Hydrodynamic Design Optimization of an Autonomous Underwater Vehicle based on Response Surface Methodology
Bir Otonom İnsansız Denizaltının Hidrodinamik Tasarımının Cevap Yüzeyi Yöntemi Kullanılarak Optimize Edilmesi

B.Alkan[1] and M. K. İşman[1]

[1] Bursa Technical University, Bursa/Turkey, bugra.alkan@btu.edu.tr
[1] Bursa Technical University, Bursa/Turkey, mkemal.isman@btu.edu.tr

***Abstract*— Autonomous underwater vehicles (AUVs), which are available for scientific, commercial and military purposes, are unmanned marine robots that perform pre-programmed tasks under deeply submerged conditions and large cruising speeds. Extensive use of AUVs in oceanographic applications necessitates the investigation into hydrodynamic hull resistance of the vehicles where it is vital in determination of the power requirements and range of the systems. In this paper, a design method using Computational Fluid Dynamics (CFD) and Response Surface Methodology (RSM) for determining the hull resistance of an AUV is addressed. Hydrodynamic parameters of fully submerged bare hull are investigated in a series of simulations with ANSYS CFX™ commercial CFD code and the response surface being analyzed for specific performance indicators. Moreover, hydrodynamic design optimization of the AUV hull with using RSM to minimize the drag coefficient is carried out, for a given performance criteria and constraints. Results demonstrate that the shape optimization in conceptual design of AUVs is possible by using numerical simulations and RSM.**

***Keywords*— Response surface methodology, optimization, central composite design, autonomous underwater vehicles, computational fluid dynamics**

***Özet*— Bilimsel, ticari ve askeri amaçlarla kullanılan otonom insansız denizaltı araçları (AUV), önceden programlanmış görevleri derin sulatı koşullarında ve yüksek seyir hızlarında gerçekleştiren robotlardır. Bu araçların okyanus araştırmalarında sıklıkla tercih ediliyor olması ve araçların güç gereksinimleri ve sistemin çalışma menzilinin belirlenmesinde önemli rol oynaması nedeniyle, araçların hidrodinamik gövde dirençlerinin incelenmesi bir zorunluluk olarak öne çıkmaktadır. Bu bildiride, aracın gövde direnci, hesaplamalı akışkanlar dinamiği (CFD) ve cevap yüzeyi yöntemi (RSM) kullanılarak araştırılmıştır. Su altındaki çıplak gövdenin hidrodinamik parametreleri ANSYS CFX™ ticari CFD kodu kullanılarak bir dizi benzetimle incelenmiş olup, cevap yüzeyi çeşitli performans kriterleri göz önünde tutularak belirlenmeye çalışılmıştır. Aracın gövde tasarımı belirli tasarım kısıtları ve performans kriterleri göz önünen alınarak RSM ile optimize edilmiştir. Sonuçlardan yola çıkılarak, otonom insansız denizaltı araçlarının konseptsel gövde tasarımları RSM ve sayısal benzetimler kullanılarak iyileştirilebileceği gösterilmiştir.**

Anahtar Kelimeler— Cevap yüzeyi yöntemi, optimizasyon, Merkezi kompozit tasarım, Otonom insansız denizaltı araçları, Hesaplamalı akışkanlar dinamiği

I. INTRODUCTION

An autonomous underwater vehicle (AUV) is a pre-programmed robotic tool that operates through the ocean without any real-time operator control [1]. In the past several decades, applications of these vehicles are spread over many different areas such as; coastal defense, offshore petroleum explorations, maritime archeology and oceanographic studies [2]. Depending on the rapid evolution in AUV technologies, endurance and reliability of the AUVs are significantly increased to operate in harsh marine environments.

Since AUVs carry its power source and most of them are not capable of re-charging their batteries during the task, it is desired that the hull is designed in such a way that the drag is minimized. Hence, the relationship between the propulsion and vehicle hydrodynamics is needed to be taken into consideration, especially in early design steps. However, evaluating the drag forces of a prototype in experimental facilities (towing tanks, circulating water channels or wind tunnels) is very expensive and time consuming procedure. Therefore, optimization of hydrodynamic properties of bare hull forms based on the experimental results is not a practical option for naval engineers and architects [3]. Consequently, selection of the optimal AUV hull form is generally obtained from the results of the previous design experiences and conventional design schemes.

In the last decade, various optimization studies related with the hull hydrodynamics have been accomplished to develop feasible and trustworthy hull resistance estimation especially available for marine vehicles with geometrically complex bodies. Ref. [4] has proposed a simple computational fluid dynamics (CFD) tool to optimize mono-hull ships for which the total drag resistance is minimized. Ref. [5] has considered the problem of optimizing the total resistance of the catamarans. Ref [6] has developed empirical relations to estimate drag forces of deeply submerged AUVs. Ref [7] has

proposed a parametric sensitivity analysis for a ducted AUV by using response surface methodology (RSM) and CFD techniques.

From the above studies, the optimal design of the energy efficient hull is becoming one of the most important challenges in the unmanned maritime technologies.

In this study, the optimal values of the designing parameters of an AUV shape are numerically acquired using quadratic model of response surface methodology, associated with numerical optimization method to reach the minimal drag coefficient of the AUV hull.

The RSM relates to the regression analysis and the statistical design of experiments for constructing the global optimization and is one of the most widely used methods to solve the optimization problem in the manufacturing environments [8]. To achieve the low drag coefficient under the given design constraints, the predictive model for hydrodynamic characteristics will be created using RSM.

This paper is structured as follows. Section 2 describes the CFD modeling and numerical validation. Section 3 presents the proposed RSM and its procedure. Section 4 depicts the variance analysis, optimization implementations and the conformation experiments of selected hull form under given design constraints. Finally, conclusions are presented in Section 5.

II. CFD METHODOLOGY

In this study, all CFD experiments are performed by a standard RSM design called central composite design (CCD). According to this scheme, seven geometrical hull parameters and advance speed of the vehicle are selected as design and operation variables, respectively. In accordance with this approach, the fluid flow for ninety different AUV models are numerically investigated by using commercial finite volume code in ANSYS CFX™ 14.

For CFD calculations, fluid flow is modeled using incompressible (Eq. 1), isothermal Reynolds Averaged Navier-Stokes (RANS) equations (Eq. 2) to determine Cartesian flow field and pressure distribution of the water around bare hull.

$$\frac{\partial \bar{u}_i}{\partial x_1} \tag{1}$$

$$\frac{\partial \overline{u_i}}{\partial t} + \frac{\partial \overline{u_i u_j}}{\partial x_j} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left\{\nu\left(\frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i}\right)\right\} - \frac{\partial \overline{u_i' u_j'}}{\partial x_j} + F_i \tag{2}$$

In these equations, t is time; u_i is the i_{th} component of the fluid velocity; ρ is density of fluid; p is pressure and F_i is the external force of i_{th} element due to the gravity, torque and power.

A. Hull Modeling

The geometry of revolution is divided into four distinct sections: nose, middle, transition and tail. Linear head curve proposed in Ref [9] is applied to the nose segment. According to this streamlined curve equation given in the Eq. (3), radius at a point along the x-axis $r_n(x)$ is defined as a function of the nose coefficients of c_1 and c_2.

$$r_n(x) = c_1\sqrt{x} + \frac{x}{16}c_1(5x^3 - 21x^2 + 53x - 35) + c_2\ \frac{1}{6}x(x-1)^3 + 1 - (x-1)^4 \tag{3}$$

The middle and tail sections of the AUV is shaped based on streamlined curve proposed in the Ref [9] which provides the design flexibility for the optimization process. The shapes of the middle and tail sections are determined from Eq. (4) and (5), respectively:

$$r_m(x) = d \tag{4}$$

$$r_t(x) = d - x\left(\frac{d}{L_t}\right) \tag{5}$$

In these equations, $r_m(x)$ is the radius at a point along the middle section; $r_t(x)$ is the radius at a point along the tail section; d is the maximum hull diameter and L_t is the length of the tail section.

In order to observe the effect of the curve smoothness between middle and tail sections, a corner radius (R) which is defined as a function of the length of the tail and a weighted coefficient (c_3) is used.

By means of this assumption, transition corner is determined from Eq. (6).

$$R = c_3 L_t \tag{6}$$

According to the above descriptions, axis-symmetric hull layout and geometrical parameters of the AUV are illustrated in Fig. 1

B. Domain Boundaries

Solution of the Navier-Stokes equations can only be achievable by using a finite volume domain [10].

In this study, fluid domain displayed in Fig.2 is selected as a half of a cylinder. The length of the domain is 6.5 vehicle lengths (L) and the diameter is 40d.

The length of the tank behind of the AUV (4 L) is longer than the length of the tank forward of the AUV (1.5 L). Therefore, the wake of the AUV can be seen from the analyses. Also, a symmetry plane is used to reduce the computational time.

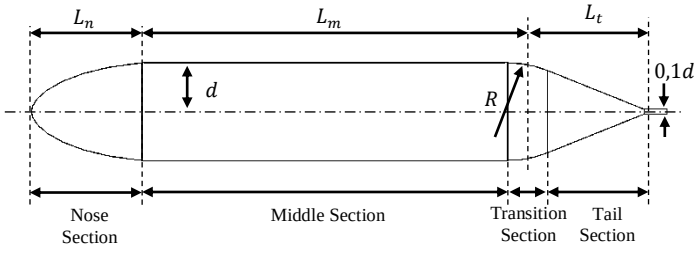

Figure 1: Parameterization of the hull geometry

In order to determine velocity of the AUV, inlet velocity at the front of the water tank was set to be equivalent to advance speed of the vehicle.

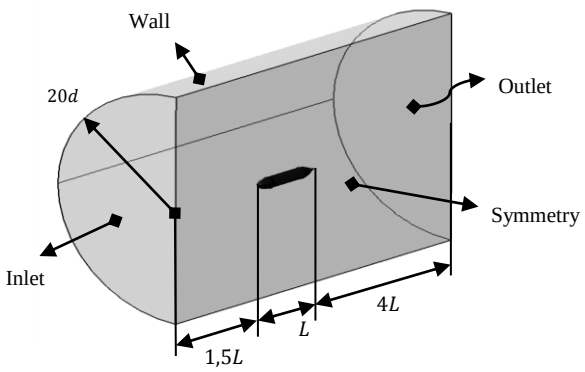

Figure 2: Computational Finite Volume Domain

C. Turbulence Model and Mesh Density

The solver adopted in CFD simulations is the ANSYS CFX™ 14, using the "$(k-\omega)$ shear stress transport" turbulence model detailed in [11].

In near wall region, turbulence wall function model was set to automatic wall function. The size of the boundary layer elements are given in function of the non-dimensional wall distance y^+ which must be close to 1 as possible [12]. Non-dimensional wall distance is defined as;

$$y^+ = \frac{\rho y u^*}{\mu} \tag{7}$$

where, y is the distance to the body surface; $u^*$is the friction velocity*;* ρ is the density of the fluid and μ is the local dynamic viscosity of the fluid.

Mesh density one of the most important factor to achieve a cost effective solution in optimization related numerical analysis studies [7].

Solution accuracy and computational time are heavy depends on the number of cells. Mesh structure and averaged y^+values for selected analyses are given in Fig. 3 and Table 1, respectively.

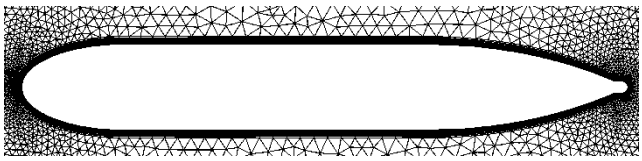

Figure 3: Mesh structure of the model (Run number: 3)
(Model Properties: c_1:2, c_2:-7.5, d: 0.50227m, L_n: 0.5m, L_m: 1.75m, L_t: 0.55m, c_3: 0.8)

D. Computer Simulation

Simulations were run on a desktop pc running 64 bit Windows 7 with 16 GB of RAM.

Solutions were carried out using the high resolution advection scheme. The residual error was reduced to fourth orders of magnitude.

E. Model Verification

In highly submerged conditions, the drag experienced by an axis-symmetric AUV travelling in axial direction is a direct result of the water viscosity [10].

The viscous effects can be divided into two components: the skin friction drag and the form drag. To estimate these effects, empirical relationships are available which allow initial estimates of the drag.

Table 1: Bare hull mesh properties for selected cases.

Selected Cases	Geometric properties of the model	Total number of elements	Averaged y^+ over hull
1 (*Best y^+result among experiments*)	*c_1:2, c_2:-15, d: 0.1m, L_n: 0.2m, L_m: 3m, L_t: 0.8m, c_3: 0.1*	789254	0.999292
2	*c_1:2, c_2:-7.5, d: 0.25m, L_n: 0.4m, L_m: 1.75m, L_t: 0.55m, c_3: 0.8*	712455	1.064578
3	*c_1:4, c_2:0, d: 0.1m, L_n: 0.8m, L_m: 3m, L_t: 0.8m, c_3: 0.1*	782425	0.987907
4 (Worst y^+ result among experiments)	*c_1:4, c_2:-15, d: 0.4m, L_n: 0.8m, L_m: 3m, L_t: 0.3m, c_3: 0.1*	802457	1.299578

ITTC 57 correlation line is an empirical formulation which is accepted as a standard by ITTC. This formulation is commonly used for the estimation of the skin friction in surface ship designs. According to this formulation, skin friction (C_f) component of the viscous drag can be found as a function of Reynolds number (R_N) [13].

$$C_f = \frac{0.075}{((log(R_N))-2)^2} \tag{8}$$

To complete the estimation of the viscous drag coefficient, this coefficient is multiplied by a from factor ($1+k$) which is a function of the hull parameters. Ref [10] proposed a form factor equation which includes vessel length (L) and diameter (D).

$$(1+k) = 1 + 1.5(\frac{D}{L})^{\frac{3}{2}} + 7(\frac{D}{L})^3 \tag{9}$$

Axial drag force acting on an AUV operating at a constant speed can be estimated by using Eq. (10) proposed in Ref [7].

$$R_f = 0.5\,\rho\,C_f\,A_f|u|\,u \tag{10}$$

In this equation, C_f is coefficient of the friction drag, ρ is the density of water, A_f is the submerged front area of the vehicle and u is the advance speed of the vehicle. According

to Ref [7], SST model have good agreement with the ITTC-57 estimation.

According to these recommendations, ITTC-57 estimation was selected as a tool to verify CFD results of proposed models. Fig 4 compares numerical results and ITTC-57 estimations for the model's axial drag force.

Verified model's principal dimensions;

- Total vehicle length 2.8 m
- Maximum hull diameter 0.25 m
- Speed range 0.5 to 3 m/s
- y^+ check ~ 1
- Number of elements 798321
- Selected turbulence model $k-\omega$ SST

According to these results, drag force predictions of the CFD and ITTC-57 correlation line have a good agreement. Therefore, CFD method can be regarded as a valid and reliable tool for the optimization analysis.

F. Design of Experiments

Eight different design and operation parameters have been investigated including maximum hull diameter, length of the nose, middle and tail sections, transition coefficient, nose coefficients and advance speed of the vehicle. Table 2 shows the levels of the designing parameters. A total of 90 experiments were conducted at the designated conditions based on the face-centered CCD. The drag coefficient C_d is investigated as a desired response.

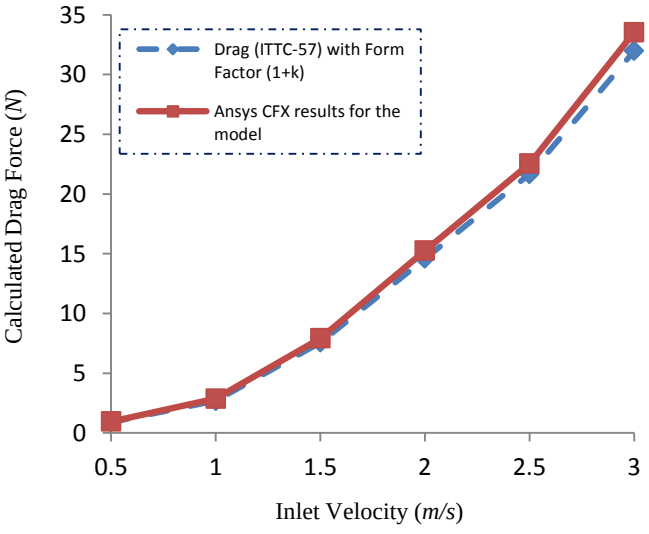

Figure 4: Comparison of the axial drag force for selected model

Each combination of experiments will be repeated two times under the same conditions at different times to a acquire mode accurate results. Some of models used in the numerical experiments are illustrated in the Fig. 5.

III. RESPONSE SURFACE METHODOLOGY

RSM is an empirical design approach for establishing the relationship between design variables and responses within the predetermined design constrains.

It is a sequential experimentation scheme for developing and optimizing the factual models [3]. Along these lines, RSM is a statistical approach that is useful for the modeling and analysis of problems in which the response is affected by several design variables.

As a consequence of using the design of experiments and applying regression analysis, the modeling of the desired responses related with the independent design variables can be achieved. In RSM, the quantitative form of relationship between the response and design variables can be defined as follows;

$$y_{(x)} = F(x_1, x_2, x_3, x_4, \dots, x_n) \mp \varepsilon \qquad (11)$$

where $y_{(x)}$ is the desired response; F is the response surface; $x_1, x_2, x_3, x_4, \dots, x_n$ are the design variables; and ε is the fitting error.

In this study, the collection of CFD results adopts the face-centered CCD and the approximation of f will be proposed using the fitted second-order polynomial regression model which is called the quadratic model. The quadratic model of f can be written as follows;

$$F = a_0 + \sum_{i=1}^{n} a_i x_i + \sum_{i=1}^{n} a_{ii} x_i^2 + \sum_{i<j}^{n} a_{ij} x_i x_j + \varepsilon \qquad (12)$$

where a_i represents the linear effect of x_i, a_{ii} represents the quadratic effect of x_i and a_{ij} reveals the linear-by-linear interaction between x_i and x_j.

Table 2: Scheme of design parameters and their levels.

Symbol	Factor	Unit	Levels	
			Low (-1)	High (+1)
A	Nose coefficient 1(c_1)	m	0	4
B	Nose coefficient 2 (c_2)	m	-15	0
C	Maximum hull diameter (d)	m	0.1	0.4
D	Length of the nose section (L_n)	m	0.2	0.8
E	Length of the middle section (L_m)	-	0.5	3
F	Length of the tail section (L_t)	-	0.3	0.8
G	Transition coefficient (c_3)	-	0.1	1.5
H	Advance velocity of vehicle (u)	m/s	0.5	3

The RSM sequential procedure and its procedure for the determination of the design parameters with optimal performance characteristic including seven steps are summarized as follows:

- Defining the independent design variables and drag coefficients of the bare hull with the design constraints,
- Adopting the face-centered CCD to plan the experimental design,
- Performing the regression analysis with the quadratic model of response surface (F),
- Calculating the statistical analysis of variance (ANOVA) for the independent input variables and to find which parameter significantly affects the axial drag force,
- Obtaining the optimal design parameters within the design constraints,
- Performing the confirmation analysis.

IV. Results and Discussion

According to the DOE results, the conformation of the goodness of the fit is greatly required to analyze the numerical runs. The analysis of variance (ANOVA) is usually applied to summarize the performed tests.

A. ANOVA Analysis

According to backward swept ANOVA analysis, "*Prob.>F*" value of the model is less than 0.0001. This indicates that the model is considered to be statistically significant, which is desirable as it demonstrates that the terms in the model have a significant effect on the response.

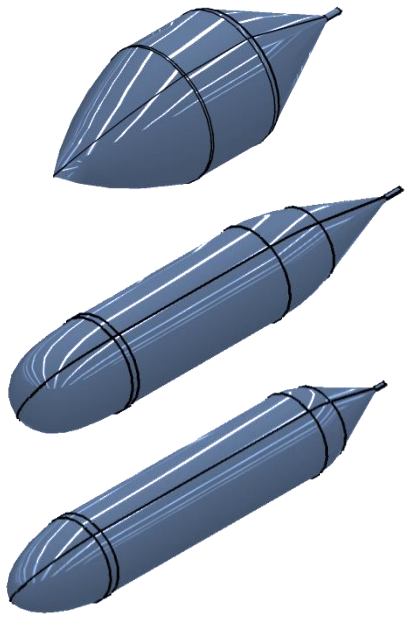

Fig. 5 Selected DOE models
(Top: c_1*:0,* c_2*:0,* d*: 0.4m,* L_n*: 0.8m,* L_m*: 0.5m,* L_t*: 0.8m,* c_3*: 0.1,*
Middle: c_1*:4,* c_2*:0,* d*: 0.1m,* L_n*: 0.2m,* L_m*: 0.5m,* L_t*: 0.3m,* c_3*: 1.5,*
Bottom: c_1*:2,* c_2*:-7.5,* d*: 0.1m,* L_n*: 0.2m,* L_m*: 0.5m,* L_t*: 0.3m,* c_3*: 0.8*)

In the same manner, the main effect of factors A, C, D, E, F and interactions of the factors; AB, AC, AD, AF, BG, CD, CE, CF, CG, DE, DG, DH, EF, FG, A^2, D^2, A^2C, A^2D, A^2E, A^2H and AB^2 are significant model terms. The other model terms can be regarded as not significant due to their "*Prob.>F*" value that is greater than 0.0500.

B. Optimization of Design Parameters

Optimization of AUV with the design constrains of space limitation in this study is to find the optimal values of design parameters for minimizing the drag coefficient of the body.

A design optimization procedure was therefore conducted to find optimal value of the design variables (design parameters:c_1, c_2, d, L_n, L_m, L_t, c_3 and operation parameter: u) considering given constraints (given in Table 3) and the drag coefficient of the bare hull.

C. Result of the Design Optimization

In the given initial conditions and zero lift angles, numerical optimization method optimizes the shape of a certain AUV.

According to the optimization result, best suitable design parameter values for given design constraints are given in the Table 4.

D. Confirmation Experiments

In order to verify adequacy of the quadratic model obtained, three confirmation run experiments are performed. The data from the confirmation runs and their comparisons with the predicted values for the drag coefficient are listed in the Table 5.

From these analyses, the residual and percentage error calculated are relatively small. All the experimental values for confirmation run are within the 90% prediction interval. Therefore, quadratic model achieved is considered to be good but improvement is still required for the obtained model.

Table 3: Design Constraints.

A	Nose coefficient 1(c_1)	In range of 2 to 4
B	Nose coefficient 2 (c_2)	In range of -15 to -7.5
C	Maximum hull diameter (d)	Not less than 0.15 m
D	Length of the nose section (L_n)	Not less than 0.4 m
E	Length of the middle section (L_m)	Not less than 1.25 m
F	Length of the tail section (L_t)	Not less than 0.375 m
G	Transition coefficient (c_3)	In range of 0.5 to 1.5
H	Advance velocity of vehicle (u)	Equal to 1.75 m/s

Table 4: Optimized values of the designing parameters.

Factor	Original Value	Optimized Value
Nose coefficient 1(c_1)	2	3.58
Nose coefficient 2 (c_2)	-7.5	-11.18
Maximum hull diameter (d)	0.25 m	0.189 m
Length of the nose section (L_n)	0.5 m	0.474m
Length of the middle section (L_m)	1.75 m	1.43 m
Length of the tail section (L_t)	0.55 m	0.379 m
Transition coefficient (c_3)	0.8	0.61
Advance velocity of vehicle (u)	1.75 m/s	1.75 m/s
Drag Coefficient	0.108857	0.044562
Hydrodynamic Performance Enhancement		% 59.06

Table 5: Confirmation experiments.

Test Case	Predicted Value	Actual Value	Error
c_1:4, c_2:0, d: 0.1m, L_n: 0.2m, L_m: 0.5m, L_t: 0.8m, c_3: 1.5 u:0.5	0.1367	0.1380	% 0.94
c_1:2, c_2:-7.5, d: 0.25m, L_n: 0.5m, L_m: 1.75m, L_t: 0.55m, c_3: 0.8 u:1.75	0.1186	0.1105	% 7.33
c_1:4, c_2:-15, d: 0.1m, L_n: 0.2m, L_m: 3m, L_t: 0.3m, c_3: 0.1 u:3	0.2601	0.2540	% 2.40

V. CONCLUSION

An optimization methodology using RSM is applied to minimize the drag coefficient of a streamline profiled AUV with the design constraints in this study. The following conclusions are obtained.

- RSM based design method, one of the several optimization tool in naval architecture, is employed for finding the optimal values of seven design parameters and one operation parameter that would produce a design with the minimum drag coefficient.
- According to the RSM, the main and interaction effects of A (Nose coefficient 1), C (Maximum hull diameter), D (Length of the nose section), E (Length of the middle section), F (Length of the tail section) play significant role over the hydro dynamical hull resistance.
- It is also observed that drag response is highly non-linear. Because of this reason, relatively high residual errors are obtained during confirmation tests.
- In order to reduce the prediction errors of the obtained model and eliminate non-linearity, RSM study based on the radial basis function (RBF) is required to be performed.

ACKNOWLEDGMENT

Authors would like to thank to Res. Asst. Emre Demirci and Emre Doruk of Bursa Technical University for their help during the CFD experiments.

REFERENCES

[1] A. Budiyono, "*Advances in unmanned underwater vehicle technologies: Modeling control and guidance perspective"*. Indian Journal of Marine Sciences, Vol 38(3), pp 282-295, 2009.

[2] P. Wang, B. Song, Y Wang and L Zhang, "*Application of concurrent subspace design to shape design of autonomous underwater vehicle*", 8th ACIS International Conference on Software Engineering, Artificial Intelligence, Networking and Parallel/Distributed Computing, 2007

[3] A. Alvarez, D. Boyle, R.B. Eaton, E. Lynch, T. Stepanchick and R. Yon, "*Hull hydrodynamic optimization of autonomous underwater vehicles operating at snorkeling depth*," Ocean Engineering, 36(1), pp 105-112

[4] S. Percival, D. Hendrix and F. Noblesse, "*Hydrodynamic optimization od ship hull forms*," Applied Ocean Research, 23, pp 337-355, 2001

[5] Subramanian, V.A., Patrick Joy, "*A method for rapid hull form development and resistance estimation of catamarans*," International Journal of Marine Technology, Vol.38, Number 1, Spring 2004, 5 – 11

[6] Barros, E. A., Progress towards a method for predicting AUV derivatives, IFAC Conference on Manuevering and Control of Marine Craft, 2006

[7] T. Joung, K. Sammut, F. He and S. Lee, "*Shape optimization of an autonomous underwater vehicle with a ducted propeller using computational fluid dynamics analysis*," International Journal of Naval Ocean Engineering, 4-44, 56, 2012

[8] M. Vishwakarma, V. Parashar and V. K. Khare, "*Geometrical characteristics of streamlined shapes. Technical Report 2962*," International Journal of Engineering Research and Applications, Vol.2, Issue 6, pp 787-792, 2012

[9] P.S. Granville, "*Optimization and regression analysis of surface roughness for electric discharge machining of EN-19 Alloy steel using tungsten copper electrode*," The David W. Taylor Naval Ship Research and Development Centre: DTNSRDC, Bethesda, M.D.

[10] Phillips, A., Furlong, M. and Turnock, S.R., "*The use of computational fluid dynamics to access the hull resistance of concept autonomous underwater vehicles*," Ocean IEEE, 2007

[11] Menter, F.R, "*Zonal two-equation k-ω turbulence model for aerodynamic flows*," AIAA Paper 1993-2906, 1993.

[12] CFX, ANSYS CFX,, Release 10.0, ANSYS LTD., 2005

[13] ITTC, "*ITTC Quality Manual – CFD General Uncertinity Analysisin CFD Guildines for RANS Codes"*, 1999