WIND SAIL ANALYSIS USING COMPUTATIONAL FLUID DYNAMICS SIMULATION

A. Sulisetyono [1], A. Nasirudin[2], and FA Wibowo[3]

[1]Department of Naval Architecture And Shipbuilding Engineering, Institut Teknologi Sepuluh Nopember, Surabaya, sulisea@na.its.ac.id

[2]Department of Naval Architecture And Shipbuilding Engineering, Institut Teknologi Sepuluh Nopember, Surabaya, anasirudin@na.its.ac.id

[3]Department of Naval Architecture And Shipbuilding Engineering, Institut Teknologi Sepuluh Nopember, Surabaya,

ABSTRACT

This paper described the behavior of rigid and untwisted sail model in the fluid dynamic analysis. The two and three dimensional wind sail was modeled numerically using CFD technique to determine the influence of the main sail design variables such as draft, camber and angle of attack due to the forces coefficients of sail. The 10-sail models were developed, and each sail model was simulated at the variation of the angle of attack such as 15°, 20° and 25° respectively. For certain case, the numerical results were validated by the results of wind tunnel model testing in term of drag and lift coefficient. The simulation showed the design of sail models, which generated maximal efficiency and the most optimal thrust force, were the sail models with camber value 20% and 45% of draft.

Keywords: *sail design, computational fluid dynamics, wind tunnel testing*

1. INTRODUCTION

One of the shipbuilding technologies developed is a ship propulsion system by using renewable energy sources. This is motivated by the increasingly prices of oil which affects the operational cost of vessel, and the impact of burning fuel by ship to the pollution in the marine environment. The energy crisis in the world can be anticipated by the reducing oil consumption in shipping industry, and begin to switch to the renewable energy sources such as sunlight and wind.

Indonesia has developed the utilization of renewable energy as a driving force (propulsion) on the ship “KLM. Maruta Jaya” which was a 900 DWT general cargo vessels using sail technology as the prime propulsion, and a DC motor as a means of driving aids. Ship "Maruta Jaya" was the concept of energy-saving ships that was collaboration project between the Governments of Indonesia and Germany in the 1980s (Wiriadidjaja, 1997).

Sail design is an important part in the generating of ship speed optimally. The study was conducted by Kartika, (1996) shows the sail performance of 900 DWT ships “Maruta Jaya” was not optimized yet in generating the sail trust. The development of sail design in order to improve the performance of sail is still necessary.

Commonly, the thrust of sail was influenced by the natural conditions such as, velocity, density of air; operator skill such as position of the sail against the wind direction/angle of attack); and the geometry of the sail such as, wide sail shape, camber and draft.

This paper is proposed to analyze the sail models that was developed based on the dimensions of the main sail “KLM Maruta Jaya”. Numerical analysis was conducted by the numerical approach of CFD (Computational Fluid Dynamics) as well as the wind tunnel testing. Furthermore, this paper described the effect of variable shape of the sail (camber and draft) and the angle of attack against the thrust.

The variations of draft and camber of sail were determined based on the research performed by (Collie, SJ, 2006) and (Nasirudin, A. 2002). There were 10 types of sail modeled in which have the variation value of camber and draft. The angle of attack was determined in the range of angle 10°-20° follow Marchaj (1982).

The simulation results were obtained by CFD techniques acceptable while those were validated by the results of sail model testing in wind tunnel.

2. COMPUTATIONAL AND EXPERIMENTAL METHOD

2.1 Computational Setup

The cross-flow problems of the 2D sail was solved by using ANSYS software. In this numerical simulation of the flow was assumed turbulent, steady, adiabatic and incompressible. Reynolds Number of the air as the working fluid was 4.33 10^6, with the fluid density (ρ) = 1.225 kg/m3 and viscosity (μ) = 1.79 x 10^{-5} kg/m sec. The flow was assumed turbulent when the Reynold Number value of 10^6<Re<10^7 (SJ Collie, 2006), and the turbulence model standard k-ε was selected with the pressure correction was 0,001.

The sail size was modeled in the CFD geometry with certain value of chord (*c*), the length of the peak curvature (Y_{max}), and lies on a flat surface (X_d). The Sail was a rectangular with dimensions of *l* (p) = 32 m, *c* (chord) = 12.31 m, *t* (thickness) = 0.2 m, *A* (area) = 393.92m^2, *AR* (aspect ratio) = 2.6.

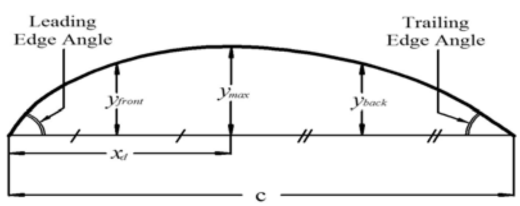


Figure 1. Cross-section of sail (S. J. Collie, 2006)

Table 1 Geometry of sail in 2-D

No.	Model	Camber and Draft (%)	Geometry	
			Ymax (m)	Xd (m)
0	O	cb0, dr0	0	-
1	A	cb5, dr45	0.6155	5.5395
2	B	cb5, dr50	0.6155	6.155
3	C	cb5, dr55	0.6155	6.7705
4	D	cb10, dr45	1.231	5.5395
5	E	cb10, dr50	1.231	6.155
6	F	cb10, dr55	1.231	6.7705
7	G	cb20, dr45	2.462	5.5395
8	H	cb20, dr50	2.462	6.155
9	I	cb20, dr55	2.462	6.7705

Figure 1 shown the cross section of the sail, where the value Y_{max} and X_d was determined in accordance with camber and draft variation of the model, and those can be seen in Table 1. The thick of rigid untwisted sail is about 0.2 m. The variation of camber on the sail was set at 0%, 5%, 10% and 20% respectively have a draft of 45%, 50% and 55%. Fluid boundary condition that was used in modeling as has been illustrated in Figure 2 while the length = 15*c = 184.605 m and width = 5*c = 61.535 m.

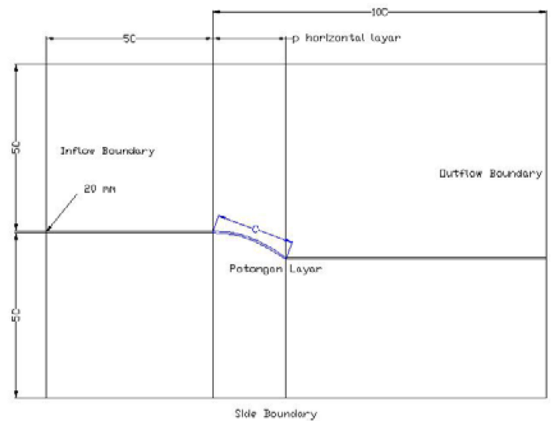


Figure 2 Modeling of sail in 2-D

The 2D sail model with geometry in Table 1 had three variations of angle of attack such as 15°, 20° and 25°; and the number variations of the sail were 30 models. The boundary conditions of sail model were determined while speed or pressure hold on the boundaries of sail geometry. At the inflow and side boundaries were obtained with the fluid velocity, V_x = 5.14 m/s and V_y = 0 m/s. At the outflow boundary was determined with the value of pressure = 0 Pa, and the velocity value on the surface of sail was 0 m /s.

Grid independence tests was carried out to obtain the number of elements efficiently in producing a good accuracy of output data with time computing a shortest as possible. Tests were conducted with a variety of elements including 5860, 13,300, 16,770 and 23,110 elements. The grid independence test was performed on the model A with a camber of 5%, draft of 45% and 15° angle of attack.

The simulation results in terms of the pressure (p) and shear force (τ) at each node on the sail surface was performed for all models of the sail. Based on the distribution of pressure and shear forces on each node, it can be calculated the amount of aerodynamic forces on the sail. Pressure distribution was illustrated through the relationship graph of pressure coefficient (Cp) and the ratio of long-distance nodes chord (x /c). Pressure coefficient itself can be obtained from equation (1).

$$C_p = \pi r^2 = \frac{P-P_\infty}{q_\infty} = \frac{P-P_\infty}{\frac{1}{2}\rho v^2} \quad (1)$$

where: C_P = pressure coefficient, P = pressure contours of the sail (on the node), P_∞ = pressure on the free flow (free stream pressure), 1.361 N/m2 (obtained at the side), ρ = density of air, 1.225 kg/m3, v = air velocity, 5.14 m/s, q_∞ = dynamic pressure-free flow 16.182 N/m2.

2.2 Experimental Setup

Tests on-sail model was carried out in laboratory wind tunnel. Wind tunnel as shown in Figure 3 is a place of testing specimens having rectangular section with a length of 150 cm, height of 66 cm, 66 cm wide and maximum speed of 21 m /s and the motor power of 5.5 KW.

Figure 3 Open circuit, Subsonic Wind tunnel (ITS)

The test section on Wind tunnel was about 2465.31 cm^2. Scale model was adapted follow the test section in order to avoid blocking of the flow uniform in wind tunnel due to the size of the model. The scale of model can be defined, 1:100. In this testing the model O and model A was selected in which model O with a camber of 0% and the model A with a camber of 5% and 45% draft. The model has a span or height, 660 mm, length of chord (c) = 123.1 mm, thickness = 20 mm, pressure taps = 26 holes and the static pressure tap mounted on the center position p (l) of the test model.

Wall pressure tap serves the measurement of fluid flow static pressure that was mounted on the surface of 2D sail model. Pivot static tube was used to measure the stagnation pressure and static pressure of the fluid flow behind the object. The manometer was used to display the pressure value measured by a wall pressure tap and pivot static tube. With Froude similitude equation, the air flow rate was defined about V_m = 1.54 m /s or the value that comes closest to that number. Fan rotation frequency was setup to produce the wind speed desired on the sail model. Results of measurement by each pressure tap on the sail model were used to calculate the pressure distribution on the surface of the model.

3. RESULTS

The result of the pressure distribution of 2D sail model was obtained by using CFD technique and wind tunnel test. Both results of pressure distribution were compared.

3.1 Pressure Distribution by CFD

The pressure distributions were resulted from the CFD simulation for all models in Table 1, and its can be classified into four typical models since each other have a different pressure distribution significantly. First, the model of flat plate O, second A, B, C with a camber of 5%, third in model D, E, F with a camber of 10%, and fourth models G, H, I with a camber of 20%. Those four typical models were analyzed for each group represented by one model. Figure 4 and 5 show the pressure distribution of model O and model A due to angle of attack 15^o, 20^o, and 25^o respectively

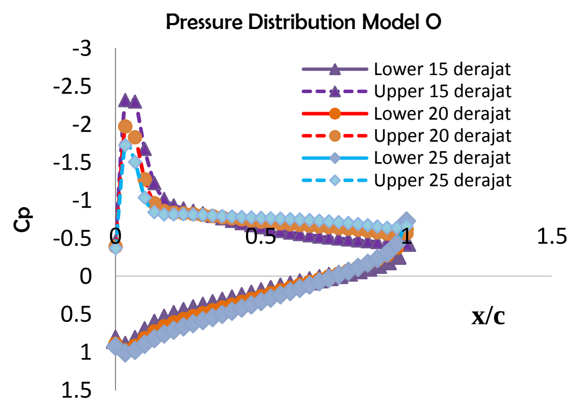


Figure 4 Pressure distribution of model O

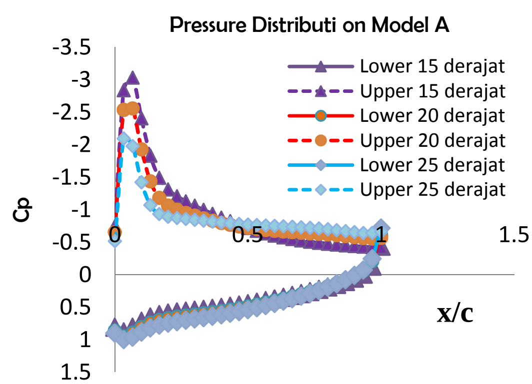
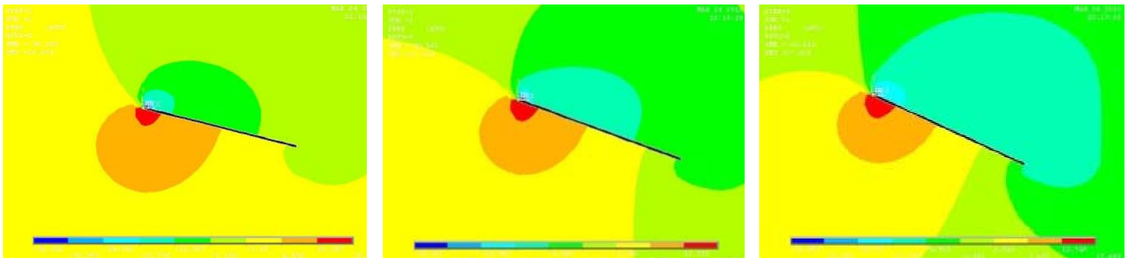


Figure 5 Pressure distribution of model A

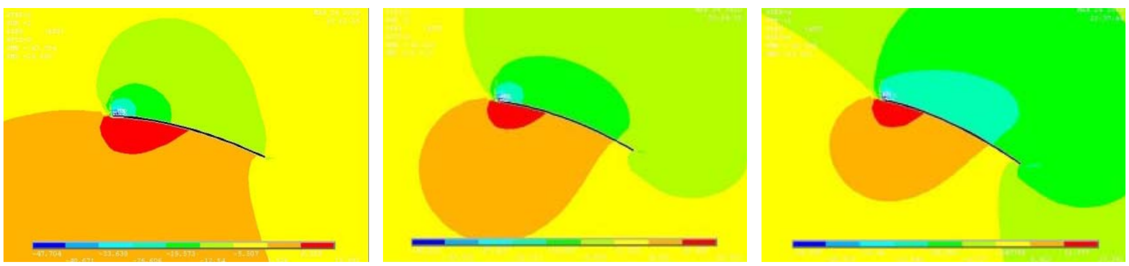
In general, the sail model has pressure distribution close to airfoil; because the principle work of sail has similarity with airfoil. A typical airfoil and the sail has a high transition point which was negative (negative pressure) on the upper part and the stagnation point (positive pressure) on the lower part. Both points are important in generating lift force of the sail, and the lift contributes greatly generate thrust on the sail.

Figure 4 through 9 show that the difference angle of attack 15^0, 20^0 and 25^0 affect the distribution pressure on the contour model for typical 1 (model O), typical 2 (model A), typical 3 (model D) and typical 4 (model G) respectively.



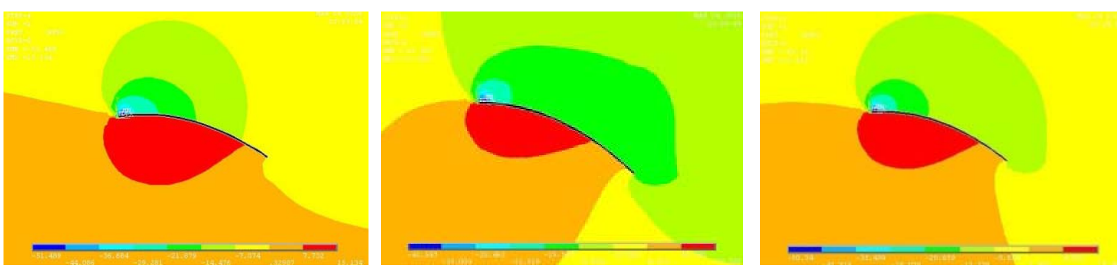
Pressure Contour, α = 15° | Pressure Contour, α = 20° | Pressure Contour, α = 25°

Figure 6 Pressure distribution of typical 1 (model O)



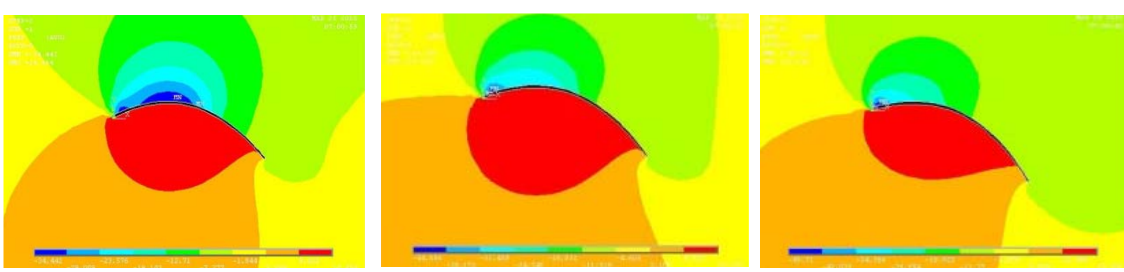
Pressure Contour, α = 15° | Pressure Contour, α = 20° | Pressure Contour, α = 25°

Figure 7 Pressure distribution of typical 2 (model A)



Pressure Contour, α = 15° | Pressure Contour, α = 20° | Pressure Contour, α = 25°

Figure 8 Pressure distribution of typical 3 (model D)



Pressure Contour, α = 15° | Pressure Contour, α = 20° | Pressure Contour, α = 25°

Figure 9 Pressure distribution of typical 4 (model G)

3.2 Pressure Distribution by Experiment

Figure 10 and 11 show the pressure distribution of 2D sail model from wind tunnel tests for model O with camber 0% and model A with camber 5%, draft 45% respectively.

In the figures, while the angle of attack was getting bigger, the transition point getting smaller and opposite the stagnation point become bigger. On the upper and lower parts of both models, the bubble separation does not occur, and only on model A the trailing edge separation evident with decreasing pressure from the reattachment.

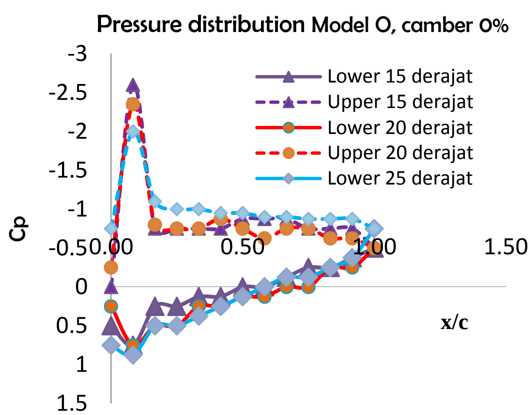


Figure 10 Pressure of model O, (camber 0%)

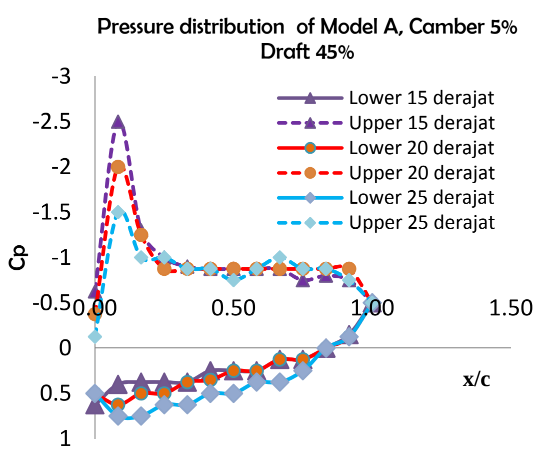


Figure 11 Pressure of model A, (camber 5%)

Relatively small difference both methods of pressure distribution is shown in Figure 12 through 17. Comparison of CFD simulation and testing models in wind tunnel were expressed with error. Percentage error can be defined by the following equation:

$$Error = \left|\frac{(CFD\ result - WT\ result)}{WT\ result}\right| \times 100\%$$

With the C_{FT} in each method, the difference between the both results were obtained in Table 2. The error was defined about 2.28% to 7.87%. This shows that the both results has a small difference under 10%.

Tabel 2 Error of the force resultant coefficient

	C_{FT} WT		C_{FT} CFD		Error (%)	
α	Mod. O	Mod. A	Mod. O	Mod. A	Mod. O	Mod. A
15°	1.01	1.25	0.99	1.35	2.39	7.87
20°	1.07	1.26	1.05	1.33	2.28	5.42
25°	1.20	1.31	1.12	1.34	6.80	2.24

However, the percentage of error was expected to reach smaller values when the pressure tap on the 2D model wind tunnel more or adjusted by the number of nodes on the surface of the CFD model. In addition, errors can also be contributed by the precision of measurement on the manometer.

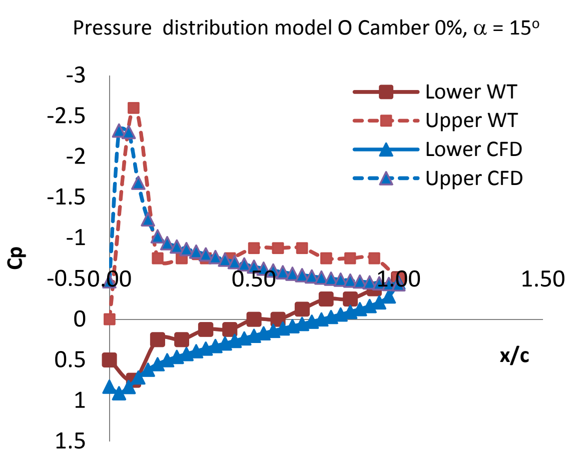


Figure 12 Pressure of model O, (camber 0%, α = 15°)

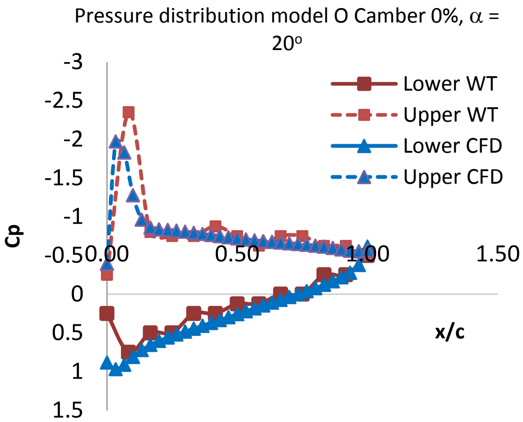


Figure 13 Pressure of model O, (camber 0%, $\alpha = 20^0$)

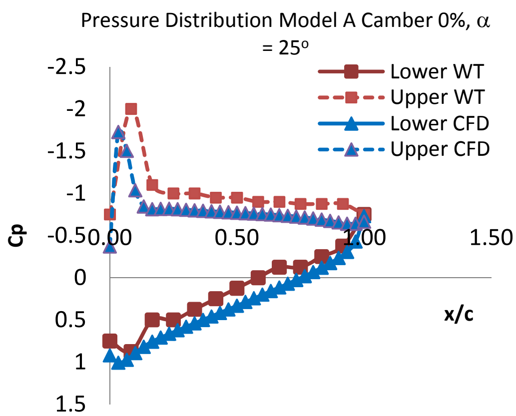


Figure 14 Pressure of model O, (camber 0%, $\alpha = 25^0$)

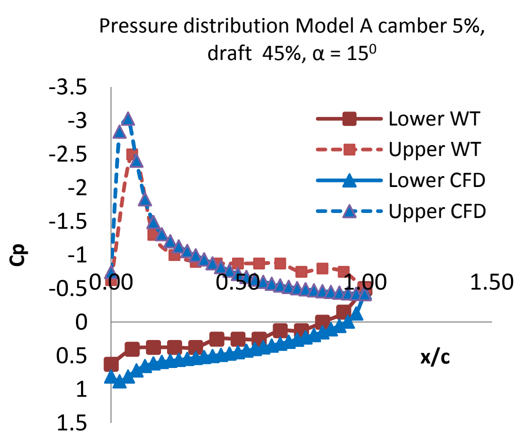


Figure 15 Pressure of model A, (camber 5%, $\alpha = 15^0$)

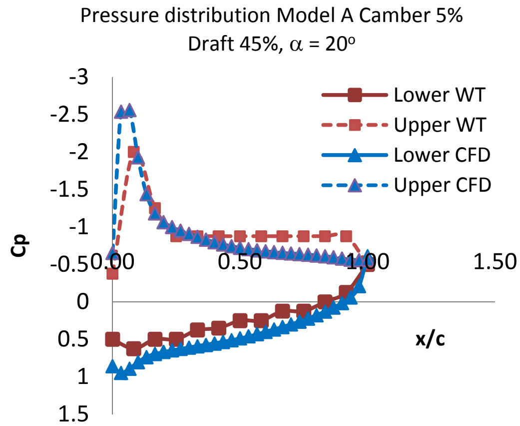


Figure 16 Pressure of model A, (camber 5%, $\alpha = 20^0$)

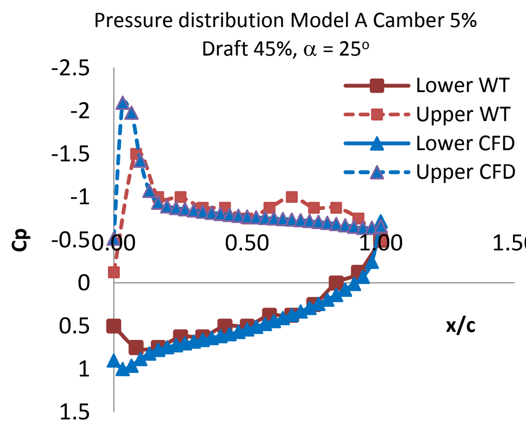


Figure 17 Pressure of model A, (camber 5%, $\alpha = 25^0$)

3.3 Trust of Sail

The thrust of sail can be expressed in terms of trust coefficient, where the sail which has the largest thrust coefficient means that the screen has the largest thrust as well. The magnitude of thrust coefficient can be expressed in terms of driving force, C_R and heeling force, C_H. Figure 18 show the three variations of wind angle direction due to ship (the apparent wind angle, β) such as 30°, 60° and 90° with the value of C_R and C_H.

Figure 9 shows that the model G (camber 20% Draft 45%) have the most optimum thrust on all three variations of the apparent wind angle. The polar diagram shows the value of C_R and C_H coincides on models which have the same camber design. And the camber design has a big influence on the thrust generated than the draft design.

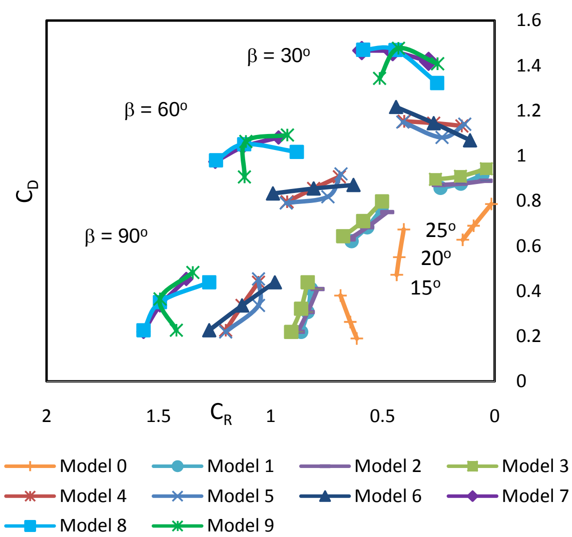


Figure 18 Polar diagram of C_R and C_H

4. CONCLUSIONS

The analysis of 10 (ten) sail design were done by using approach of CFD simulation and testing models in wind tunnel. The variation percentage of the draft on a sail design that were 45%, 50% and 55% cause a slight decrease in the lift force, but increase in the drag force. While the percentage of camber on the model increase with 5%, 10% and 20%, the lift force also increase significantly. The simulations show the design of sail models which generated maximal efficiency and the most optimal thrust force, were the sail models with camber value 20% and draft 45%. The comparison results of CFD and wind tunnel testing showed the two-dimensional analysis has the largest percentage of 7.87% error. This error occurs caused by several factors, the biggest factor error impact are the process of calculating the force on the wind tunnel tests are less accurate, and scales that are used less worthy style.

REFERENCES

[1] Bao, F. and Dallmann, U.Ch, "Some phisycal aspects of separation bubble on a rounded backward facing step", Science Direct, 2003.

[2] Collie, S.J., "Application of computational Fluid Dynamics to Two Dimensional Downwind Sail Flows", PhD Thesis, Department of Mechanical Eng. and Eng. Science, University of Auckland New Zealand (2006).

[3] Kartika, C.A., "Analisis Teknis Perencanaan Layar sebagai Penggerak Utama pada KLM Maruta Jaya 900 DWT", BSc. final project, ,Jurusan Teknik Sistem Perkapalan, ITS Surabaya (1996).

[4] Marchaj C. A., "Sailing Theory and Practice", New York, United State. (1982)

[5] Miyata, H., and Lee, Y.W., "Application of CFD Simulation to Design of Sails", Jour. of Marine Science and Technology, 4:163-172. (1999)

[6] Nasirudin,A., "Desain Layar Simulasi CFD", BSc final project, Teknik Perkapalan, ITS Surabaya. (2002)

[7] Versteeg, H.K., and Malalasekera, W., "An Introduction to Computational Fluid Dynamics", Longman Scientific& Technical, England (1995)

[8] Wiriadidjaja, S., "Pengembangan Kapal Layar Modern untuk Menunjang Perdagangan antar Pulau di Indonesia, ITS Surabaya. (1997).