

Implementation and Initial Validation of the Singhal Cavitation Model in *OpenFOAM*

N.L. Mulcahy¹, P.D.M. Brady², D. Ranmuthugala³ and D. McGuckin²

¹School of Mechanical and Manufacturing Engineering
 UNSW Sydney, New South Wales 2052, Australia

²Pacific Engineering Systems International
 Glebe, New South Wales 2037, Australia

³Defence Science and Technology Group
 Melbourne, Victoria 3207, Australia

Abstract

Cavitation fundamentally affects the seakeeping performance of marine platforms. Both numerical methods and physical experiments are required to efficiently investigate cavitation about complex geometries. Herein, the Kunz, Merkle and Schnerr-Sauer models that are native to the OpenFOAM software toolkit were investigated. Because of the many parameters that have to be tuned, it was concluded that these models were unreliable for use against geometries where validation data for comparison are not available.

Instead, the Singhal Full Cavitation Model was implemented in OpenFOAM as this model only has two parameters to tune, although in practice, the parameters actually remain constant. The results of flow over a nominally two-dimensional NACA 66 (MOD) foil were validated against those from the CFD-ACE+ (ACE+) software tool and physical experiments. The low-Reynolds number turbulence models performed well at all cavitation numbers. In contrast, while the high Reynolds number turbulence models performed well at low cavitation numbers, they showed significant variation from the expected inception point.

Introduction

Cavitation is a flow phenomenon of liquids such that as the pressure of a flowing liquid is reduced to its vapour pressure, or even below, portions of the liquid “boil” and vapour pockets are formed as shown in Fig.1. These vapour pockets may coalesce into sheets and remain stable or collapse back to the liquid phase as they are transported into regions of higher pressure. Irrespective of sheet or bubble forms, when the vapour collapses to liquid in the presence of a solid surface, significant forces can be generated against the adjacent boundary. Further, these forces can be of such magnitude and frequency as to cause significant structural and/or vibrational damage. In addition to the potential physical damage, cavitating flow regions create large acoustic signatures, which are undesirable for maritime platforms.

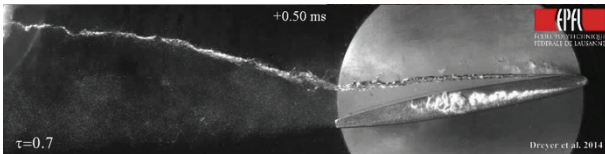


Figure 1. Tip vortex and sheet cavitation of a foil (from [4]).

Cavitation may be investigated using either numerical or physical methods. In fact, both are mutually beneficial, with neither perfect by itself. This work concerns itself with numerical techniques, using the open source computational fluid dynamics (CFD) software toolkit, OpenFOAM and, in particular, the v1706 release [14].

OpenFOAM is a C++ toolkit designed to solve non-linear continuum mechanics problems. It comprises many numerical methods and variety of “applications” that allow the solution of specific flow regimes such as heat transfer, supersonic flow or fluid structure interactions (FSI). One such application, *interPhaseChangeFoam*, designed for cavitating flows, is packaged natively with three cavitation models, namely the Kunz, Merkle, and Schnerr-Sauer models

The Kunz model, as implemented within OpenFOAM, is based on two mass transfer equations [9], vaporisation or destruction (liquid to vapour)

$$\dot{m}^- = \frac{C_{dest} \rho_v \gamma \min[0, p - p_{sat}]}{0.5 \rho_l U_\infty^2 t_\infty} \quad (1)$$

and condensation or production (vapour to liquid) [5]

$$\dot{m}^+ = \frac{C_{prod} \gamma^2 (1 - \gamma) \rho_v \max[p - p_{sat}, 0]}{t_\infty \max[p - p_{sat}, 0.01 p_{sat}]} \quad (2)$$

in which γ is the liquid volume fraction, ρ_v is vapour density, ρ_l the liquid density, p the pressure, p_{sat} the saturation pressure, t_∞ the free stream time scale, U_∞ the free stream velocity, and C_{dest} and C_{prod} are coefficients.

The implemented Merkle model [12] has similar, yet different, vaporisation and condensation terms, namely:

$$\dot{m}^- = \frac{C_{dest} \min[p - p_{sat}, 0] \gamma \rho_l}{0.5 U_\infty^2 t_\infty \rho_v} \quad (3)$$

$$\dot{m}^+ = \frac{C_{prod} \max[p - p_{sat}, 0] (1 - \gamma)}{0.5 U_\infty^2 t_\infty} \quad (4)$$

While the implementation of the Schnerr & Sauer model [15] defines the vaporisation and condensation equations as

$$\dot{m}^- = -C_{dest} (1 + \gamma_{nuc} - \gamma) \gamma P_{coeff} (1 - H(p_{sat} - p)) \quad (5)$$

and

$$\dot{m}^+ = C_{prod} (1 - \gamma) \gamma P_{coeff} H(p - p_{sat}) \quad (6)$$

in which P_{coeff} is a term expressing the bubble dynamics and is the same for both condensation and vaporisation, and γ_{nuc} is the nucleation site volume fraction. This model is based on a modified Rayleigh-Plesset equation for bubbly flows. The Unit Step (or right continuous Heaviside) function, $H(x)$, returns unity when x is non-negative and zero otherwise. Two

additional parameters are required for the S&S model, specifically the bubble density, n , and the nucleation site diameter, d_{nuc} . Critically, each of these three models each dependent on four tuneable constants (as in Table 1).

Model	Parameter
Kunz	$U_{cs}, t_{cs}, C_{prod}, C_{dest}$
Merkle	$U_{cs}, t_{cs}, C_{prod}, C_{dest}$
Schnerr-Sauer	$n, d_{nuc}, C_{prod}, C_{dest}$

Table 1. OpenFOAM's native cavitation models' tuneable constants

A literature review [5, 8-10, 12, 15], which was confirmed through internal testing (not published here), shows that the cavitation simulations are *very* sensitive to the values of the constants, and the values of the constants can vary by many orders of magnitude.

Given a standard CFD methodology where a validation study is performed on a known configuration before moving to an unknown geometry, it is concluded that it would be impossible to make the leap into the unknown with such a wide variation in the parameters. Therefore, the focus of the research shifted to the Singhal cavitation model [19] as it is based on two parameters, which are only rarely modified [6].

Conceptually, the **Singhal full cavitation model** is closest to the Schnerr-Sauer model in that it is derived from bubble dynamics. However, there is an explicit recognition of the turbulent fluctuations on the pressure field, and the resultant mass transfer rates that leads to mass transfer equations of

$$R^- = C_{dest} \frac{V_{ch}}{\sigma} \rho_v \rho_l \sqrt{\frac{2}{3} \frac{p_v - p}{\rho_l}} (1 - f_v) \quad (7)$$

and

$$R^+ = C_{prod} \frac{V_{ch}}{\sigma} \rho_v \rho_l \sqrt{\frac{2}{3} \frac{p - p_v}{\rho_l}} f_v \quad (8)$$

in which f_v is the vapour fraction, V_{ch} is a characteristic slip velocity between the liquid and the vapour with the remaining p and ρ terms representative of the pressures and velocities for the respective liquid (l) and vapour (v) phases.

The **NACA66(MOD) two-dimensional (2D) foil** of Brockett [2] has been used extensively for the validation of nominally two-dimensional sheet cavitation [1, 7, 11, 13, 16-18]. The reference experimental investigation [3] is undertaken in the GALCIT High Speed Water Tunnel at the California Institute of Technology. The high-speed working section is 1.27 m long with a cross section of 152 mm by 762 mm and is subjected to operational flow speeds up to 30.5 m/s and pressures from 689 kPa down to water vapour pressure [20].

Computational Models

The reference computational model created for CFD-ACE+ (ACE+) [6] was based on a C-topology with clustering at the trailing edge as shown in Fig.2. All ACE+ simulations were run with the default settings, specifically steady state mode and central differencing with a $k-\omega$ SST turbulence model, with wall functions if the turbulence model Reynolds number was high, and without wall functions if that number was low.

Grid convergence (GC) studies were primarily focussed on the lift and drag. Foil lift was stable for all configurations and was considered converged. In contrast, the drag force results

converged but not towards the reference value. This result is to be expected because high-Re models simply do not capture the boundary layer, yet the boundary layer has been shown to be crucial to foil drag simulation. Similarly, for the cavitation GC, the measured sizes of the cavitating regions were converged. However, for the high-Re models, inception was predicted 22 % lower than the experiment. While final analysis with OpenFOAM of the low Re data is continuing, the ACE+ results were both converged and accurate near the inception point.

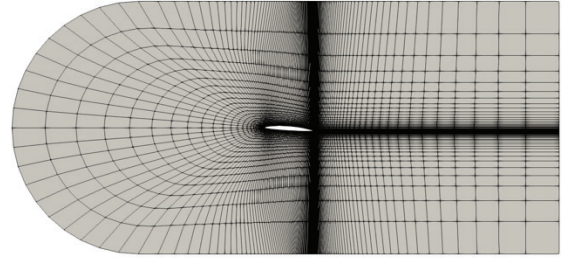


Figure 2. C-Topology sample mesh for CFD-ACE+ simulations.

In contrast, as *interPhaseChangeFoam* is an unsteady flow solver, the unrealistically small time steps required by the Courant-Friedrichs-Levy (CFL) constraint in the small cells near the trailing edge made the C-topology unsuitable. Therefore, the trailing edge was modified and rounded such that a boundary layer mesh could be fitted as shown in Fig 3.

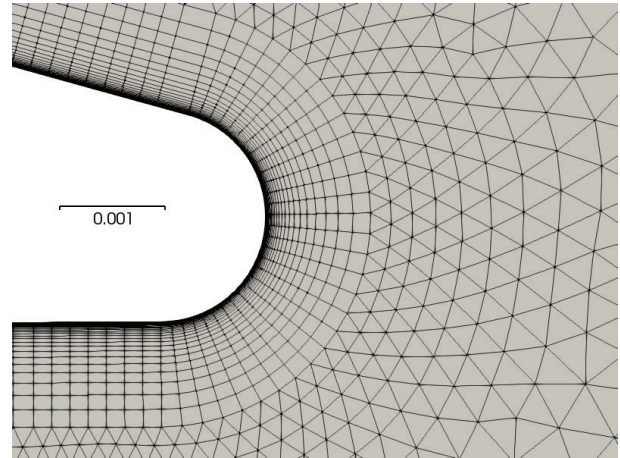


Figure 3. Modified trailing edge mesh for OpenFOAM low-Re model with 1 mm scale bar for reference.

Two simulation groups were investigated, a high-Re number $k-\omega$ SST model group with wall functions and a low Re number $k-\omega$ SST model group without wall functions but with small y^+ . All simulations were run at 4° angle of attack with a free stream velocity of 12.05 m/s.

Results and Discussion

The results from the ACE+ simulations are stable and reliable. At the highest pressures tested no regions of vapour are visible. As the pressure is decreased, vapour cavities form which, with the exception of 40 kPa, are stable with no flapping, as shown in Fig.4. At 40 kPa free stream pressure, the trailing edge of the bubble is observed in a flapping cycle where vapour bubbles separated into the free stream (not shown). In contrast, OpenFOAM's separate cavitation models all run as unsteady, with flapping and shedding observed to begin between 80 kPa to 100 kPa. Depending on the

definition of stable, no OpenFOAM simulation reached a steady state as there is always some fluctuation in the results. This is shown in the pressure field plot of Fig.5 as the vortices are visualised as low-pressure regions rolling across the upper surface of the foil and into the near wake.

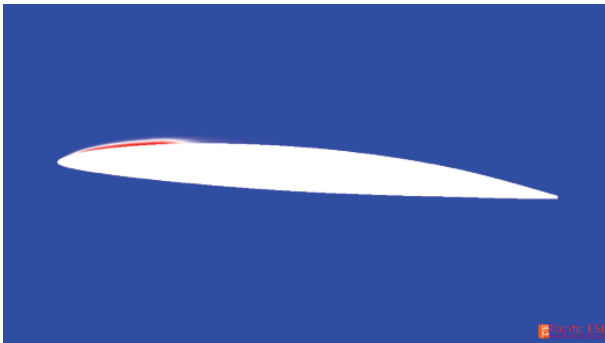


Figure 4. Indicative vapour distribution from CFD-ACE+ with a stable cavitation bubble at 60 kPa freestream pressure.

This is more conceptually appealing than the stationary results from ACE+ as cavitation, by its physical definition, is an unsteady process. The OpenFOAM flow field results therefore seem far more realistic.

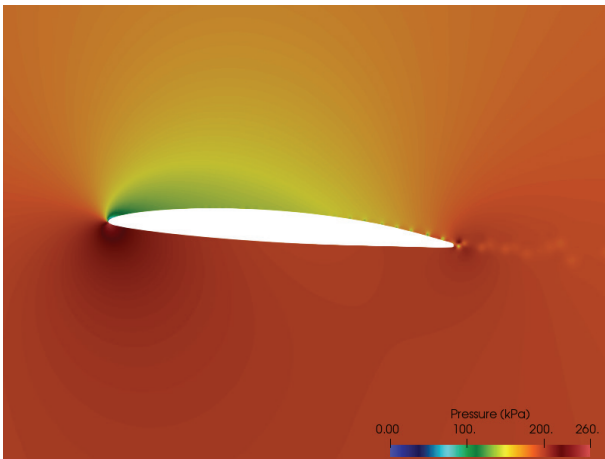


Figure 5. Indicative pressure field from a high-Re OpenFOAM model with no cavitation bubble. The free stream pressure is 180 kPa.

This then leads to a potential classification problem, i.e. how does one define the average length for an unstable cavity? The remainder of this flow field discussion will step through the OpenFOAM low-Re number results summarised for the 120 kPa, 80 kPa and 60 kPa free stream pressures as shown in Fig.5, Fig.7 and Fig.8, respectively.

Beginning with the 120 kPa simulation, the observed cavitation bubble is small, which would be expected given the experimental inception pressure is 128 kPa [3]. Of interest in Fig. 6 is the bubble forming ~ 2 mm from the leading edge and the fact that the bubble is mostly detached from the foil. This detached state is consistent with experimental images of moving bubbles and contrasts with the ACE+ result (not shown) where the bubble is continuously attached to the foil. As the free stream pressure is dropped to 80 kPa, (Fig.7) and 60 kPa (Fig.8), true unsteady phenomena are observed where isolated vapour bubbles detach from the trailing edge of the main vapour region. At 80 kPa, the unsteady region is ~ 5 mm.

While shown as a detached bubble in Fig.7, more often than not, the “detached” region does rejoin to form a much longer bubble. This result is qualitatively aligned with experimental

data that indicates that cavitation sheets can decay into bubbles. Numerically though, it highlights the complexity of choosing the location from which the cavitation bubble is measured? It could be argued that the correct location is where the gas ($f=0$) and liquid ($f=1$) phases meet, i.e. $f=0.5$, i.e. ~ 18 mm in Fig. 7, or at the last measured area of $f=0.5$ which is closer to 25 mm.

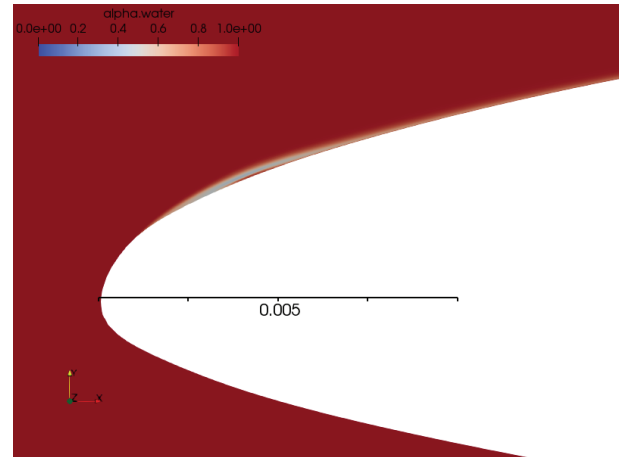


Figure 6. Indicative vapour distribution from OpenFOAM with an unstable cavitation bubble at 120 kPa freestream pressure with 5 mm scale bar.

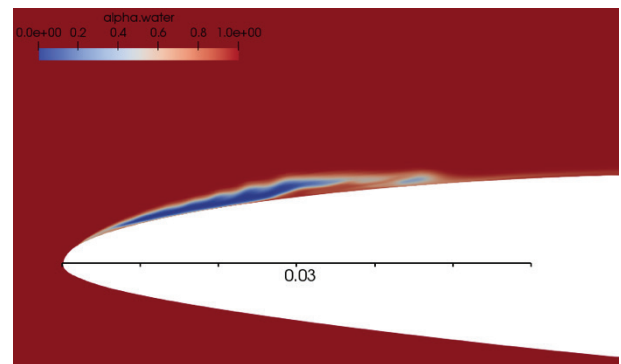


Figure 7. Indicative vapour distribution from OpenFOAM with an unstable cavitation bubble at 80 kPa freestream pressure with 30 mm scale bar.

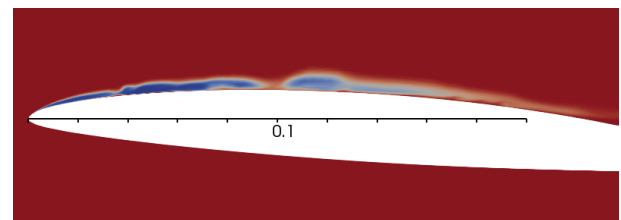


Figure 8. Indicative vapour distribution from OpenFOAM with an unstable cavitation bubble at 60 kPa freestream pressure with 100 mm scale bar.

Furthermore, this trend of bubble shedding and joining continues as the free stream pressure is dropped to 60 kPa, as shown in Fig.8. At this pressure, the cavity length would be expected to be in the order of 60 % of the chord length, or 90 mm relative to the 100 mm rule shown in Fig.8. In this case, the indicative image shows the opposite problem to Fig.7 in that the cavity is decaying quite well at 90 mm and would nominally be measured to there, yet there is a significant discontinuity at ~ 50 mm from the leading edge. For this configuration, the known result was 90 mm. However, the question remains if the 50 mm be more appropriate for an

unknown situation. With such questions still open, continuing investigations of the effect of low-Re number turbulence modelling on the Singhal full cavitation model in OpenFOAM are ongoing.

With some measurements still to be refined for the low-Re OpenFOAM models, the measured cavity lengths of from both CFD-ACE+ and the hi-Re number OpenFOAM models were reasonably accurate well at both very low pressures and very high pressures, as shown in Fig.8. However, there is a need to improve the low-Re number OpenFOAM model. As shown in Fig.8, CFD-ACE+ accurately predicts the inception point to ± 5 kPa whereas high-Re OpenFOAM does not begin cavitating until 100 kPa, nearly 30 kPa below the experimental inception pressure.

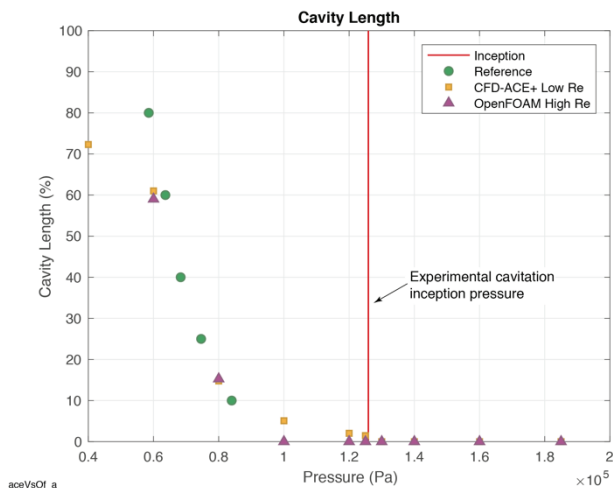


Figure 9. Cavity length from the low-Re ACE models and the high-Re OpenFOAM models. Inception is highlighted from experiments [3].

Conceptually this result is explainable because leading-edge sheet cavitation is driven by the boundary layer and the interactions of pressure and velocity fluctuations therein. In the case of a high-Re number model, the boundary layer is simply not simulated. Instead it is estimated using wall functions. We conclude that this approach is lacking. It could be concluded that, see for example Figure 6, the low-Re number OpenFOAM results are equivalent to the low-Re CFD-ACE+ results but this must be confirmed across all pressures.

Conclusions

This paper presents the initial results of a research effort to implement the Singhal Full Cavitation Model in the open source CFD code OpenFOAM. The model has been validated for 2D sheet cavitation against an experimental data set as well as an established commercial code. Limitations in the implementation have been identified at fine cell resolutions that make it more difficult to extract a single length but may in fact provide a better representation of the underlying physics by resolving the basic unsteadiness of cavitating flows.

Acknowledgments

The authors acknowledge the financial and in-kind support provided by the Australian Research Council through Linkage Project LP150100567. Insightful feedback from Mr Fred Mendonca of ESI Group's OpenCFD Ltd has been invaluable.

References

- Baloga, P., *Water Tunnel Tests on NACA-66 (modified) and YS-920 Foil Sections With and Without Surface Roughness*. 1982, Graduate Aeronautical Laboratories, California Institute of Technology: Pasadena, CA, U.S.A.
- Brockett, T., *Minimum Pressure Envelopes for Modified NACA-66 Sections With NACA $\alpha=0.8$ Camber and BUSHIPS Type I and Type II Sections*. 1966, David Taylor Model Basin: Washington D.C., U.S.A.
- Dimotakis, P.E., et al., *Two-dimensional NACA 66(MOD) Hydrofoil High Speed-Water Tunnel Tests*. 1987, Graduate Aeronautical Laboratories, California Institute of Technology: Pasadena, CA, U.S.A.
- Dreyer, M., et al., *Mind the gap: a new insight into the tip leakage vortex using stereo-PIV*. *Experiments in Fluids*, 2014. **55**: 1849.
- Erney, R.W., *Verification And Validation Of Single Phase And Cavitating Flows Using An Open Source CFD Tool*, in *Mechanical Engineering*. 2008, Pennsylvania State University: PA, U.S.A.
- ESI Group, *CFD-ACE+ 2016.0 Manual*. 2016, U.S.A.: ESI Group.
- Gaggero, S. and D. Villa, *Steady cavitating propeller performance by using OpenFOAM, StarCCM+ and a boundary element method*. *Journal of Engineering for the Maritime Environment*, 2017. **231**(2): p. 411-440.
- Kunz, R.F., et al., *Multi-phase CFD Analysis of Natural and Ventilated Cavitation about Submerged Bodies*, in *3rd ASME/JSME Joint Fluids Engineering Conference*. 1999, ASME: San Francisco, CA, U.S.A.
- Kunz, R.F., et al., *A preconditioned Navier-Stokes method for two-phase flows with application to cavitation prediction*. *Computers and Fluids*, 2000. **29**: p. 849-875.
- Lindau, J.W., et al., *APPLICATION OF PRECONDITIONED, MULTIPLE-SPECIES, NAVIER-STOKES MODELS TO CAVITATING FLOWS*, in *Fourth international symposium on cavitation*, H. Kato, Editor. 2001: Pasadena, CA, U.S.A.
- Liu, D., *The numerical simulation of propeller sheet cavitation with a new cavitation model*. *Procedia Engineering*, 2015. **126**: p. 310-314.
- Merkle, C.L., J.Z. Feng, and P.E.O. Buelow. *Computational modeling of the dynamics of sheet cavitation*. in *3rd International Symposium on Cavitation*. 1998. Grenoble, France.
- Michael, T.J., *Development and validation of a sharp interface cavitation model*, in *Mechanical Engineering*. 2013, The University of Iowa: Iowa, U.S.A.
- OpenCFD Ltd, *OpenFOAM*. 2017, OpenCFD Ltd: Bracknell, United Kingdom.
- Schnerr, G., H and J. Sauer, *Physical and Numerical Modeling of Unsteady Cavitation Dynamics*, in *4th International Conference on Multiphase Flow*. 2001: New Orleans, U.S.A.
- Shen, Y.T., *Cavitation-Free Buckets of YS-920 and NACA 66 (MOD) Foil Sections*. 1982, David W. Taylor Naval Ship Research and Development Center, Department of the Navy: Bethesda, MA, U.S.A.
- Shen, Y.T. and P.E. Dimotakis, *Viscous and Nuclei Effects on Hydrodynamic Loadings and Cavitation of a NACA 66 (MOD) Foil Section*. *Journal of Fluids Engineering*, 1989. **111**(3): p. 306-316.
- Shen, Y.T. and R. Eppler, *Wing Sections for Hydrofoils - Part 2: Nonsymmetrical Profiles*. *Journal of Ship Research*, 1981. **25**(3): p. 191-200.
- Singhal, A.K., et al., *Mathematical Basis and Validation of the Full Cavitation Model*. *Journal of Fluids Engineering*, 2002. **124**: p. 617-624.
- Ward, T.M., *The Hydrodynamics Laboratory at the California Institute of Technology—1976*. *Journal of Fluids Engineering*, 1976. **98**(4): p. 740-748.