Science Arts & Métiers (SAM)
is an open access repository that collects the work of Arts et Métiers Institute of Technology researchers and makes it freely available over the web where possible.

This is an author-deposited version published in: https://sam.ensam.eu
Handle ID: .http://hdl.handle.net/10985/7624

To cite this version :

Annie-Claude BAYEUL-LAINE, Abir ISSA, Gérard BOIS, Sophie SIMONET - The importance of the knowledge of flow stream in water sump pump - In: Congrès Français de Mécanique (21; 2013 ; Bordeaux), France, 2013-08-26 - Congrès français de mécanique (21; 2013; Bordeaux) - 2013

Any correspondence concerning this service should be sent to the repository Administrator : scienceouverte@ensam.eu

The importance of the knowledge of flow stream in water sump pump

A.C. BAYEUL-LAINÉ[a], S. SIMONET[a] , A. ISSA[b] , G. BOIS[a]

a. Arts et Métiers PARISTECH, LML, UMR CNRS 8107, Bd Paul Langevin, 59655 VILLENEUVE D'ASCQ
b. *Faculté de Génie Civil, Université de Damas, Baramkeh-Mazzé, Syrie*

Résumé:

De nombreuses méthodes expérimentales et numériques ont été développées en vue d'obtenir une meilleure description de l'écoulement interne dans les pompes. Ces calculs sont, en général, réalisés dans l'hypothèse de conditions aux limites uniformes et stationnaires, en particulier à l'aspiration. Cependant, peu d'études s'intéressent à des conditions aux limites non uniformes, particulièrement dans le cas des bassins d'aspiration des pompes. L'entrainement de tourbillons d'air dans les conduites d'aspiration est un problème important car ils réduisent les performances de la pompe. Le papier étudie l'écoulement du fluide dans le bassin d'aspiration. Des expériences ont été réalisées en vue de sélectionner la meilleure position du tuyau d'aspiration pour éviter l'apparition des tourbillons. Les études numériques tentent de reproduire le comportement du fluide dans le cas des expériences et de confirmer l'influence des paramètres géométriques sur la structure de l'écoulement dans le bassin. Ces études numériques ont mis en évidence la nécessité d'une modélisation instationnaire diphasique pour une meilleure compréhension des écoulements.

Abstract:

Numerous different experimental and numerical methods have been developed and are already improved in order to get rather good flow descriptions inside pump components. These calculations are generally realized assuming uniform and steady flow conditions, in particular for the inlet conditions. However, few studies are devoted to non-uniform upstream conditions, especially in case of water pump sump before entering the pump inlet tube. Free air-core vortex occurring at the water-intake pipe is an important problem encountered in hydraulic engineering because they reduce pump performances and may have large effects on the operating conditions. The paper deals with flow pattern inside sump pump. Experiments were conducted in order to select best positions of the suction pipe of water-intake specific configurations. The numerical studies try to reproduce the flow pattern of these experiments and confirm the influence of geometrical parameters on the flow structure in such a sump. These numerical studies points out the need of unsteady calculation in order to get a better understanding of flow behaviour including two-phases flow models.

Keywords: **pump, sump, simulation, RANS, VOF**

1 Introduction

This work is an extension of a previous study started in 2006 (Issa [1] in LML). Several sump configuration cases have been numerically investigated using one specific commercial code and are based on the initial geometry proposed by Constantineascu and Patel [2]. The results, obtained with a structured mesh, were strongly dependent on main geometrical sump configuration such as the suction pipe position, the submergence of the suction pipe and the turbulence model. Part of the results shows a good agreement with experimental investigations already published by the Iowa Institute of Hydraulic Research [3] in order to reduce non uniformities of specific flow and geometrical conditions. More basic studies have been also conducted to establish empirical criteria for vortex formation and avoidance [2-8]. More recently, CFD Benchmarks have been performed by Matsui et al [5] in order to compare different software results with experimental data. Numerical simulations were realized by Isabasoiu et al [6] with the commercial code FLUENT version 6.0. These simulations didn't take in account air entrainment. The goal of the numerical approach used in this paper is to find the influence of the upstream flow level at the intake channel of the suction sump in the swirling flow development. Few papers were found discussing about air entrainment. The main difficulty was to take account the effect of free surface and to model it.

Lucino et al [7] verified the ability of commercial CFD (computational fluid dynamics) code to predict the formation of vortices in a pump sump with FLOW 3D using LES (Large Eddy Simulation). The free surface was tracked by means of a split lagrangian method for the advection of the VOF (volume of fluid). The fluid was considered monophasic and incompressible. The numerical results demonstrated the capability of the model to identify the observed vortices in the physical model. The calculated vortex highest strength values were consistent with the air-entrained vortices. The floor vortex reached the highest vorticity magnitude and suggested the possibility of a cavitating core at the operating condition of lower submergence. Shulka et al [8] used the commercial code CFX to carry out two-phase flow simulation to capture air entrainment. They used a steady state solution. Turbulence was modelled by a k-ε model. The paper dealt with numerical and experimental results. The CFD model predicted the flow in sufficient details to identify the locations, size and the strength of the vortices. The air entrainments and its location are well captured. The results are in accordance with the experimental investigation.

2 Experimental geometric models

The experimentations were realized by Issa Abir in the Hydraulic Laboratory at "Université de Génie Civil" in Damas [1]. The main aim of this experimental study is to detect the best position of the intake pipe which didn't lead to troubles at the inlet of the pump. It is essential to ensure that a pump intake supply water with the most uniform velocity at the entry of the impeller. The fluid flow in pump intakes is rather complex, often with swirl-free flows.

The experimental apparatus consisted of a closed loop containing a sump (FIG. 1), a water-sump pump realized in transparent acrylic walls, a centrifugal pump, a metallic pipe at the outlet of the pump, a flow rate-meter, a valve to control the mass flow rate. The following measurements were realized: the tangential velocity in the pipe, the static pressure at the wall of the inlet pipe for different angular positions, the pressure on different parts of the loop, the mass flow rate. Formations of free surface vortices and subsurface vortices were visually observed.

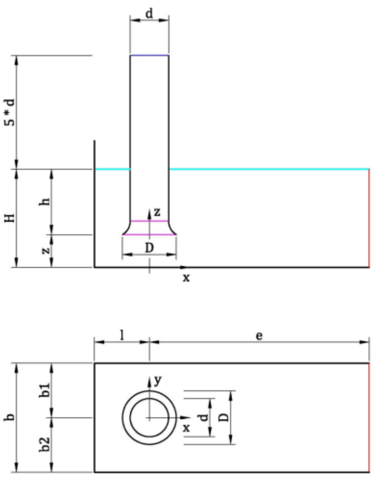

FIG. 1. Shape of sump pump

FIG. 2. Two symmetric vortices

Numerous experimental configurations (192 cases) were tested by combining parameters (b/D=1.5, 2, 2.5, 3; l/D=0.5, 0.75, 1, 1.25; z/D=0.3, 0.5, 0.75,1). These parameters are given in function of baffle intake interior diameter D (D=180 mm). These parameters are the depth of the sump b, the distance l between the pipe and the back side of the sump, the clearance distance from floor z and, the submergence depth for the pipe h (FiG. 1). All measures were realized for a constant flow rate of Q_m equal to 18 kg/s and for the following non dimension numbers: $Re_d = 1{,}8\ 10^5$; $5.56\ 10^4 < Re_h < 8.33\ 10^4$; $Fr = 1.45$; $We = 6160$.

The experimental investigation revealed that there is formation of air entrainment when b and h are small. All air entrainment vortices locate along the height between the pipe and the back wall. One or two air entrainment vortices have been observed. For example, two symmetric swirls can be observed in FIG. 2 for l=1,25D. If b is small, sometimes, only one air entrainment vortex appears; this swirl is often located in the middle of the sump. It has been found that the most important parameters that influence the occurrence of air entrainment are the depth of the sump b, the distance between the pipe and the back wall l, the submergence depth of the pipe h and the clearance distance from floor z.

Overall experimental analysis gave the following results. Larger vamues of l increases the potential air entrainment vortices. The size and the magnitude of these swirls increase too. When b increases, the air entrainment vortices decrease in magnitude until they become vortices with no air entrainment or decrease to finally disappear. The beginning and the magnitude of these swirls decrease with the increase of the values of z. When z is equal to the diameter of the bell mouth D, the air entrainment swirl has a low magnitude if b is high (b/D>2.5) and l small. This is the case for low velocity at the free surface. Increasing the value of submergence decreases the possibility of air entrainment vortices

3 Calculations

The simulations were performed with the commercial code Star CCM+ V6.06. The main difficulty of such simulations is to well represent the behaviour of the free surface. In previous studies [1], one phase models were tested with a symmetry plane boundary at the free surface. In the present study the calculations were done with two phases model (water and air phases) (FIG. 3) in case of steady state and in case of unsteady state. In order to well represent air entrainment, according to experimental results, the next parameters have been chosen as a reference numerical case: b/D=1.5; l/D=1.25; h/D=1.2; z/D=0.3. Experimentations pointed out air entrainment in this specific case. Tests with a lower submergence depth of the pipe h equal to D were also realized.

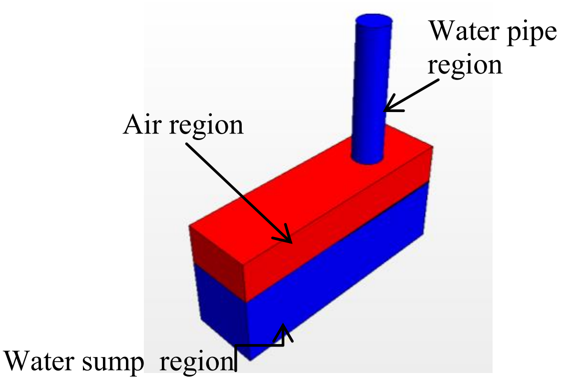

FIG. 3 Model : shape and regions

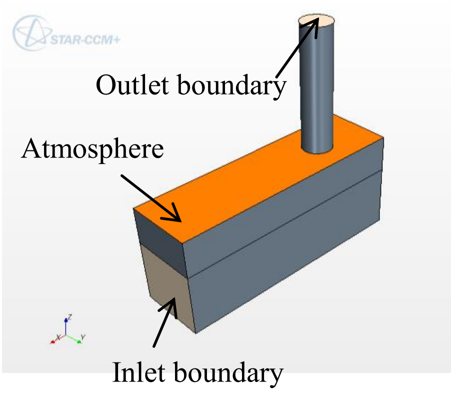

FIG. 4 boundaries conditions

The reference mass flow rate is the one used in the experimental apparatus. In order to well visualize the air vortices, numerical tests with higher mass flow rate were also calculated. The calculation domain was divided into three regions: the air region, the sump region and the pipe region. So for the simulation of the sump pump three linked meshes were used as it can be seen in FIG. 3. The boundary condition at the inlet consisted of a mass flow rate (18 and 24 kg/s). The boundary condition at the outlet was the same constant mass flow rate going out of the domain. This was located at the top face of the pipe. The boundary condition at the horizontal boundary of the air region is an absolute pressure inlet equal to atmospheric pressure (FIG. 4). The fluid was considered a two phase flow (air and water), incompressible for the water phase and as an ideal gas for the air phase, at a constant temperature of 293 K. Buoyant option, with density difference fluid buoyant model, is assigned to all domains. Buoyant reference density of air is assigned to all domains. The gravity acceleration was set to 9.81 m/s². Two turbulence models were tested: the realizable two-layer k-ε model which combines the Realizable k-ε model [9] with the two-layer approach [10] and the SST k-ω model [11]. Volume of Fluid (VOF) is used to model the interface between the two phases. It is a simple multiphase model that is well suited to simulate flows of several fluids on numerical grids capable to resolve the interface between the mixture phases. The two phases are assumed to share the same velocity, pressure and temperature fields. As a consequence, the same set of basic governing equations describing momentum, mass and energy transport in a single-phase flow is solved for an equivalent fluid whose physical properties are calculated as a function of the physical properties of its constituent phases and their volume fractions. The initial condition for the volume fraction of phase is given in FIG. 3: air for air region (in red colour) and water for water pipe region and water sump region (in blue colour).

A steady state solution can depict the formation of an air core but it is well known that the phenomena are always transient. The steady and unsteady two phases models were calculated in different cases. The steady models are used as initial conditions for the unsteady models. A multi-domain model allows using different target sizes. Mesh was refined in pipe zone and in the interface between air and water at free surface. A polyhedral mesh with prism layers is used for all calculations (7 prism layers in the sump and 10 in the pipe). A cell growth rate of 1.02 is used. All test cases are resumed in table 1 (i=1 for steady state and i=2 for unsteady state). The reference case is defined for the reference geometry and for the reference mass flow rate (cases Ai for the k-ε turbulence model and case Bi for the k-ω turbulence model). Two influenced parameters were tested: the influence of mass flow rate (cases Ci and Di) and the influence of submergence (cases Ei and Fi)

Table 1. Calculation cases

cases	Turbulence model	h/D	$Q_m/18$	Number of cells (M cells)	Re_h	Re_d	Fr	We
Ai	k-ε	1,2	1	1.3	83 333	191 000	1,45	4108
Bi	k-ω	1,2	1	1.3	83 333	191 000	1,45	4108
Ci	k-ε	1,2	1,5	1.3	125 000	286 500	2,18	6162
Di	k-ω	1,2	1,5	1.3	125 000	286 500	2,18	6162
Ei	k-ε	1	1	0.8	100 000	191 000	1,45	4108
Fi	k-ω	1	1	0.8	100 000	191 000	1,45	4108

4 Results

The main aspect of this research is to examine the possibilities of this commercial code to visualize the air entrainment or to detect the ability of identifying the free surface vortices. Visualizing air entrainment is easy: volume fraction (VF) of air or water can be drawn in different cut planes as can be seen in FIG. 5. Free surface vortices can be detected by the examination of pressure field, of the Z vorticity in a plane just below the interface, of the contours of pressure surface near free surface, of the streamlines or of the air cores from free surface (FIG 6-8). The steady calculations (A1, B1, C1, D1, E1 and F1) are used as initial conditions for unsteady calculations (A2, B2, C2, D2, E2 and F2). The steady calculations hardly converged for cases A1, B1, C1 and D1. On the contrary the convergence for the cases E1 and F1 is of good quality. For unsteady calculations, considering the time taken by a particle to go from the inlet to the centre of the bell mouth in a straight way, as a characteristic time T (2.5 s for $Q_{m=}$18 kg/s and 1.67 s for Q_m=24kg/s), a total time of 10 s has been chosen for all cases with a time step of 5. 10^{-3} s. The chosen time step has been chosen taking a caracteristic time based on mean inlet flow velocity and intake tube diameter. The unsteady calculations lead to best convergences than steady calculations (residuals less than 1e-5) for the two turbulence models. The two turbulence models give unsteady and unstable results, except for the case E2, as it can be seen in FIG. 9.

For the reference geometry and the reference mass flow rate (cases A2 and B2), the calculations with the k-ε turbulence model (A2) don't really conduct to air entrainment. For the k-ω omega turbulence model, the examination of volume fraction of air at 5 mm under the level of free surface points out the birth of swirls but air entrainment was not pointed up. The examination of Z-vorticity in the same plane and the examination of streamlines confirm this tendency. But the streamlines don't show air cores occurrences. For the reference geometry, but with higher mass flow rate (1.5*initial mass flow rate), the k-ε model doesn't really show air entrainment by the examination of volume fraction of air. However, streamlines coming from the centre of free surface vortices point out air cores. The air cores can be characterized by the strong density of loops as it can be seen in FIG. 6. Unfortunately, the examination of contours of pressure surface doesn't prove it. The k-ω turbulence model points out air entrainment as can be observed in FIG. 5. The unsteady state of the vortex can also be seen in this figure. The cut plane, in which air entrainment is pointed up, allows detecting the location of a part of the vortex. All indicators of all entrainment are in good agreement. The magnitude of air core is greater in case of k-ω turbulence model (75s^{-1} for about 30 s^{-1} in case of k-ε turbulence model). So a larger mass flow rate increases the probability of air entrainment as it has been demonstrated by the two turbulence models. Furthermore the behaviour of water in sump pump is highly unsteady, unstable and intermittent. This conclusion wasn't drawn by our experimental results but was already published by other authors [13]. The time chosen to observe phenomenon seems to be too small: these first results showed that the behaviour of water is highly unsteady and no real periodicity appears.

For the last cases (Ei, Fi) the observation time was increased to 15 s. For lower submergence and the reference mass flow rate, the two models allow the detection of air entrainment. On the contrary of the results of previous cases, the k-ε turbulence model leads to symmetric results which are in good agreement with the experimental results (FIG. 9). The k-ω turbulence model gives asymmetric and unstable results as it can be seen in FIG. 10. During the first five seconds, the two turbulence models seem to give some equivalent results. After then the k-ω turbulence model leads to intermittent results. The magnitude of one of the two air cores increases as the magnitude of the other increases but this phenomenon doesn't seem to be periodic. The potential air entrainment has been detected by the visualization of air cores, of contours of pressure plane and streamlines from free surfaces The VF of air doesn't allow this visualization probably due to the unstable position of air cores.

t=3.85 s **t=4.35 s** **t=4.85 s** **t=5.35 s**

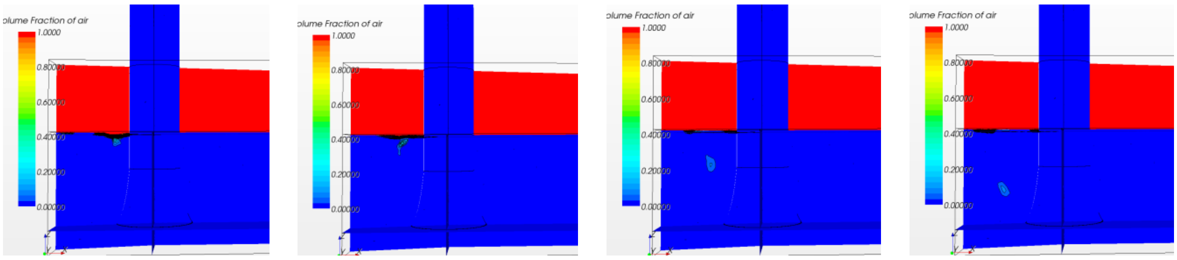

FIG. 5 two phases model : air entrainment observed for the D2 model (k-ω turbulence model, Q_m=24 kg/s)

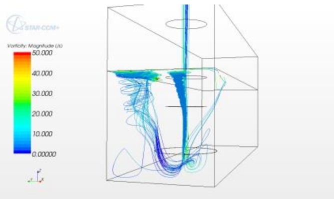

FIG. 6 Air cores, case B2, reference geometry, k-ε turbulence model, Q_m=24 kg/s, t=10 s

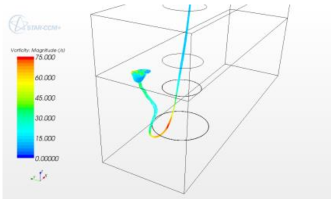

FIG. 7 Air cores, case D2, reference geometry, k-ω turbulence model, Q_m=24 kg/s, t=5.35 s

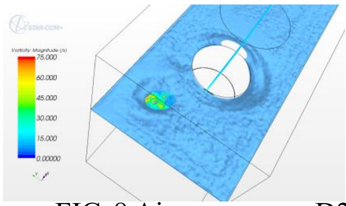

FIG. 8 Air cores, case D2, reference geometry, k-ω turbulence model, Q_m=24 kg/s, t = 5.35 s

t=15s (case E2)

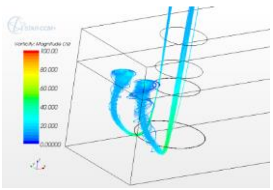

FIG. 9 Air core k-ε turbulence model

t =5 s (2.0*T) **t = 10 s (6.0*T)** **t =15 s (6.0*T)**

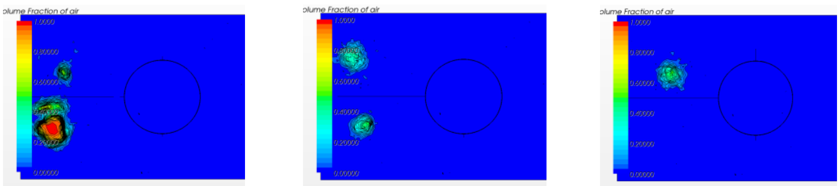

FIG.10 two phase model: VF of air at 5 mm under the level of free surface for the F2 model (k-ω turbulence model)

5 Conclusion

The present study shows the ability for a commercial code to predict the two phase flow behaviour inside a pump sump with good agreement with experimental investigations. Unless one specific result, this study allows establishing some rules that must be applied regarding the ability of Star CCM+ code to predict air entrainment in sump pump.

All cases, unless case E2 (k-ε turbulence model), prove that the flow basic characteristics in the sump pump are highly unsteady, unstable and intermittent whatever the turbulence model is. Unsteady two phases flow model allows visualizing the behaviour of water in sump pump taking into account the real influence of free surface. The air entrainment visualization seems to be easier by using the volume of fluid of air. But this parameter can only be caught in defined cut planes. The great difficulty is the unsteady position of the air core making the choice of cut planes not so easy. Unsteady numerical results also show that the strength of the two vortices is not equal and changes with time. One vortex grows up and at the same time the second one decreases; this phenomenon seems to have a specific frequency based on the mean stream velocity and on the diameter of the tube. The potential air entrainment begins for the submergence equal to 1.2D (reference case) for the reference mass flow rate for the k-ω turbulence model.

Nomenclature

	b_1		[m]	Pipe left wall distance
	b_2		[m]	Pipe right wall distance
	d		[m]	Pipe intake interior diameter
	D		[m]	Baffle intake interior diameter
	e		[m]	Pipe back wall distance
	Fr	$V/(gd)^{1/2}$	[-]	Froude number for the pipe submergence
	g		[m/s^2]	Acceleration due to gravity
	H		[m]	Water level in the sump-pump
	h		[m]	Submergence depth for the pipe
	h_b		[m]	Baffle height
	l		[m]	Pipe back side distance
	Q_m		[kg/s]	Mass flow rate
	Re_d	VD/ν	[-]	Reynolds number in the pipe
	Re_h	$Q_m/\rho\nu h$	[-]	Reynolds number in the sump
	t		[s]	time
	T		[s]	Characteristic time
	U		[m/s]	Mean velocity in the sump
	V		[m]	Mean velocity in the intake pipe
	W		[m]	Pump-sump width
	We	$V^2\rho D/\sigma$	[-]	Weber number
	z		[m]	Clearance distance from floor
	ν		[m^2/s]	kinematic viscosity
	ρ		[kg/m^3]	Water density
	σ		[N/m]	Coefficient of surface tension
	ω		[s^{-1}]	Specific dissipation rate
Index	$_1$			Subscripts for level free surface inside sump
	$_2$			Subscripts for level free surface inside pipe

References

[1] Issa A 2009 Etudes hydrauliques de l'influence des géométries des basins sur l'alimentation des pompes Ph. D Thesis Arts et Metiers PARISTECH Ecole doctorale.

[2] Constantineascu G and Patel V C 1998 Numerical Model for Simulation of Pump Intake Flow and Vortices *J. of hydraulic Engineering. Div ASCE* **vol 124** 123-134

[3] Nakato T 1988 Hydraulic-Laboratory Model Studies of the Circulation-Water Pump-Intake Structure, Florida Power Corporation, Crystal River. Units 4 and 5 *IIHR Rep. No.320*, Iowa Inst.of Hydr. The Univ of Iowa. Iowa City.

[4] Anwar HO and Amphlett M B 1980 Vortices at Vertically in Vertex Intake *J. Hydr. Res.*, **18(2)** 123-134

[5] Matsui J, Kamemoto K, and Okamura T 2006 CFD Benchmrak and a Model Experiment on the Flow in Pump Sump *Proc. of 23th IAHR Symp.* October 17-21Yokohama 110-16

[6] Isbasoiu EC, Muntean T, Safta C A, Stanescu P 2005 Swirling flows in the suction sumps of vertical pumps, theoretical approach *Workshop on vortex dominated flows* June Timisoara Romania pp 17-22

[7] Lucino C, Gonzalo Dur SL 2010 Vortex detection in pump sumps by means of CFD *, XXIV latin American congress on hydraulics* November Punta del este Uruguay

[8] Shulka Kshirsagar JT 2008 Numerical prediction of air entrainment in pump intakes *Proc. of the24th int. pump users Symp.* pp 29-33

[9] Shih, Liou WW, Shabbir A, Yang Z and Zhu J 1995 A New k-ε Eddy Viscosity Model for High Reynolds Number Turbulent Flows: Model, Development and validation *Computers and Fluids* **Vol 24 No 3** pp227-238

[10] Rodi W 1991 Experience with Two-Layer Models Combining the k-ε Model with a One-Equation Model Near the Wall *29th Aerospace Sciences Meeting* January 7-10 Reno NV AIAA 91-0216

[11] Menter FR 1994 Two-equation eddy-viscosity turbulence modeling for engineering applications *AIAA Journal* **32(8)** pp. 1598-1605