these variants by CFD simulations of water flow.

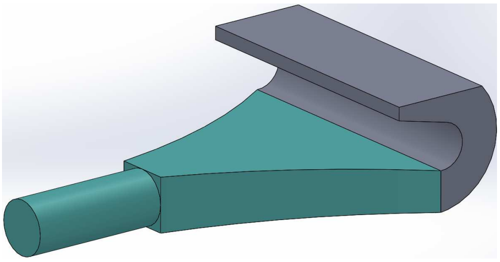
An entirely relative comparison is performed in step S4 through comparing produced pressure losses at certain fluid flow. This step targets to answer which of proposed conceptions would be developed in detail further. It is an important step, where the decision is based on CFD results analysis and study.

Next step – S5 – concentrates in manufacturing constraints implementation. The design is developed also as conception (without detailed presentation), but based on certain restraints that would result in technological solution. All geometry design building is performed over used in preliminary phases 3D model. Edited new – more detailed – virtual prototype is to be used in the next step – S6 – where new CFD analysis is performed. It includes also platform and intends to simulate water dispersion in outlet air media. Several analyses are run as to characterize design at different fluid flows and to obtain pressure – fluid flow characteristic of explored design. Design improvements could be performed in this step, based on obtained simulation results.

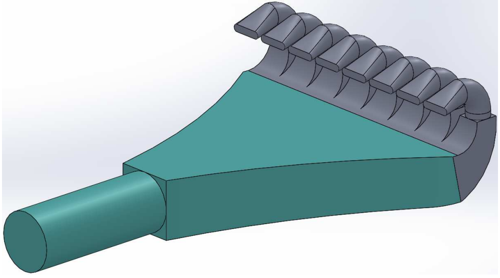
Last step – S7 – is connected to detailed design development. Previously used virtual prototype in S6 is used only as a basis to develop new, manufacturing oriented 3D geometry model. This model needs to be acceptable for further 2D documentation.

3. CONCEPT VARIANTS AND SIMULATIONS

Further, design concepts (S2) and their evaluation by numerical simulations (S3) are presented in detail. Concept #1 explores single nozzle design, as it is shown on Fig. 3 A/, while concept #2 – multiple nozzles (on Fig. 3



A/ Concept variant #1



B/ Concept variant #2

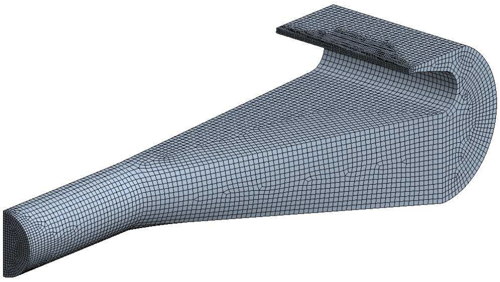
Fig. 3. Examined concept variants.

B/). Both designs use same pump and pump side part, marked in green on the figure below. Manufacturing constraints presumes that the modeled nozzle assembly is a sheet metal part.

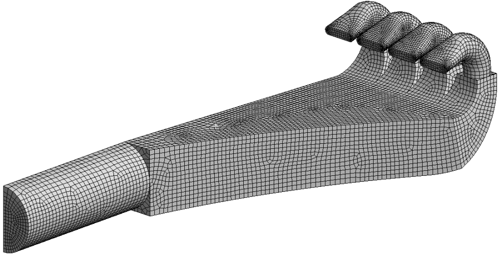
Built numerical CFD models are halves of examined 3D models, because of their symmetry as geometry and boundary conditions, and are presented on Fig. 4. Boundary conditions represent incoming constant water flow of 1.5m^3/s, applied at pump coupling surface (start of tube that replaces the pump), and outlet to atmosphere (nozzles end surface(s)).

A sample of the obtained analysis results is shown on Fig. 5 – in common scale for both examined variants. It is easily seen that concept variant #1 shows higher output velocity – average value of 19 m/s.

Comparison between both examined variants is performed in step 4, where the major output parameter is pressure drop. This is presented graphically on Fig. 6 below. It is seen that single nozzle variant shows less pressure drop, which together with higher outlet velocity makes it better candidate.

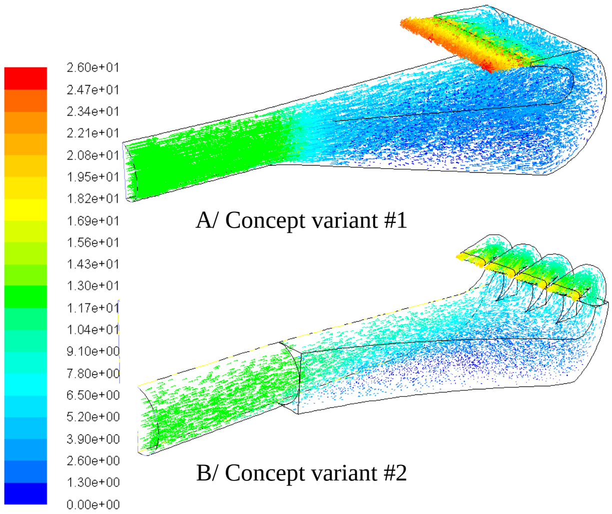


A/ Concept variant #1



B/ Concept variant #2

Fig. 4. CFD models of concept variants.



A/ Concept variant #1

B/ Concept variant #2

Fig. 5. CFD analyses results by concept variants, V, m/s.

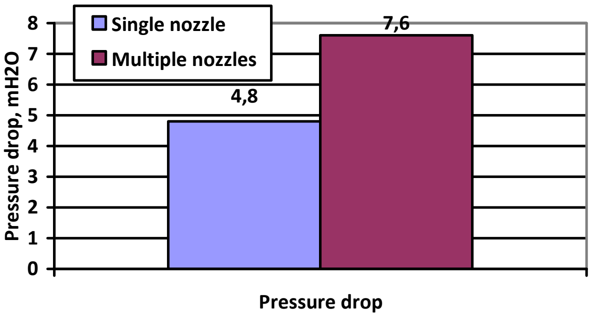


Fig. 6. Comparison between variants by pressure drop.

4. DESIGN DEVELOPMENT

Next step – S5 – involves manufacturing technology requirements implementation and – development of suitable design. Chosen concept variant #1 is redesigned to be produced by sheet metal components – as it is shown on Fig. 7. All specifics for sheet metal parts bending and requirements for welded assembly manufacturing are kept.

This design development uses initially built 3D model of concept #1, used for CFD simulations – as a basis for new geometry shapes and dimensions. Connections to the surrounding structural frame are provided in means of a support by a frame (transparent component on the same figure). Because of its size, the part is divided in two – longitudinally – by ribs (seen on the second part of Fig. 7). This is necessary as to obtain required stiffness of designed part as to withstand hydraulic loads. This dividing rib is an important new element by fluid flow point of view. Several ribs were also added – in nozzle zone – by the same technological and design reasons. Entry zone (transition between circular and rectangular cross section) of distributor is also shaped based on technology requirements for bending.

Another specific, based on manufacturing and design constraints, is the inlet part – long tube is removed, and the nozzle assembly is directly attached to the pump. Also, a flap is added to provide control of outgoing fluid flow (orange component on the figure).

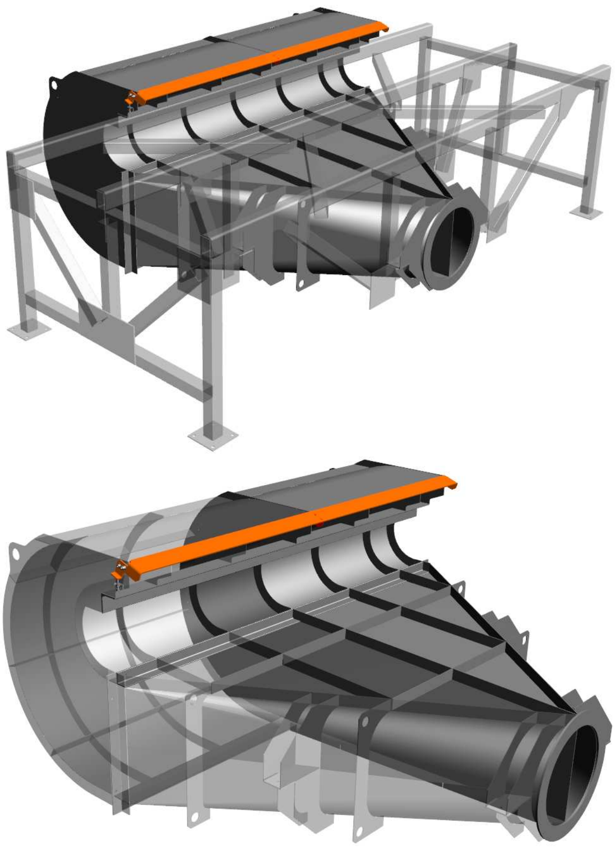


Fig. 7. Design development of single nozzle assembly.

5. FINAL DESIGN ANALYSIS

5.1. Design characterization

All these performed changes in the design reflect also fluid flow behavior, especially added longitudinal ribs and flow division in two separate parts. Developed model is used to obtain its negative (fluid volume) and to prepare input geometry for subsequent CFD simulations. This fully guarantees correspondence between designed and simulated geometries as both 3D models are connected and any change impacts both 3D files. Numerical model includes also more detailed presentation of pump contained water volume. This also could result in worsened flow parameters and needs to be included, because of motor placement inside pump volume.

Nevertheless of existing symmetry, full model is used as to give more detailed view of examined design performance at work hydraulic load conditions. Again, inlet fluid flow of 1.5m^3/s is set and outlet to atmosphere is used as boundary conditions. Meshed CFD model is shown on Fig. 8. Dense mesh is an important prerequisite to obtain more adequate results.

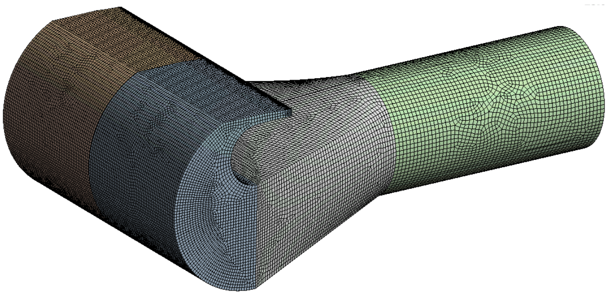


Fig. 8. Meshed model of final design.

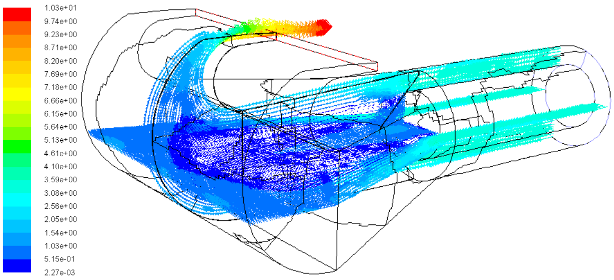


Fig. 9. Velocity vectors fields, V, m/s.

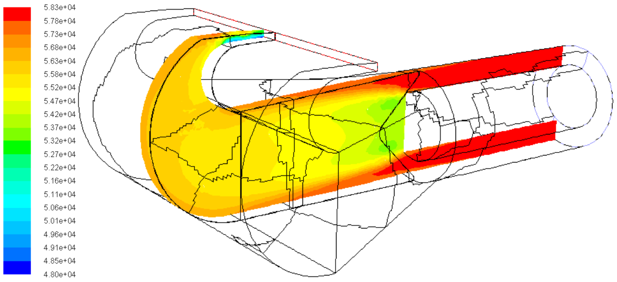


Fig. 10. Total pressure distribution field, p, Pa.

Obtained analysis results are presented by velocity vectors fields in two perpendicular planes on Fig. 9. Important results are pressure drop distribution as to evaluate contribution to its forming by separate design components. Distribution field of total pressure in longitudinal section is shown on Fig. 10. Both figures show low velocities zone after pump, in the middle of distributor volume. This could be improved by adding stream line component after the pump.

Another important point is characterization of pressure components at different incoming pump fluid flows. Such results are obtained by running multiple analyses at proper fluid flow values. This helps to choose pump model and to review entire flow conception. Major target remains to use fluid flow of 1.5 m^3/s and all examined points are higher than this value. These characteristics are shown on Fig. 11.

Pressure components and their contribution to total overall pressure is also important information. Needed total pressure value is formed by next components:

$$p_t = p_{GH} + p_{DP} + p_{NA}\,,$$

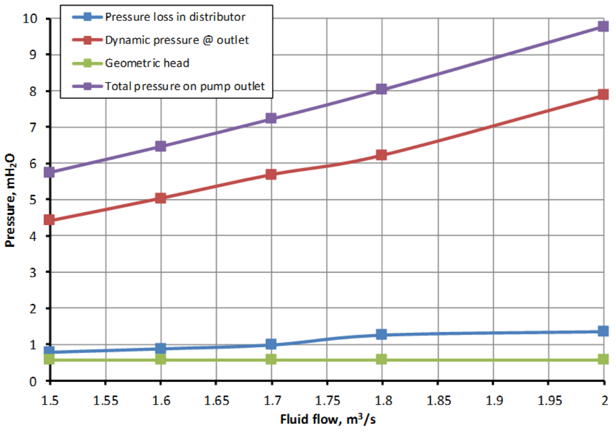


Fig. 11. Pressure loss characteristics.

where: p_t – total pressure needed to be supplied by the pump; p_{GH} – geometric head, pressure needed to bring up the stream; p_{DP} – dynamic pressure, needed to obtain velocity profile on nozzle outlet; p_{NA} – pressure loss in nozzle assembly.

5.2. Overall performance

Overall performance of examined structure is mostly defined by water layer and its velocity. This is simulated using Euler Volume of Fluid (VOF) coupled multiphase model technique in CFD analysis. The VOF model can model two or more fluids by solving a single set of momentum equations and tracking the volume fraction of each of the fluids throughout the domain. This formulation relies on the fact that two or more fluids (or phases) are not interpenetrating. Flow is generally governed by the forces of gravity and inertia. Typical applications include the prediction of jet breakup, the motion of large bubbles in a liquid, and the motion of liquid after a dam break, and the steady or transient tracking of any liquid-gas interface. Thus, it allows evaluating volume of water contained in the air over platform. This is an indicative parameter for water dispersion and together with velocity distribution in the water layer gives important information about the expected device performance.

Used simulation model contains pump, nozzle assembly and platform with air zone above it. Entire model is meshed and no symmetry conditions are used. A fluid flow of 1.5 m^3/s is applied at pump entry and the outlet is set to atmosphere. Gravity is included as to model entire process of water/air interaction. Again, dense mesh is used as to model more correctly water air interfaces and highly nonlinear fluid flow. Special attention is paid on nozzle zone, as it is expected to have higher level of nonlinearity. Built mesh model is presented generally on Fig. 12 below.

Major parameter then is the percentage of water contained in the air and it could be presented in different layers. Because of highly turbulent flow, the mixture is uneven and 50% water is used as a basis for visualization of the flow. The results are presented twice – for the entire modeled region and only over platform – both are shown on Fig. 13.

Analysis results overview shows good performance of the device that corresponds to initially set up parameters (velocity and water layer thickness). A single improvement is needed to be included in final design – marked in first analysis needed additional stream line component after the pump. This is expected to improve performance by decreasing pressure losses inside nozzle assembly. This total pressure component is not decisive for its value, but is also important and needs to be improved. Its implementation is not expected to be problematic as by sheet metal technology specifics.

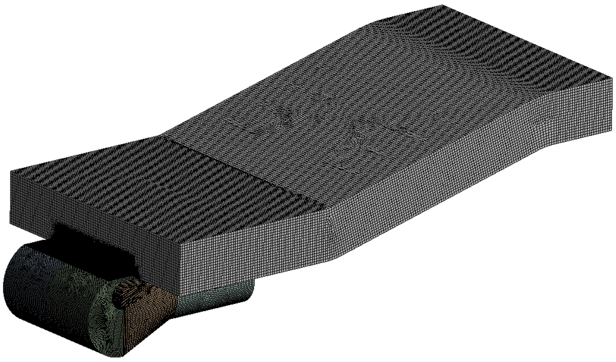
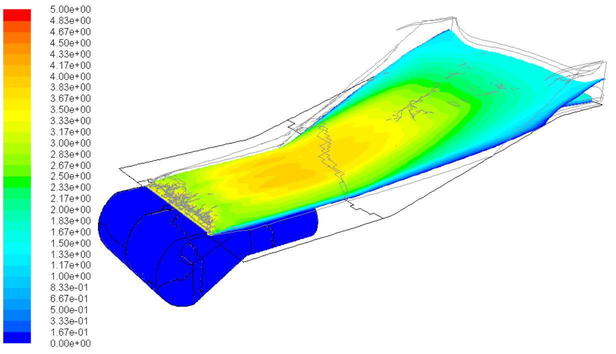
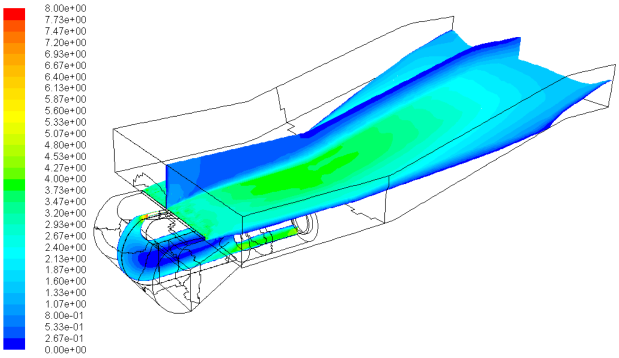


Fig. 12. Mesh model for water mixing simulation.



A/ Nozzle, pump and water 50% layer



B/ Longitudinal cross section and water 50% layer

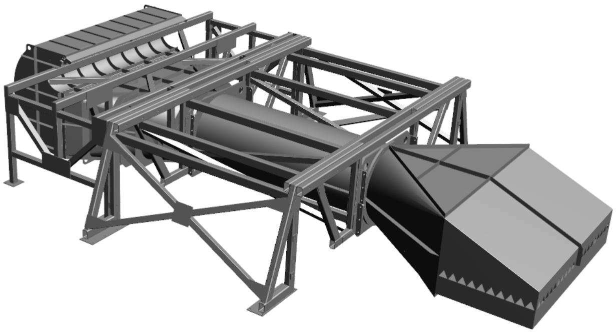
Fig. 13. Overall performance. Velocity distribution, m/s.

6. DETAILED DESIGN DEVELOPMENT

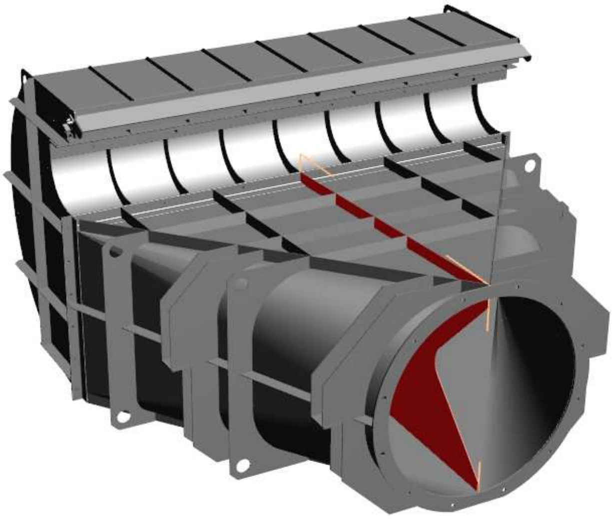
Detailed design was finalized, using final design analyses results and recommendations. It is also compliant to manufacturing constraints and design specifics. Pump assembly includes nozzle subassembly, mounted pump and inlet subassembly and is shown on Fig. 14 A/. It is submerged in the water tank and is supported by the shown on the same figure frame. Stream line component is replaced by mid side rib cut in such a shape as to decrease pressure loss and to improve overall hydraulics performance of nozzle assembly, especially in the zone of distributor. It is shown more detail on Fig. 14 B/.

Additional design improvements (as added external ribs or changes in sheet metal thicknesses) were applied, based on separate structural analyses that were not included in current study. Further improvements could be performed, based on performed tests over prototype, especially in the means of flap angle set up.

Further steps in design development process include documentation (entirely based on developed 3D model), prototyping, design improvement and manufacturing support (using directly 3D models for sheet metal cutting programs and unfold geometry reconstruction). Complete process is based on using common CAD models as to keep overall consistency of entire data. Thus, data manipulation for a complex assembly is enhanced that de-



A/ Pump and nozzle assemblies



B/ Nozzle assembly with modified longitudinal rib.

Fig. 14. Developed detailed design.

creases possibilities for various misalignments as well as impacts product development process overall time.

7. CONCLUSIONS

Proposed coupled approach for new products development process is demonstrated based on virtual engineering techniques application. It is based on alternation between CAD modeling and design and numerical analyses over same model data.

A successful implementation of the proposed approach is demonstrated, based on development of nozzle assembly design for flowboarding training device. Detailed product design is evolved through initial conception study and characterization to final design exploration, using contemporary tools, applied over common virtual prototype. Thus, the advantages of this approach are presented that includes combining manufacturing constraints with technical requirements that results in a perspective design, decreasing required time for its development.

ACKNOWLEDGEMENTS: This research study is performed by the support of project DUNK-01/03 “University Scientific and Research Complex” of National Science Fund, Ministry of Education, Youth and Science, Bulgaria.

REFERENCES

[1] J. Maguire, *Power and Global Sport: Zones of Prestige Emulation and Resistance*, Routledge, London, 2005.

[2] G. Huang, K. Bryden, D. McCorkle, *Interactive Design Using CFD and Virtual Engineering*, 10th AIAA/ISSMO Multidisciplinary Analysis and Optimization Conference, pp.126-133, Albany, September 2004.

[3] K.M.T. Kleefsman, G. Fekken, A.E.P. Veldman, B. Iwanowski, B.Buchner, *A Volume-of-Fluid based simulation method for wave impact problems*, Journal of Computational Physics, No. 206, 2005, pp. 363–393.

[4] C.W. Hirt, B.D. Nichols, *Volume Of Fluid (VOF) Method for the Dynamics of Free Boundaries*, Journal of Computational Physics, 39, pp 201-225, 1981.