CAV2009

7th International Symposium on Cavitation

Conference Information &
Book of Abstracts

August 16th -20th, 2009

Rackham Building,
915, E. Washington St.
University of Michigan, Ann Arbor, USA

cavitation.engin.umich.edu

UNIVERSITY OF MICHIGAN

Welcome!

On behalf of the local organizing committee and the conference co-chairs, I would like to welcome you to Ann Arbor and the 7th International Symposium on Cavitation: CAV2009.

The aim of the symposia series is to promote the worldwide exchange of cavitation knowledge. The inaugural meeting of the series was held in Sendai, Japan, in 1986. Over time, the scope and participation in this meeting has grown to encompass almost every aspect of cavitation. We have accepted 116 papers for the symposium covering a wide range of topics, including fundamental cavitation flow physics, cavitation issues associated with turbomachinery and naval systems, and new applications of cavitation in industrial and biomedical systems. We all will learn about the most recent advancements (experimental, numerical, and theoretical) in the understanding, prediction, and management of cavitating flows. Our six plenary speakers will share their insights on a range of interesting and important subjects.

I would like to thank the Scientific Committee for their help in the paper review process. Their efforts are vital to maintaining the quality of the symposium. Moreover, the technical papers judged by the Scientific Committee to be of the highest quality and interest will be selected for publication in a special issue of the *ASME Journal of Fluids Engineering*.

Finally, I would like to thank the Local Organizing Committee for all of their effort in bringing this meeting about. I would particularly like to thank the Ms. Jane Ritter, Mr. Harish Ganesh, Dr. Natasha Chang (the Chair of the Local Organizing Committee), and the UM Conference Services for their tremendous contribution to the success of the symposium.

On behalf of my conference Co-Chairs, Prof. Joseph Katz and Dr. Georges Chahine, I extend to you a warm welcome to Michigan.

Prof. Steven L. Ceccio
University of Michigan, Ann Arbor

Table of Contents

1. *Welcome note* *ii*
2. *People of CAV2009* *iv*
3. *Plenary Talks* *v*
4. *Instructions for Authors* *viii*
5. *Symposium Tour and Banquet* *viii*
6. *Schedule* *ix*
7. *Abstracts* *1*
8. *Index for abstracts* *29*
9. *Floor plan* *37*
10. *Maps* *38*
11. *Parking and Commuting* *41*
12. *Contacts Conference Organizers* *42*

People

Program Chairs

Steven Ceccio
Joseph Katz
Georges Chahine

Program Committee

Natasha A. Chang (Local Chair)
David R. Dowling
Zoran Filipi
J. Brian Fowlkes
Wei Shyy
Armin W. Troesch

Scientific Committee

Roger Arndt, *University of Minnesota*
Francois Avellan, *Ecole Polytechnique Federale de Lausanne*
Goeran Bark, *Chalmers University of Technology*
Laurence Briancon-Marjollet, *Bassin d'Essais des Carenes*
Tim Colonius, *California Institute of Technology*
Larry Crum, *University of Washington*
Luca d'Agostino, *University of Pisa*
Mohamed Farhat, *Ecole Polytechnique Federale de Lausanne*
John E. Field, *University of Cambridge*
Jean-Pierre Franc, *Laboratoire des Ecoulements Geophysiques et Industriels Grenoble*
Toshiaki Ikohagi, *Tohoku University*
Stuart Jessup, *Naval Surface Warfare Center- Carderock Division*
Hiroharu Kato, *The University of Tokio*
Valery Kedrinskii, *Lavrentyev Institute of Hydrodynamics*
Ki Han Kim, *Office of Naval Research*
Kwang-Yong Kim, *Inha University*
Spyros Kinnas, *University of Texas at Austin*
Ivan Kirschner, *Alion Science and Technology Corporation*
Gert Kuiper, *Consultant*
Detlef Lohse, *University of Twente*
Yoichiro Matsumoto, *The University of Tokyo*
Knud Aage Moerch, *Technical University of Denmark*
Kirill V. Rozhdestvensky, *Saint Petersburg State Marine Technical University*
Vladimir Serebryakov, *Inst. of Hydromechanics of Ukrainian National Academy of Science*
Bernd Stoffel, *Darmstadt University of Technology*
William Straka, *Applied Research Laboratory- Pennsylvania State University*
Shu Takagi, *The University of Tokio*
Yoshinobu Tsujimoto, *Osaka University*
Tom van Terwisga, *Maritime Research Institute Netherlands & Delft Technical University*
Yulin Wu, *Tsinghua University*

Plenary Talks

Cavitation erosion: towards a new approach – *Prof. Jean-Pierre Franc, University of Grenoble, France*

Monday, August 17 2009, 9.15-10.05 AM

Prof. Franc is the Research Director (CNRS), Turbomachinery and Cavitation Research Group, Laboratory of Geophysical and Industrial Fluid Flows (LEGI) of the Grenoble University Insitut National Polytechnique de Grenoble (INPG) and Université Joseph Fourier, France. He has published extensively in the area of cavitation, and is the author of 'La cavitation: mécanismes physiques et aspects industriels', and the co-author of 'Fundamentals of Cavitation'.

Physical and mathematical problems of hydrodynamics for high speed underwater motion with supercavitation – *Dr. Vladimir V. Serebryakov, Institute of Hydromechanics - Kiev, Ukraine*

Monday, August 17 2009, 1.00-1.50 PM

Dr. Vladimir Serebryakov, Ph.D., leading scientist of Institute of Hydromechanics of National Academy of Sciences of Ukraine, project manager is known expert in the field of High Speed Hydrodynamics including supercavitation, drag reduction and propulsive systems, dynamics and hydro elastics problems, sub-, supersonic flows in water. Double high education: shipbuilding engineering and physics-mathematics sciences. Post graduate 1969-1972 at the Institute of Hydromechanics of NASU. After that he for over 25 years has been closely collaborating with Prof. Georgy Logvinovich - father founder of the famous Russian torpedo Shkval. Dr. Serebryakov is author of asymptotic theory for axisymmetric supercavitating flows in incompressible fluid, for subsonic and supersonic speeds. He developed equations which expressed known principle of "Independence of the cavity expansion" introduced by G. Logvinovich. At present these equations are seen as one of the most effective way for practical estimation of supercavitation flows. Over 100 papers, National Award of 2002 on science and engineering, DAAD stipendium - Germany 2002, Brain Power stipendium - South Korea 2006-2007, member of sci. com. of CAV2001- USA, 2003-Japan, 2006-Netherlands, High Speed Hydrodynamics scientific school "HSH": HSH2002, 2004, 2006, 2008, SuperFAST2008 – Russia.

Numerical aspects of the collapse of non-spherical bubbles- *Prof. Hiroyuki Takahira, Osaka Prefectural University, Japan*

Tuesday, August 18 2009, 8.30-9.20 AM

Hiroyuki Takahira is currently a Professor of the Department of Mechanical Engineering at the Osaka Prefecture University. His current research interests are bubble dynamics, cavitation, gas-liquid two phase flows, and computational fluid dynamics. Hiroyuki Takahira received his B.S. and M.S. degrees in Mechanical Engineering from Kyoto University in 1985 and 1987, respectively. He received his Doctor of Engineering degree from Kyoto University in 1992. He joined Kyoto University in 1988 and subsequently worked about 8 years as an instructor and lecturer of the Department of Mechanical Engineering. In 1995, he joined Osaka Prefecture University as an associate professor of the Department of Energy Systems Engineering. He was promoted to a full professor of the Department of Mechanical Engineering at the Osaka Prefecture University in 2004. He was awarded the JSME Young Engineers Award in 1993, the JSME Medal for Outstanding Paper in 1999, and the Frontier Award of JSME Fluids Engineering Division in 2008.

Naval Propeller Cavitation: Historical Development of Design, Evaluation and Prediction- *Dr. Stuart Jessup, Naval Surface Warfare Center, Carderock Division, USA*

Tuesday, August 18 2009, 1.25-2.15 PM

Dr. Jessup attended MIT from 1970-1976 receiving his BS and MS in Ocean Engineering. He then began his career at the Naval Surface Warfare Center Carderock Division as a member of the Propulsor Branch within the Hydromechanics Department. In 1989 he received his PhD from The Catholic University of America.

Dr. Jessup developed as a propeller designer and an experimental scientist conducting research related to improving the design process and the overall quality of naval propulsors. In 1982 Dr. Jessup developed Laser Doppler Velocimetry (LDV) for use in measuring detailed propeller blade flows, including blade boundary layers. In 1988, he began the development of arbitrary propeller blade section technology for the improvement of propeller cavitation performance. This led to installation of an advanced blade section propeller on the DDG-79 Flight IIa class. In 2002 Dr. Jessup was promoted to the position of Senior Scientist for Hydrodynamics for the U.S. Navy. In recent years he has investigated unsteady flows related to the ASDS, UUV docking, and propellers operating in crashback. Presently he is working on the DDG-1000 SOE development and investigating propeller operation in heavy seas.

Dr. Jessup received The Washington Academy of Science Engineering Science Award in 1986, the NSWCCD David W. Taylor Award for Scientific Achievement in 1996, the Navy Meritorious Civilian Service Award in 2000, the ASNE American society of Naval Engineers Gold medal award in 2004 and the SNAME Davidson Medal in 2008. Dr. Jessup was also inducted into the NAE, National Academy of Engineers in 2007.

Nozzle-geometry-dependent breakup of diesel jets by ultrafast x-ray imaging: implication of in-nozzle cavitation- *Dr. Jin Wang Argonne National Lab, USA*

Thursday, August 20 2009, 8.30-9.20 AM

Dr. Jin Wang, Physicist and Group Leader for Time-Resolved Research at the Advanced Photon Source (APS) of Argonne National Laboratory (ANL), earned his doctoral degree in physical chemistry from The Ohio State University in 1994. After so, he was appointed a post-doctoral fellow at Exxon Research and Engineering Company. He continued his research at ANL in 1995 as a post-doctoral fellow, and was promoted to assistant physicist in 1997, physicist in 2001, group leader in 2003. His research interest includes emerging science and engineering on advanced combustion of conventional and alternative fossil and bio-fuels, structure-function relationships in dynamical systems. His is currently working on dynamics and structure of high-pressure, high-speed fuel sprays for energy applications, kinetics and dynamics of metal/polymer nanocomposites and interaction between high-power and short-pulse laser and solid state surfaces. Wang has co-authored or authored more than 100 journal article publications including those in Nature, Science, Nature Physics, Advanced Materials, and Physics Review Letters. Wang received numerous awards, including the Best Paper Presentation Award of the ASME Internal Combustion Engine Division in 2006, the University of Chicago Distinguished Performance Award in 2005, the US Department of Energy National Laboratory R&D Award in May 2002, the Finalist, Discover Magazine Technology Innovation Awards in 2001.

Cavitation Modeling: bridging the gap between micro- and macro-scales.- *Dr. Georges Chahine, Dynaflow, USA*

Thursday, August 20 2009, 1.25-2.15 PM

Dr. Georges Chahine, President and founder of Dynaflow has acquired a very broad academic background - civil engineering in 1970 from University St Joseph, Beirut Lebanon (ESIB), naval architecture, 1972, and Engineering Doctorate in Fluid Mechanics, 1974 (from ENSTA, Paris) and Doctorat d'Etat és-Sciences in Applied Mathematics, 1979 (U. Pierre and Marie Curie, Paris). He spent eight years in academia and led a research group on the study of interface phenomena (ENSTA, Paris), then another eight years with the engineering firm, Tracor Hydronautics Inc., directing the Fluid Mechanics and Materials Science Department before founding Dynaflow in 1988. He has published more than 300 technical papers and reports and has three patents – two on decontamination of liquids with the DynaJets® cavitating jets and one on a cross flow filtration system. Dr. Chahine has very actively contributed to the field of cavitation and bubble dynamics and has directed numerous investigations on cavitating and vortical flows, on water jet technology, and in various acoustic and hydrodynamics fields.

Instructions for Authors

Authors are allotted 20 minutes for their presentation, and 5 minutes are reserved for questions after the talk. A LCD projector will be available in each of the conference rooms. Authors are expected to bring their own portable computer. Please contact the organizers if you require the loan of a portable computer- gharish@umich.edu 734-709-7280

Symposium Tour and Banquet

CAVS 2009 attendees will spend an evening touring the Ford Rouge plant, followed by a banquet reception at the Henry Ford Museum. The tour and banquet will be held on Wednesday, August 19th.

Transportation has been arranged to take conference participants and guests to and from the venues. Buses will leave from Rackham Auditorium at 1:40 pm (after lunch). The tour of the Ford Rouge Complex will take place from approximately 3:00 pm to 5:00 pm. Then, we will return to the buses for transportation to the Henry Ford Museum. Dinner will take place from 6:00 pm to 8:00 pm, and them participants will be brought back to Rackham Auditorium.

Tentative CAV2009 Schedule

	SUNDAY 8/16/2009	MONDAY 8/17/2009	TUESDAY 8/18/2009	WEDNESDAY 8/19/2009	THURSDAY 8/20/2009	
8.00 AM		Registration/Breakfast *Rackham Building Assembly Hall, 4th Floor* ***8.00-8.45 AM***	Registration/Breakfast *Rackham Building Assembly Hall, 4th Floor* ***8.00-8.30 AM***	Registration/Breakfast *Rackham Building Assembly Hall, 4th Floor* ***8.00-8.30 AM***	Registration/Breakfast *Rackham Building Assembly Hall, 4th Floor* ***8.00-8.30 AM***	**8.00 AM**
8.05 AM						8.05 AM
8.10 AM						8.10 AM
8.15 AM						**8.15 AM**
8.20 AM						8.20 AM
8.25 AM						8.25 AM
8.30 AM			Invited Paper/Plenary 3 *Rackham Building Amiphitheatre, 4th floor Prof. Takahira* ***8.30-9.20 AM***	Tech Session 8 ***8.30 - 9.45 AM*** ***8.30 - 8.55 AM*** A22 CAVTURBO B22 NUMCOMP C22 CAVERO	Invited Paper/Plenary 3 *Rackham Building Amiphitheatre, 4th floor Dr Wang* ***8.30-9.20 AM***	**8.30 AM**
8.35 AM						8.35 AM
8.40 AM						8.40 AM
8.45 AM		Welcome Introduction *Rackham Building Amphitheater, 4th Floor* ***8.45-9.15 AM***				**8.45 AM**
8.50 AM						8.50 AM
8.55 AM				***8.55 - 9.20 AM*** A23 CAVTURBO B23 NUMCOMP C23 CAVERO		8.55 AM
9.00 AM						**9.00 AM**
9.05 AM						9.05 AM
9.10 AM						9.10 AM
9.15 AM		Invited Paper/Plenary 1 *Rackham Building Amiphitheatre, 4th floor Prof. Jean-Pierre Franc* ***9.15-10.05 AM***				**9.15 AM**
9.20 AM			Session Travel ***9.20-9.30 AM***	***9.20 - 9.45 PM*** A24 CAVTURBO B24 NUMCOMP C24 CAVERO	Session Travel ***9.20-9.30 AM***	9.20 AM
9.25 AM						9.25 AM
9.30 AM			Tech Session 4 ***9.30 - 10.45 AM*** ***9.30 - 9.55 AM*** A10 CAVNAVAL B10 NUMCOMP C10 ACCAV		Tech Session 11 ***9.30 - 10.45 AM*** ***9.30 - 9.55 AM*** A30 BUBDYN B30 CAVTURBO C30 HYDROCAV	**9.30 AM**
9.35 AM						9.35 AM
9.40 AM						9.40 AM
9.45 AM				Break *Rackham Building Assembly Hall 4th floor* ***9.45-10.05 AM***		**9.45 AM**
9.50 AM						9.50 AM
9.55 AM			***9.55 - 10.20 AM*** A11 CAVNAVAL B11 NUMCOMP C11 ACCAV		***9.55 - 10.20 AM*** A31 BUBDYN B31 CAVTURBO C31 HYDROCAV	9.55 AM
10.00 AM						**10.00 AM**
10.05 AM		Break *Rackham Building Assembly Hall 4th floor* ***10.05-10.35 AM***		Tech Session 9 ***10.05 - 11.20 AM*** ***10.05 - 10.30 AM*** A25 NUMCOMP B25 ADVEXP C25 BUBDYN		10.05 AM
10.10 AM						10.10 AM
10.15 AM						**10.15 AM**
10.20 AM			***10.20 - 10.45 PM*** A12 CAVNAVAL B12 NUMCOMP C12 ACCAV		***10.20 - 10.45 PM*** A32 BUBDYN B32 CAVTURBO C32 HYDROCAV	10.20 AM
10.25 AM						10.25 AM
10.30 AM				***10.30 - 10.55 AM*** A26 NUMCOMP B26 ADVEXP C26 BUBDYN		**10.30 AM**
10.35 AM		Session Travel ***10.35-10.45 AM***				10.35 AM
10.40 AM						10.40 AM
10.45 AM		Tech Session 1 ***10.45 - 12.00 PM*** ***10.45 - 11.10 AM*** A1 CAVNAVAL B1 NUMCOMP C1 ADVEXP	Break *Rackham Building Assembly Hall 4th floor* ***10.45-11.10 AM***		Break *Rackham Building Assembly Hall 4th floor* ***10.45-11.10 AM***	**10.45 AM**
10.50 AM						10.50 AM
10.55 AM				***10.55 - 11.20 PM*** A27 NUMCOMP B27 ADVEXP C27 BUBDYN		10.55 AM
11.00 AM						**11.00 AM**
11.05 AM						11.05 AM
11.10 AM		***11.10 - 11.35 AM*** A2 CAVNAVAL B2 NUMCOMP C2 ADVEXP	Tech Session 5 ***11.10 - 12.25 PM*** ***11.10 - 11.35 AM*** A13 CAVTURBO B13 BUBDYN C13 CAVENVI		Tech Session 12 ***11.10 - 12.25 PM*** ***11.10 - 11.35 AM*** A33 NUMCOMP B33 BIOMED C33 CAVPOW	11.10 AM
11.15 AM						**11.15 AM**
11.20 AM				Break *Rackham Building Assembly Hall 4th floor* ***11.20-11.40 AM***		11.20 AM
11.25 AM						11.25 AM
11.30 AM						**11.30 AM**
11.35 AM		***11.35 - 12.00 PM*** A3 CAVNAVAL B3 NUMCOMP C3 ADVEXP	***11.35 - 12.00 PM*** A14 CAVTURBO B14 BUBDYN C14 ADVEXP		***11.35 - 12.00 PM*** A34 NUMCOMP B34 BIOMED C34 CAVPOW	11.35 AM
11.40 AM				Tech Session 10 ***11.40 - 12.30 AM*** ***11.40 - 12.05 AM*** A28 CRYOGEN B28 NUMCOMP C28 SUPERCAV		11.40 AM
11.45 AM						**11.45 AM**
11.50 AM						11.50 AM
11.55 AM						11.55 AM
12.00 PM		Lunch *Rackham Building Assembly Hall 4th floor* ***12.00-1.00 PM***	***12.00 - 12.25 PM*** A15 CAVTURBO B15 BUBDYN C15 SONO		***12.00 - 12.25 PM*** A35 NUMCOMP B35 BIOMED C35 CAVPOW	**12.00 PM**
12.05 PM				***12.05 - 12.30 AM*** A29 CRYOGEN B29 NUMCOMP C28 SUPERCAV		12.05 PM
12.10 PM						12.10 PM
12.15 PM						**12.15 PM**
12.20 PM						12.20 PM
12.25 PM			Lunch *Rackham Building Assembly Hall 4th floor* ***12.25-1.25 PM***		Lunch *Rackham Building Assembly Hall 4th floor* ***12.25-1.25 PM***	12.25 PM
12.30 PM				Lunch *Rackham Building Assembly Hall 4th floor* ***12.30-1.30 PM***		**12.30 PM**
12.35 PM						12.35 PM
12.40 PM						12.40 PM
12.45 PM						**12.45 PM**
12.50 PM						12.50 PM
12.55 PM						12.55 PM
1.00 PM		Invited Paper/Plenary 2 *Rackham Building Amiphitheatre, 4th floor Dr. Seberyekov* ***1.00-1.50 PM***				**1.00 PM**
1.05 PM						1.05 PM
1.10 PM						1.10 PM
1.15 PM						**1.15 PM**
1.20 PM						1.20 PM
1.25 PM			Invited Paper/Plenary 2 *Rackham Building Amiphitheatre, Dr Jessup* ***1.25-2.15 PM***		Invited Paper/Plenary 2 *Rackham Building Amiphitheatre, 4th floor Dr Georges Chahine* ***1.25-2.15 PM***	1.25 PM
1.30 PM				Meet for Bus Ride ***1.30-1.40 PM***		**1.30 PM**
1.35 PM						1.35 PM
1.40 PM				BUS to Ford Rouge ***1.40 -3.00 PM***		1.40 PM
1.45 PM						**1.45 PM**
1.50 PM		Session Travel ***10.35-10.45 AM***				1.50 PM
1.55 PM						1.55 PM
2.00 PM		Tech Session 2 ***2.00 - 3.15 PM*** ***2.00 - 2.25 PM*** A4 BUBDYN B4 CAVTURBO C4 HYDROCAV				**2.00 PM**
2.05 PM						2.05 PM
2.10 PM						2.10 PM
2.15 PM			Session Travel ***2.15-2.25 PM***		Session Travel ***2.15-2.25 PM***	**2.15 PM**
2.20 PM						2.20 PM
2.25 PM		***2.25 - 2.50 PM*** A5 BUBDYN B5 CAVTURBO C5 HYDROCAV	Tech Session 6 ***2.25 - 3.40 PM*** ***2.25 - 2.50 PM*** A16 NUMCOMP B16 BIOMED C16 CAVERO		Tech Session 13 ***2.25 - 3.40 PM*** ***2.25 - 2.50 PM*** A36 NUMCOMP B36 CAVTURBO C36 HYDROCAV	2.25 PM
2.30 PM						**2.30 PM**
2.35 PM						2.35 PM
2.40 PM						2.40 PM
2.45 PM						**2.45 PM**
2.50 PM		***2.50 - 3.15 PM*** A6 BUBDYN B6 CAVTURBO C6 HYDROCAV	***2.50 - 3.15 PM*** A17 NUMCOMP B17 BIOMED C17 CAVERO		***2.50 - 3.15 PM*** A37 NUMCOMP B37 CAVTURBO C37 HYDROCAV	2.50 PM
2.55 PM						2.55 PM
3.00 PM				Ford Rouge Tour		**3.00 PM**
3.05 PM						3.05 PM
3.10 PM						3.10 PM
3.15 PM		Break *Rackham Building Assembly Hall 4th floor* ***3.15 - 3.40 PM***	***3.15 - 3.40 PM*** A18 NUMCOMP B18 BIOMED C18 CAVERO		***3.15 - 3.40 PM*** A38 NUMCOMP B38 CAVTURBO C38 HYDROCAV	**3.15 PM**
3.20 PM						3.20 PM
3.25 PM						3.25 PM
3.30 PM						**3.30 PM**
3.35 PM						3.35 PM
3.40 PM		Tech Session 3 ***3.40 - 4.55 PM***				3.40 PM

3.45 PM		***3.40 - 4.05 PM*** A7 NUMCOMP	B7 SUPERCAV	C7 CRYOGEN	Break *Rackham Building Assembly Hall 4th floor* ***3.40 - 4.05 PM***			***3.00-5.00 PM***	Closing ***3.45-4.05 PM***	**3.45 PM**
3.50 PM										3.50 PM
3.55 PM										3.55 PM
4.00 PM	Reception-Registration *Rackham Building Terrace, 4th Floor* ***4.00-6.00 PM***									**4.00 PM**
4.05 PM					Tech Session 7 ***4.05 - 5.20 PM***					4.05 PM
4.10 PM		***4.05 - 4.30 PM*** A8 NUMCOMP	B8 SUPERCAV	C8 CRYOGEN	***4.05 - 4.30 PM*** A19 CAVNAVAL	B19 NUMCOMP	C19 HYDROCAV			4.10 PM
4.15 PM										**4.15 PM**
4.20 PM										4.20 PM
4.25 PM										4.25 PM
4.30 PM										**4.30 PM**
4.35 PM		***4.30 - 4.55 PM*** A9 NUMCOMP	B9 SUPERCAV	C9 CRYOGEN	***4.30 - 4.55 PM*** A20 CAVNAVAL	B20 NUMCOMP	C20 HYDROCAV			4.35 PM
4.40 PM										4.40 PM
4.45 PM										**4.45 PM**
4.50 PM										4.50 PM
4.55 PM										4.55 PM
5.00 PM					***4.55 - 5.20 PM*** A21 CAVNAVAL	B21 NUMCOMP	C21 HYDROCAV	Break and Travel		**5.00 PM**
5.05 PM										5.05 PM
5.10 PM										5.10 PM
5.15 PM										**5.15 PM**
5.20 PM										5.20 PM
5.25 PM										5.25 PM
5.30 PM										**5.30 PM**
5.35 PM										5.35 PM
5.40 PM										5.40 PM
5.45 PM										**5.45 PM**
5.50 PM										5.50 PM
5.55 PM										5.55 PM
6.00 PM										**6.00 PM**
								Ford Musuem and Dinner ***6.00-8.00 PM***		
								Bus to Ann Arbor		
	SUNDAY 8/16/2009	MONDAY 8/17/2009			TUESDAY 8/18/2009			WEDNESDAY 8/19/2009	THURSDAY 8/20/2009	

Legend

ACCAV	Accoustic Cavitation	**CAVPOW**	Cavitation in hydraulic and power systems
ADVEXP	Advanced Experimental Methods	**CAVTURBO**	Cavitation in Turbomachineries
BIOMED	Cavitation in Biomedical Applications	**CRYOGEN**	Cryogenic Cavitation
BUBDYN	Bubble Dynamics	**HYDROCAV**	Hydrodynamic Cavitation
CAVENVI	Cavitation for environmental remediation	**NUMCOMP**	Numerical Computation of Cavitating Flows
CAVEROS	Cavitation Erosion	**SONO**	Sono-Luminescence Cavitation
CAVMANUF	Cavitation in manufacturing processes	**SUPERCAV**	Super Cavitation
CAVNAVAL	Cavitation in Naval Context		

08/17/2009 Monday

Techical Session 1 10.45 - 12.00 PM	Cavitation in Naval Context	Numerical Computation of Cavitating Flows	Advanced Experimental Techniques
	Session Chair: G Kuiper *Session Co Chair: R Arndt*	*Session Chair: Tim Colonius* *Session Co Chair: M.Farhat*	*Session Chair: B.Stoffel* *Session Co Chair: H. Kato*
10.45-11.10 AM	**70** Rudder gap flow control for cavitation suppression, *J. Oh, H. Lee, K. Shin, C. Lee, S. Rhee, J. Suh, H. Kim*	**107** A modified SST k-omega turbulence model to predict the steady and unsteady sheet cavitation on 2D and 3D hydrofoils, *D. Li, M. Grekula, P. Lindell*	**5** Interfacial cavitation nuclei studied by scanning probe microscopy techniques, *K.Mørch*
11.10-11.35 AM	**79** A panel method for trans-cavitating marine propellers, *S.Brizzolara*	**49** RANS simulations of a 3D sheet-vortex cavitation *I. Oprea, Norbert Bulten*	**13** Detection of cavitation erosion through acoustic emissions techniques, *A. Boorsma, P. Fitzsimmons*
11.35-12.00 AM	**114** Blade section design of marine propellers with maximum inception speed, *Z. Zeng, G. Kuiper*	**17** Inertia controlled instability and small scale structures of sheet and cloud cavitation, *S. Schmidt, G. Schnerr, M.Thalhamer*	**38** PIV-LIF determination of mean velocity field and reynolds stress tensor in a cavitating mixing layer, *V. Aeschlimann, S. Barre*
Techical Session 2 2.00 - 3.15 PM	Bubble Dynamics	Cavitation in Turbo-machinery	Hydrodynamic Cavitation
	Session Chair: H. Takahira *Session Co Chair: G.Bark*	*Session Chair: W.Shyy* *Session Co Chair:Y.Tsujimoto*	*Session Chair: K.Mørch* *Session Co Chair: KH Kim*
2.00-2.25 PM	**53** Influence of shock-bubble and bubble-bubble interactions on the collapse of a cluster of bubbles, *K. Kobayashi, Y. Jinbo, H. Takahira*	**59** Transient hydroelastic analysis of surface-piercing propellers, *Y. Young, B. Savander*	**112** Effects of surface characteristics on hydrofoil cavitation, *M. Williams, E. Kawakami, E. Amromin, R. Arndt*
2.25-2.50 PM	**57** On pressure and temperature waves within a cavitation bubble, *P. Pelz, A. Ferber*	**50** Cause of cavitation instabilities in three-dimensional inducer, *D. Kang, K. Yonezawa, H. Horiguchi, Y. Kawata, Y. Tsujimoto*	**182** Experimental investigations of flow structure and turbulence associated with vanishing propeller tip vortex cavitation, *Y. Chow, W. Tsai, Y. Lee*
2.50-3.15 PM	**172** Shock propagation in polydisperse bubbly flows, *K. Ando, T. Colonius, C. Brennen*	**92** Cavitation and flow instabilities in a 3-bladed axial inducer designed by means of a reduced order analytical model, *A. Cervone, L. Torre, A. Pasini, D. Baccarella, L. d'Agostino*	**9** Unsteady dynamics of cloud cavitating flows around a hydrofoil, *G. Wang, B. Zhang, B. Huang, M. Zhang*
Techical Session 3 3.40 - 4.55 PM	Numerical Computation of Cavitating Flows	Supercavitation	Cavitation in Cryogenics
	Session Chair: S.Kinnas *Session Co Chair: Y. Matsumoto*	*Session Chair: V.Serebryakov* *Session Co Chair: I. Kirschner*	*Session Chair: L.d'Agostino* *Session Co Chair: M.Farhat*
3.40-4.05 PM	**45** Cavitating propeller flows predicted by rans solver with structured grid and small reynolds number turbulence model approach, *T. Sipilä, T. Siikonen, I. Saisto, J. Martio, H. Reksoprodjo*	**110** A Dynamic test platform for evaluating control algorithms for a supercavitating vehicle, *A. Hjartarson, H.Mokhtarzadeh, E. Kawakami, G. Balas, R. Arndt*	**77** Surrogate-based modeling of cryogenic turbulent cavitating flows, *C. Tseng, W. Shyy*
4.05-4.30 PM	**73** Large eddy simulation of cavitation inception in a high speed flow over an open cavity, *E. Shams, S. Apte*	**72** High speed motion in water with supercavitation for sub-, trans-, supersonic Mach Numbers, *V. Serebryakov , I. Kirschner, G. Schnerr*	**63** Development and validation of new cryogenic cavitation model for rocket turbopump inducer, *N. Tani, S. Tsuda, N. Yamanishi, Y. Yoshida*
4.30-4.55 PM	**89** Investigation of turbulent modulation by cavitation for subgrid-scale modeling in LES, *K. Okayabashi, T. Kajishima*	**142** High-speed photography of supercavitation and multiphase flows in water entry,*H. Shi, M. Itoh*	**36** Thermodynamic effects on cryogenic cavitating flow in an orifice, *K. Niiyama, S. Hasegawa, S. Tsuda, Y. Yoshida, T. Tamura, M. Oike*

08/18/2009 Tuesday

Techical Session 4 9.30 - 10.45 PM	Cavitation in Naval Context	Numerical Computation of Cavitating Flows	Accoustic Cavitation
	Session Chair: W.Straka *Session Chair: K. Rozhdestvensky*	*Session Chair: S. Kinnas* *Session Co Chair: Y. Matsumoto*	*Session Chair: D. Dowling* *Session Co Chair: T.Colonius*
9.30-9.55AM	**100** Shallow angle water entry of ballistic projectiles. *Tadd Truscott, A.Techet, D. Beal*	**8** Numerical analysis of hydrofoil ventilated cavitation under wave impact, *E. Amromin*	**7** Weakly nonlinear analysis of dispersive waves in mixtures of liquid and gas bubbles based on a two-fluid model, *T. Kanagawa,T. Yano, M. Watanabe, S. Fujikawa*
9.55-10.20 AM	**125** The influence of aerodynamic pressure on the water-entry cavities formed by high-speed projectiles, *J. Aristoff*	**97** A boundary element method for the strongly nonlinear analysis of surface-piercing hydrofoils, *V. Vinayan, S. Kinnas*	**129** Imaging the effect of acoustically induced cavitation bubbles on the generation of shear-waves by ultrasonic radiation force, *J. Gateau, M. Pernot, M. Tanter, M. Fink*
10.20-10.45 AM	**146** Control experiments with a semi-axisymmetric supercavity and a supercavity-piercing Fin, *M. Wosnik, R. Arndt*	**94** Prediction of cavitating flow around 3-D straight/swept hydrofoils, *S. Singh, S. Kinnas*	**134** Acoustically induced and controlled micro-cavitation bubbles as active source for transcranial adaptive focusing, *J. Gateau, L. Marsac, J. Aubry, M. Pernot, M. Tanter, M. Fink*
Techical Session 5 11.10 - 12.25 PM	Cavitation in Turbo-machinery	Bubble Dynamics	Environmental Cavitation/ Sono-Luminesence
	Session Chair: W.Shyy *Session Co Chair: G.Kuiper*	*Session Chair: Jean-Pierre Franc* *Session Co Chair: B.Stoffel*	*Session Chair: K.Mørch* *Session Co Chair: J.Katz*
11.10-11.35 AM	**161** Prediction research on cavitation performance for centrifugal pumps, *W. Yong, L. Houlin, Y. Shouqi, T. Minggao, W. Kai*	**74** A hybrid lagrangian-eulerian approach for simulation of bubble dynamics, *S. Apte, E. Shams, J. Finn*	**37** Development of ballast water treatment technology by mechanochemical cavitations, *T. Yoshimura, S. Kubota, T. Seo, K. Sato*
11.35-12.00 PM	**15** Prediction of impeller speed dependence of cavitation intensity in centrifugal pump using cavitating flow simulation with bubble flow model, *M. Fukaya, Y. Tamura, Y. Matsumoto*	**46** Modeling and analysis of a cavitating vortex in 2D unsteady viscous flow, *J. Bosschers*	**103** Cavitation as a microfluidic tool, *C. Ohl, P. Quinto-Su, R. DIjkink, R. Gonzalez, F. Prabowo, X. Huang, T. Wu, V. Venugopalan*
12.00-12.25 PM	**60** Rate-dependent hydroelastic response of self-adaptive composite propellers in fully wetted and cavitating flows, *Y. Young, M. Motley*	**132** Dynamics of a vapour bubble near a thin elastic platel, *M. Shervani-Tabar, M. Shabgard, M. Rezaee, R. Zabihyan*	**166** Precursor luminescence near the collapse of laser-induced bubbles in alkali-salt solutions, *H. Chu, S. Vo, T. Hsieh*
Techical Session 6 2.25 - 3.40 PM	Numerical Computation of Cavitating Flows	Bio-medical Applications	Cavitation Erosion
	Session Chair: L.d'Agostino *Session Co Chair: M.Farhat*	*Session Chair: I. Kirschner* *Session Co Chair: S. Takagi*	*Session Chair: H.Kato* *Session Co Chair:T. Van Terwisga*
2.25-2.50 PM	**52** Numerical simulation of three-dimensional unsteady sheet cavitation, *A. Koop, H. Hoeijmakers*	**177** Damage potential of the shock-induced collapse of a gas bubble, *E. Johnsen, T. Colonius, R. Cleveland*	**33** Prediction of cavitation erosion based on the measurement of bubble collapse impact loads, *S. Hattori, T. Hirose, K. Sugiyama*
2.50-3.15 PM	**18** Unsteady bubbly cavitating nozzle flows, *C. Delale, Z. Başkaya, S. Schmidt, G. Schnerr*	**98** Removal of an obstruction from a tube by a collapsing bubble, *S. Ohl, D. Pavard, E. Klaseboer, B. Khoo*	**41** Cavitation Erosion - A critical review of physical mechanisms and erosion risk models, *T. Van Terwisga, P. Fitzsimmons, E. Foeth, Z. Li*
3.15-3.40 PM	**56** A Numerical study of unsteady cavitation on a hydrofoil, *S. Kim*	**104** Bubble shock wave interaction near biomaterials, *S. Ohl, E. Klaserboer, B. Khoo*	**32** Comparison of cavitation erosion rate with liquid impingement erosion rate, *S. Hattori, M. Takinami, T. Otani*

Techical Session 7 4.05 - 5.20 PM	Cavitation in Naval Context	Super-Cavitation	Hydrodynamic Cavitation
	Session Chair: K. Rozhdestvensky *Session Co Chair: KH Kim*	*Session Chair: I. Kirschner* *Session Co Chair: G.Kuiper*	*Session Chair: V. Serebryakov* *Session Co Chair: H.Takahira*
4.05-4.30 PM	**156** Development of measurement techniques for studying propeller erosion damage in severe wake fields, *W. Pfitsch, S. Gowing, D. Fry, M. Donnelly, S. Jessup*	**145** A simple approach to estimating three-dimensional supercavitating flow fields, *I. Kirschner, R. Chamberlin, J. Grant*	**169** Physical - mathematical bases of the principle of independence of cavity expansion, *Serebryakov V*.
4.30-4.55 PM	**30** Numerical Study on Cavitation Erosion Risk of Marine Propellers Operating in Wake Flow, *N. Hasuike, S. Yamasaki, J. Ando*	**136** Air entrainment mechanisms from artificial supercavities: Insight based on numerical simulations, *M. Kinzel, J. Lindau, R. Kunz*	**164** Observations and numerical simulations of unsteady partial cavitation on 2-d hydrofoil, *X. Peng*
4.55-5.20 PM	**16** Slip effects in vortical structure behind cavitating propeller wake. *B. Paik*	**130** Controlled supercavitation formed by a ring type wing, *V. Makhrov*	**48** Cavity flow with surface tension past a flat plate, *Y. Savchenko, Y. Semenov*

08/19/2009 Wednesday

Techical Session 8 8.30 - 9.45 AM	Cavitation in Turbomachinery	Numerical Computation of Cavitating Flows	Cavitation Erosion
	Session Chair: KH Kim *Session Co Chair: V. Serebryakov*	*Session Chair: I.Kirschner* *Session Co Chair: L.d'Agostina*	*Session Chair: JP Franc* *Session Co Chair: T. Ikohagi*
8.30-8.55AM	**149** Mechanism and scalability of tip vortex cavitation suppression by water and polymer injection. *N.Chang, R.Yakushiji, H.Ganesh, S.Ceccio*	**137** An Examination of Thermal Modeling Affects to the Numerical Prediction of Large-Scale Cavitating Fluid Flow, *M. Kinzel, J. Lindau , R. Kunz*	**67** Numerical prediction of cavitation erosion in cavitating Flow, *N. Ochiai, Y. Iga, M. Nohmi, T. Ikohagi*
8.55-9.20 AM	**122** Simulation of cavitation instabilities in inducers, *A. Hosangadi, V. Ahuja, R. Ungewitter*	**141** Numerical Prediction of Cavitation and Pressure Fluctuation around Marine Propeller, *K. Sato, A. Ohshima, H. Egashira, S. Takano*	**180** On some physics to consider in numerical simulation of erosive cavitation, *G. Bark , M. Grekula, R. Bensow, N. Berchiche*
9.20-9.45 AM	**40** Analytical investigations of thermodynamic effect on cavitation characteristics of sheet and tip leakage vortex cavitation, *S. Watanabe, A. Furukawa, Y. Yoshida, Y. Tsujimoto*	**116** Numerical modeling of cavity flow on bottom of a stepped planing hull, *M. Makasyeyev*	**131** Modeling collapse aggresiveness of cavitation bubbles in hydromachinery, *P. Zima, M. Sedlář, M. Müller*
Techical Session 9 10.05 - 11.20 AM	Numerical Computation of Cavitating Flows	Advanced Experimental Techniques	Bubble Dynamics
	Session Chair: T.Colonius *Session Co Chair: G.Bark*	*Session Chair: T. Van Terwisga* *Session Co Chair: J.Katz*	*Session Chair: W.Straka* *Session Co Chair: H.Takahira*
10.05-10.30 AM	**42** Investigation on numerical schemes in the simulation of barotropic cavitating flows, *B. Marco, B. François, S. Maria-Vittorial*	**35** Application of computer vision techniques to measure cavitation bubble volume and cavitating tip vortex diameter., *L. Savio, M. Viviani, F. Conti, M. Ferrando*	**157** Model for the oscillations of the shell of a contrast agent, liquid and solid cases, *J. Naude, F. Mendez*
10.30-10.55 PM	**138** A dual-time implicit preconditioned Navier-Stokes method for solving 2D steady/unsteady laminar cavitating/noncavitating flows using a Barotropic model, *K. Hejranfar, E. Ezzatneshan, K. Hesary*	**58** Determination of the tensile strength and the nuclei spectrum by means of the In-Situ-Nozzle, *N. Hamadeh, P. Pelz, B. Stoffel, G. Ludwig*	**183** Incepting cavitation acoustic emissions due to vortex stretching *N.Chang, S.Ceccio*
10.55-11.20 PM	**87** Assessment of a central difference finite volume scheme for modeling of cavitating flows using preconditioned multiphase Euler equations, *K. Hejranfar, K. Hesary*	**179** Cavitation phenomena in a stagnation point flowl, *Y. Lu, B. Gopalan, E. Celik, J. Katz , D. Van Wie*	**167** Gas bubble growth dynamics in a supersaturated solution: Henry's and Sievert's solubility laws, *A. Kuchma, G. Gor, F. Kuni*
Techical Session 10 11.40 - 12.30 PM	Cavitation in Cryogenics	Numerical Computation of Cavitating Flows	Super-cavitation
	Session Chair: D.Dowling *Session Co Chair: Y. Matsumoto*	*Session Chair: T.Ikohagi* *Session Co Chair: W.Shyy*	*Session Chair: KH Kim* *Session Co Chair: M.Farhat*
11.40-12.05 PM	**102** A Multi-scale study on the bubble dynamics of cryogenic cavitation, *S. Tsuda, S. Takeuchi, Y. Matsumoto, M. Koshi, N. Yamanishi*	**175** Design of cavitation-free hydrofoils by a given pressure envelope, *D. Maklakov, F. Avkhadiev*	**111** Investigation of the behavior of ventilated supercavities, E. Kawakami, M. Williams, R. Arndt
12.05-12.30 PM	**65** Thermo-fluid dynamic experiment of He II cavitating flow, *M. Murakami, K. Harada*	**68** Study on unsteady cavitating flow simulation around marine propeller using a RANS CFD code, *K. Kimura,T. Kawamura, Z. Huang, A. Fujii, T. Taketani*	**21** Supercavitating motion of a wedge in a jet, *Y. Antipov, A. Zemlyanova*
12.30-12.55 PM			**86** Drag reduction for high-speed underwater vehicles, *I. Nesteruk*

08/20/2009 Thursday

Techical Session 11 9.30 - 10.45 AM	Bubble Dynamics	Cavitation in Turbo-machinery	Hydrodynamic Cavitation
	Session Chair: KH Kim *Session Co Chair: W.Straka*	*Session Chair: M.Farhat* *Session Co Chair: I.Kirschner*	*Session Chair: H. Kato* *Session Co Chair: Y. Tsujimoto*
9.30-9.55 AM	**124** Cavitation of JP-8 fuel in a converging-diverging nozzle: Experiments and modelling, *I. Dorofeeva, F. Thomas, P. Dunn*	**11** A water test facility for liquid rocket engine turbopump cavitation testing *D. Ehrlich, J. Schwille*	**153** Numerical modeling of tip vortex cavitation modification using polymer solutions, *C. Hsiao, Q. Zhang, G. Chahine*
9.55-10.20 AM	**127** Numerical simulation of three-dimensional cavitation bubble oscillations by boundary element method, *K. Afanasiev, I. Grigorieva*	**90** Generality of rotating partial cavitation in two-dimensional cascades, *B. An, T. Kajishima, K. Okabayashi*	**152** Experimental study of the effects of viscosity and viscoelasticity on a line vortex cavitation, *C. Barbier, G. Chahine*
10.20-10.45 AM	**143** Prediction of tip vortex cavitation inception on marine propellers at an early design stage, *J. Hundemer, M. Abdel-Maksoud*	**151** Numerical Investigation of Cavitating Flow through the Cascade of Arbitrary Foil, *A. Terentiev*	**135** Cavitation analysis of a double acting podded drive during ice milling, *R. Sampson, M. Atalr, N. Sasaki*
Techical Session 12 11.10 - 12.25 PM	Numerical Computation of Cavitating Flows	Bio-medical Applications	Cavitation in Power Systems
	Session Chair: G.Chahine *Session Co Chair: I.Kirschner*	*Session Chair: D.Dowling* *Session Co Chair: Y. Matsumoto*	*Session Chair: J. Wang* *Session Co Chair: Y.Tsujimoto*
11.10-11.35 AM	**139** Vorticity confinement methods for cavitating flows, *T. Hachmann, U. Lantermann, M. Abdel-Maksoud, D. Hänel*	**173** Numerical study on the surface stability of an encapsulated microbubble in the ultrasound field, *Y. Liu, K. Sugiyama, S. Takagi, Y. Matsumoto*	**22** Numerical prediction and experimental verification of cavitation of Globe type Control Valves., *Rammohan S, Saseendran S, Kumaraswamy S*
11.35-12.00 PM	**43** The partial cavity on a 2D foil revisited, *M. Hoekstra, G. Vaz*	**109** Microbubble disruption by ultrasound and induced cavitation phenomena, *Y. Tomita, R. Uchikoshi, T. Inaba, T. Kodama*	**64** Evaluation of incipient cavitation erosion for pipe wall at downstream of an orifice, *Y. Nagaya, M. Murase, S. Mizuyama, S. Hattori*
12.00-12.25 PM	**91** Cavitation in a bulb turbine, *J. Necker, T. Aschenbrenner, W. Moser*	**117** Interaction of red blood cells with arrays of laser-induced cavitation bubbles, *P. Quinto-Su, R. Dijkink, F. Prabowo, K. Gunalan, P. Preiser, C. Ohl*	**120** The classical multicomponent nucleation theory for cavitation in water with dissolved gases, *. Němec, F. Maršík*
Techical Session 13 2.25 - 3.40 PM	Numerical Computation of Cavitating Flows	Cavitation in Turbo-machinery	Hydrodynamic Cavitation
	Session Chair: L.d'Agostino *Session Co Chair: Y.Tsujimoto*	*Session Chair: R.Arndt* *Session Co Chair: B.Stoffel*	*Session Chair: H. Takahira* *Co Chair: K. Rozhdestvensky*
2.25-2.50 PM	**154** Influence of propeller presence and cavitation on a liquid nuclei population, *R. Raju, C. Hsiao, G. Chahine*	**12** Enhancement of cavitation aggressivity around a cavitating Jet by injecting low-speed water jet for cavitation shotless peening, *H. Soyama, K. Nishizawa, M. Mikam*	**27** Study on correlation between cavitation and pressure fluctuation signal using high-speed camera System, *J. Jung, S. Lee, J. Han*
2.50-3.15 PM	**99** Multiphase flow analysis of cylinder using a new cavitation model,W. Park, C. Ha, C. Merkle	**113** Blade load dynamics in cavitating and two phase flows, *M. Kjeldsen, R. Arndt*	**61** Cavitation patterns on a plano-convex hydrofoil in a high-speed cryogenic cavitation tunnel, *Y. Ito, T. Nagayama, T. Nagasaki*
3.15-3.40 PM	**62** Numerical investigation of cloud cavitation and cavitation noise on a hydrofoil section*, J. Seo, S. Lele*	**78** Numerical investigation of thermodynamic effect on unsteady cavitation in cascade*, Y. Iga, N. Ochiai, Y. Yoshida, T. Ikohagi*	**66** Pressure-wave formation and collapses of cavitation clouds impinging on solid wall in a submerged water Jet, *K. Sato, Y. Sugimoto, S. Ohjimi*

Abstracts

5. Interfacial cavitation nuclei studied by scanning probe microscopy techniques

K.A.Mørch

The new microscopy techniques for studying solid surfaces, scanning tunneling microscopy (STM) and atomic force microscopy (AFM), also offer possibilities of studying gaseous nano-voids at solid-water interfaces, i.e. cavitation nuclei. The use of STM presupposes that both the surface studied and that of the STM-tip allow electrons to be transferred to/from the location of tunneling. To detect surface nano-voids by STM it is therefore necessary that when the submerged STM-tip during scanning of a surface meets a void, the tunneling barrier is smaller along the cavity surface than if the tip moves on along the drained solid surface below the void. Likewise, the use of AFM for void detection presupposes that the liquid-gas interface of a void can supply a detectable force on the AFMtip. Otherwise, the tip will ignore the void, and only the solid surface below it will be detected. With both techniques it has proved possible to meet the demands for detection of surface nano-voids, and today their existence is well established. However, the results obtained depend on the technique of microscopy chosen, and on how it is applied, which makes the evaluation of such measurements difficult. Therefore, an analysis of the physics related to void detection by the scanning probe microscopy (SPM) techniques is important. The present paper presents this physics on the basis of experimental results obtained with SPM techniques since the early 1990'es.

7. Weakly nonlinear analysis of dispersive waves in mixtures of liquid and gas bubbles based on a two-fluid model

T. Kanagawa	*Hokkaido University, Japan*
T. Yano	*Osaka University, Japan*
M. Watanabe	*Hokkaido University, Japan*
S. Fujikawa	*Hokkaido University, Japan*

One-dimensional nonlinear dispersive waves in liquids containing a number of microbubbles are theoretically studied based on two-fluid averaged equations derived by the present authors. The set of equations consists of the conservation laws of mass and momentum for gas and liquid phases, and the equation of motion of the bubble wall. The compressibility of liquid is taken into account, and this leads to the wave attenuation due to bubble oscillations. By using the method of multiple scales, two types of equations for nonlinear wave propagation in long ranges are derived. In a moderately low frequency band, the behavior of weakly nonlinear waves is described by the Korteweg–de Vries–Burgers equation. On the other hand, in a moderately high frequency band, the nonlinear modulation of quasi-monochromatic wave train is described by the nonlinear Schr¨odinger equation with an attenuation term.

8. Numerical analysis of hydrofoil ventilated cavitation under wave impact

E. Amromin	*Mechmath LLC*

Unsteady ventilated cavitation of a hydrofoil is analyzed with coupling of the perturbed steady two-dimensional incompressible flow of water out of the cavity and the compressible one-dimensional air flow within the cavity. The air flux from cavity at its oscillating tail and along its side boundary with the water is taken into account. The employed equations include air mass conservation law and pressure constancy condition along the cavity in both media. On the cavity boundary, however, the impermeability condition is considered from the water side and the differential momentum equation from the air side. The developed model of ventilated cavitation has been verified with the already published [1] measurements of hydrodynamic loads and their pulsations on the low-drag partially cavitating hydrofoil OK-2003. A satisfactory agreement of the computed results with experimental data was manifested. Influence of the wavelength variations and air compressibility on lift and its pulsations were analyzed.

9. Unsteady dynamics of cloud cavitating flows around a hydrofoil

G. Wang	*Beijing Institute of Technology*
B. Zhang	*Beijing Institute of Technology*
B. Huang	*Beijing Institute of Technology*
M. Zhang	*Beijing Institute of Technology*

The unsteady dynamics of cloud cavitating flow around a hydrofoil are investigated by joint experimental and numerical methods. Experiments are carried out in a rectangular test section of a cavitation tunnel. A high-speed video camera is used to visualize the unsteady flow structures. The visualized data are analyzed by using a home made soft ware. The drag and lift under the cavitation condition are measured. The spectral analysis for the measured date is conducted. The computations are conducted on the two-dimensional hydrofoil section, based on a single-fluid model of the cavitation: the liquid/vapor mixture is considered as a homogeneous fluid whose composition is regulated by mass transfer equation. The RNG k- ε turbulence model with modified eddy viscosity coefficient is used for the computations, and the modified coefficient is related to the vapor and liquid densities in cavitated regions for simulating the cavitating flow. A good agreement is obtained between experimental data and numerical simulations. The cloud cavitating area is divided in two parts: attached vapor sheet in the foreside of the cavity, and unsteady two-phase mixture in the rear region in the process of cavity breaking off. The local pressure increasing induced by the re-entrant jet is the main reason to lead the cloud cavity. The adverse pressure gradient in the rear area of the cavity is mainly responsible for the generation of the re-entrant jet.

11. A water test facility for liquid rocket engine turbopump cavitation testing

D. Ehrlich *The Aerospace Corporation*
J. Schwille *The Aerospace Corporation*

Improved understanding of the physics of turbopump cavitation and its relation to engine design parameters is needed to enhance propulsion system reliability and reduce development costs. A program to investigate cavitation phenomena in liquid rocket turbopumps has been initiated at The Aerospace Corporation to improve capability to predict the phenomena. This paper presents the methodology for the design of a new water-flow cavitation test facility capable of testing a variety of rocket turbopumps over a wide range of operating conditions to simulate the thermal characteristics of cryogenic propellants. This new cavitation test facility is now operational and qualification testing is in progress. Future experiments conducted in the facility will provide valuable data for the characterization of turbopump cavitation phenomena as well as evaluation, development, and validation of cavitation models.

12. Enhancement of cavitation aggressivity around a cavitating Jet by injecting low-speed water jet for cavitation shotless peening

H. Soyama *Tohoku University*
K. Nishizawa *Tohoku University*
M. Mikami *Tohoku University*

Cavitation impact at cavitation bubble collapse can be utilized for surface enhancement in the same way as shot peening. A peening method using cavitation impact is called cavitation peening. In the case of cavitation peening, cavitation bubbles were produced by injecting a high-speed water jet into water, i.e., a cavitating jet. In order to improve effect of cavitation peening, enhancement of cavitation aggressivity around a cavitating jet is required. In the present paper, a lowspeed water jet was injected around a cavitating jet to increase cavitation aggressivity, as the low-speed water jet eliminate residual bubbles after cavitation bubble collapse. The residual bubble causes cushion effect. The cavitation aggressivity was evaluated by an erosion test using aluminum specimen to assume that large mass loss revealed large aggressivity. The injecting condition of the low-speed water jet was optimized by the erosion test. The arc height which was height of convex curve of Almen strip was also evaluated, as the arc height was normally used to evaluate peening intensity of shot peeing. The convex curve was produced by peening as the peened surface was stretched due to plastic deformation. The peening effect was investigated by measurements of residual stress and a plate bending fatigue test using stainless steel specimen. It was revealed that a maximum cumulative erosion rate was increased about 70 % by injecting the low-speed water jet around the cavitating jet at optimum condition. The increasing rate of the arc height induced by the cavitating jet with the low-speed water jet was about five times larger than that of the cavitating jet without the low-speed water jet. The fatigue strength of stainless steel specimen was increased 29 % and 17% peened by the cavitating jet with and without the low-speed water jet comparing to that of non-peened specimen, respectively.

13. Detection of cavitation erosion through acoustic emissions techniques

A. Boorsma *Lloyd's Register ODS, UK*
P. Fitzsimmons *Lloyd's Register ODS, UK*

Cavitation erosion on propellers and rudders remains a problem in the marine industry. The consequences of failing to detect the risk of erosion damage during the design phase, and early in the service life of a vessel, include reducing the speed of the vessel, unscheduled dry-dockings and repairs or replacement of the propellers or rudders. The associated costs are borne by the builder and owner and may harm their reputations within the industry.

Lloyd’s Register has developed and tested a unique measurement system, based on acoustic emission techniques, which is capable of detecting the onset of erosion damage on propellers and rudders. The system uses high frequency transducers to quantify the impulsive energy transmitted from imploding cavitation events through the material paths of rudder, propeller and shafting configurations. The acoustic emission signals from such events have been synchronised with visual observations using high speed video equipment and borescopes.

15. Prediction of impeller speed dependence of cavitation intensity in centrifugal pump using cavitating flow simulation with bubble flow model

M. Fukaya *Mechanical Engineering Research Laboratory, Hitachi, Ltd.*
Y.I Tamura *Department of Computational Science and Engineering, Toyo University*
Y. Matsumoto *Department of Mechanical Engineering, University of Tokyo*

We developed a numerical method of estimating not only cavitation erosion area but also cavitation intensity that depends on the impeller speed of pumps. Our numerical simulation code with a ‘bubble flow model’ simulates the bubble pressure and the bubble nuclei distribution in a cavitating flow in detail. We simulated impulsive bubble pressure that varied within microseconds in a centrifugal pump. The cavitation intensity was estimated by analyzing the bubble pressure and the bubble nuclei distribution.

The erosion area on the impeller blade in our pump test was visualized by using a method involving dye. The plastic deformation rate of an aluminum sheet attached in the erosion area was measured, and the cavitation intensity was estimated using an experimental database. The erosion area and cavitation intensity were measured at high and low impeller speeds. The erosion areas were both located on the suction side of the impeller blade, and they were distributed between the shroud and the mid-point of the blade near the leading edge. The measured cavitation intensity at high-speed was twice that at low-speed.

The predicted areas of high cavitation intensity agreed well with the erosion areas in the experiment though the predicted areas slightly shifted to the leading edge. The predicted cavitation intensity at high-speed doubled that at low-speed as well as the experimental result. Therefore, we confirmed that the numerical method of estimating cavitation intensity was accurate.

Next, we added three calculations while changing the impeller speed to obtain a function of cavitation intensity variations. The predicted function was a function of the impeller speed to the power, and this also corresponded to the experimental. Our code is thus effective for estimating the cavitation intensity that increases on the suction side of the impeller blade in a centrifugal pump when the impeller speed is changed.

16. Slip effects in vertical structure behind cavitating propeller wake

B. Paik MOERI/KORDI

The cavitating propeller wake is investigated in detail using PIV and shadow graph techniques to figure out the trace ability of bubble type of tracers, naturally generated by the decrease of the static pressure in a cavitation tunnel. Experiments are conducted in the conditions of the cavitating propeller wake containing strong vortices. The flow behaviors of bubble tracers are compared with those of normal solid particles. The bubbles grown from the nuclei melted in the cavitation tunnel could not influence on the additional buoyancy, producing good trace ability in the uniform flow. The comparison between bubbles and solids shows some discrepancies in the area around the tip vortex core of cavitating propeller wake. The slip velocity, indicating the difference of moving velocities between bubble and solid, induces rather high difference in the vorticity values of it. The results of comparison in terms of vorticity values showed good agreement in the region of mild Re_S (Reynolds number based on the slip velocity), however, disagreement at high Re_S over 1000. The large slip velocity and high Re_S provided a velocity difference, especially in the high velocity gradient region. The slipstream region gives a range of $15 < Re_S < 75$; however, Re_S is about 1000 at a high velocity gradient region of a tip vortex. The fitted vorticity reduction rate would give a reference for the prediction in a real flow when bubble tracers are utilized in PIV measurements of a vortical flow. The vorticity reduction rates are obtained according to the bubble size which is considered to affect the slip effect in the high velocity gradient region.

17. Inertia controlled instability and small scale structures of sheet and cloud cavitation

S. Schmidt	*Technische Universität München*
G. Schnerr	*Technische Universität München*
M Thalhamer	*Technische Universität München*

The present investigation focuses on the numerical simulation of inertia driven dynamics of 3-D sheet and cloud cavitation on a 2-D NACA 0015 hydrofoil. Special emphasis is put on the numerical analysis of the re-entrant flow, the break-up of the sheet cavity and the formation of clouds. We demonstrate that our CFD-Tool CATUM (CAvitation Technische Universität München) is able to predict even delicate 3-D flow features such as irregular break-up patterns, cavitating hairpin and horseshoe vortices, 3-D instabilities in spanwise direction and the formation and propagation of shocks due to collapsing clouds close to the trailing edge of the hydrofoil. The numerically predicted flow features agree well with the experimental observations of Kawanami et al [1].

18. Unsteady bubbly cavitating nozzle flows

C. Delale	*Istanbul Technical University*
Z. Başkaya	*Istanbul Technical University*
S. Schmidt	*Technische Universitaet Muenchen*
G. Schnerr	*Technische Universitaet Muenchen*

Unsteady quasi-one-dimensional and two-dimensional cavitating nozzle flows are considered using a homogeneous bubbly flow model. For quasi-one-dimensional nozzle flows, the system of model equations is reduced to two evolution equations for the flow speed and bubble radius and the initial and boundary value problems for the evolution equations are formulated. Results obtained for quasi-one-dimensional nozzle flows capture the measured pressure losses due to cavitation, but they turn out to be insufficient in describing the twodimensional structures. For this reason, model equations for unsteady two-dimensional bubbly cavitating nozzle flows are considered and, by suitable decoupling, they are reduced to evolution equations for the bubble radius and for the velocity field, the latter being determined by an integro-partial differential system for the unsteady acceleration. This integropartial differential system constitutes the fundamental equations for the evolution of the dilation and vorticity in twodimensional cavitating nozzle flows. The initial and boundary value problem of the evolution equations are then discussed and a method to integrate the equations is introduced. Due to a lack of an algorithm to compute two-dimensional bubbly cavitating flows presently, the numerical simulation of 2D cavitating nozzle flows is obtained by the CFD-Tool CATUM, which is based on an equilibrium phase transition model. Results obtained for a typical cavitation cycle show instantaneous high pressure pulses at instances of cloud collapses.

21. Supercavitating motion of a wedge in a jet

Y. Antipov	*Department of Mathematics, Louisiana State University*
A. Zemlyanova	*Department of Mathematics, Louisiana State University*

The problem of determining the free surface of a jet incident on a rigid wedge and the bound- ary of a cavity behind the wedge is considered. The single- and double-spiral-vortex models by Tulin are used to describe the flow at the rear part of the cavity. The location of the wedge in the jet and the sides lengths are arbitrary. This circumstance makes the flow domain doubly connected for the single-vortex model whilst it is simple connected for the double-vortex model. Both the models are solved in closed form by the method of conformal mappings. The maps are expressed through the solutions to certain Riemann-Hilbert problems. For the former model, this problem is formulated on a genus-1 Riemann surface. The double-vortex model requires the solution to a standard Riemann-Hilbert problem on a plane. It is found that the drag and lift are practically the same whilst the jet surface, the cavity boundary at the rear part and the deflection angle of the jet at infinity are different. Also, the problem of determining the parameters for the conformal mapping in the single-vortex model has two solutions. It is shown that one of the solutions leads to a non-physical shape of the cavity and needs to be discarded. The case of a wedge in a channel with a free surface is also analyzed.

22. Numerical prediction and experimental verification of cavitation of globe type control valves.

S. Rammohan FCRI, Palakkad, India
S.Saseendran Director, FCRI, Palakkad, India
S. Kumaraswamy IIT Madras, India

Globe valves are one of the oldest types of valve used for throttling applications for all sizes due to better controllability and wider range. One of the major limitations associated with the use globe valves in liquid application is cavitation and it takes place both in part open and in fully open conditions due to varied reasons. There are different designs of globe valves available but for control valve applications, cage and plug designs are widely employed. Cage and plug design consists of body, valve cage, plug and an actuating mechanism. Actuating mechanism is connected to the valve plug which is a moving part, through valve shaft. There are many investigations reported about the flow visualization and numerical simulation of normal type globe valves. But study on valves with cage and plug design are not reported in detail. The objective of the present study is to provide a three dimensional analysis of flow through a globe valve with cage and plug design with emphasis on the inception and development of cavitation in detail. Cavitation reduction is achieved by breaking the flow in the form of more than one liquid jet, thereby increasing the turbulence inside the valve flow path. The numerical simulation was done using GAMBIT to set up geometry and grid and FLUENT to solve difference equation postulated from the conservation of mass and momentum of the fluid in motion. The k-epsilon model was used for turbulence. Results of five configurations of the cage with constant flow areas and valve stroke are presented in this paper. The numerical results were verified with an experimental program employing total flow measurement and pressure drop created by the valve at full opening. The study was conducted for different jet configurations to generalize the results of the study. Experimental validation was done in the water test facility with an operating pressure of 1.6 MPa and flow rate of 0.05 m^3/s. In the study, total area of opening for the flow and the valve stroke were kept constant. Accelerometers and dynamic pressure sensors were employed to sense the severity of cavitation at different differential pressures across the test valve.

27. Study on correlation between cavitation and pressure fluctuation signal using high-speed camera system

J. Jung Samsung Heavy Industries
S. Lee Samsung Heavy Industries
J. Han Samsung Heavy Industries

This paper is presenting the correlation between the cavitation and the pressure fluctuation of a propeller. Cavitation images and pressure fluctuation signals were simultaneously acquired by a high-speed camera system connected to pressure sensors for the correlation analysis. The analysis is focusing on the growing and collapsing process of the cavitation and its corresponding patterns of pressure signals. Especially, the relationship between the shape variations of the pressure fluctuation signal and the variations of pressure amplitudes for both the first and second blade frequency components is also studied. From the analysis on the correlation between cavitation behaviors and pressure fluctuation signals, comprehensive explanations on the fundamental mechanisms of the pressure signal generation and on the moment of the peak creation in the pressure signals are found. The shape of overall pressure fluctuation signal on hull surface is affected by not only the effect of cavitation from a single blade but also the superposition of the pressure signals from two adjacent blades when the collapsing of the foregoing blade's cavity and the growing of the following blade's cavity are happening at the same time. In this manner, the overlapping phenomena cause the changes of combination of amplitude levels of each blade frequency.

30. Numerical study on cavitation erosion risk of marine propellers operating in wake flow

N. Hasuike Nakashima Propeller Co., Ltd.
S. Yamasaki Nakashima Propeller Co., Ltd.
J. Ando Kyushu University

This paper discusses the application of the CFD to cavitating flow around marine propellers operating in ship wake. Especially the emphasis was put on the tip vortex cavitation and the erosive cavitation around the trailing edge. This research found that adaptive mesh refinement methodology was effective for the resolution of tip vortex cavitation. Next, barotropy model and full cavitation model were validated, and show qualitatively agreement with the experimental results. Finally, simple four cavitation erosion indexes were applied for the estimation of the erosion risk and one index shows good agreement with the experimental results. It is concluded that the RANS CFD gives the valuable information for judging the erosion risk although its presumption accuracy and numerical stability need to be improved.

32. Comparison of cavitation erosion rate with liquid impingement erosion rate.

S. Hattori University of Fukui
M. Takinami University of Fukui
T. Otani University of Fukui

Both cavitation erosion and liquid impingement erosion are phenomena that can cause pipe wall thinning in power plants. The Code for Power Generation Facilities, Rules on Pipe Wall Thinning Management, was published by the JSME (Japan Society of Mechanical Engineers) in 2005. The code says that cavitation erosion shall be prevented either in the design stage or by daily inspection. On the other hand, liquid impingement erosion can occur in any location where a working fluid attacks a pipe wall at high flow velocities. Therefore, it is very important to evaluate the amount of erosion by liquid impingement for pipe steels quantitatively from the viewpoint of aging management. In this study, we carried out both cavitation erosion and liquid impingement erosion tests, and clarified the relation between the two erosion rates. As a result, we found that the erosion rate by cavitation increases in proportion with the 5.2th to 6.8th power of the flow velocity and that by liquid impingement with the 6.0th to 7.4th power. Moreover, a good correlation was obtained between erosion rates by cavitation and by liquid impingement. We also discussed the erosion mechanism with SEM photography, and proposed an erosion model.

33. Prediction of cavitation erosion based on the measurement of bubble collapse impact loads

S. Hattori	*University of Fukui*
T. Hirose	*University of Fukui*
K. Sugiyama	*Ebara Research Co., Ltd.*

The prediction of cavitation erosion rates is important in order to evaluate the exact life of components in fluid machineries. The measurement of impact loads in bubble collapses helps us to predict the life under cavitation erosion. In this study, we carried out the erosion tests and measured impact loads in bubble collapses with a cavitating liquid jet apparatus which complies with the ASTM G134-95 standard. The bubble collapse impact loads were measured by a piezo ceramic transducer in a cavitating liquid jet apparatus. To produce various cavitation conditions, the flow velocity was changed from 184 down to 80 m/s. We evaluated the incubation period based on a cumulative damage rule by measuring the impact loads of cavitation acting on the specimen surface and by using the “constant impact load – number of impact curve” similar to the modified Miner’s rule which is employed for fatigue life prediction. We found that the parameter $\Sigma(F_i^{\alpha}\times n_i)$ (F_i: impact load, n_i: number of impacts and α: constant) is suitable for the evaluation of the erosion life. After the constant α has been obtained under two different cavitation conditions, we can predict the incubation period with the cavitating liquid jet method under yet another condition, provided that the bubble collapse impact loads are measured.

35. Application of computer vision techniques to measure cavitation bubble volume and cavitating tip vortex diameter.

L. Savio	Dinav Genoa University
M. Viviani	Dinav Genoa University
F. Conti	Fincantieri C.N.I
M. Ferrando	Dinav Genoa University

In present paper application of computer vision techniques to propeller cavitation experiments is presented. These techniques are widely adopted in many different environments and therefore they are well documented. They are also attractive from an economic point of view, due to relative low cost of the hardware involved. Nevertheless their application to study propeller behavior in cavitation tunnel is not straightforward, because of the nonstandard environment. However the adoption of these techniques may open a wide field of investigation and can result in a deepening of knowledge in propeller cavitation phenomena. In particular, obtained data can be linked to connected topics, such as propeller radiated noise or pressure signature, providing a better understanding on the sources of these effects, and invaluable information for validation of computer simulations. Present paper traces a possible path to develop an experimental technique, covering theoretical points as well as data analysis strategies and other practical aspects. All techniques are presented through practical application, thus making clearer their points of strength and their shortcomings. Besides achieved results, possible improvements and future developments are outlined.

36. Thermodynamic effects on cryogenic cavitating flow in an orifice

K. Niiyama	*Japan Aerospace Exploration Agency*
S.i Hasegawa	*Japan Aerospace Exploration Agency*
S. Tsuda	*Japan Aerospace Exploration Agency*
Y. Yoshida	*Japan Aerospace Exploration Agency*
T. Tamura	*Foundation for Promotion of Japanese Aerospace Technology*
M. Oike	*Ishinomaki Senshu University*

Temperature depression in a cavitating orifice flow was experimentally investigated with liquid nitrogen in order to clarify the influence of turbulent flow around a bubble on thermodynamic effects on cavitation. The temperature began to decrease at the outlet of the orifice when the cavitation number decreased below 0.84. Moreover, the temperature depression became larger as the cavitation number became smaller. In addition, the temperature depression also became greater as the flow velocity became lower when the cavitation numbers were equal. Based on theoretical considerations and experimental results, the difference of temperature depression can be considered to be caused by the enhancement of thermal transport around bubbles due to the turbulent flow. In addition, if thermal transport is enhanced as mentioned above, the temperature in the area where the cavitation collapses can become higher than that upstream of the orifice due to the temporary breakdown of the heat balance between the inception and collapse of cavity bubbles.

37. Development of ballast water treatment technology by mechanochemical cavitations

T. Yoshimura	*Kure National College of Technology*
S. Kubota	*Kure National College of Technology*
T. Seo	*Kure National College of Technology*
K. Sato	*Babcock-Hitachi K. K.*

In order to solve environmental problems caused by ballast water, which is a serious problem worldwide, ballast water purification using cavitation was investigated. Previously, only the collapse pressure of cavitation was used for the treatment of planktons. In this study, however, processing by cavitation containing a chemical and the mechanical processing of cavitation were applied. A Venturi nozzle and an ejector nozzle were used to inject sodium hypochlorite. It was found that the ejector nozzle had higher processing performance than the Venturi nozzle, and was effective for high flow rates of ballast water. It was clarified that high plankton extinction ratios could be obtained using mechanochemical cavitation provided by ejector nozzles.

38. PIV-LIF determination of mean velocity field and Reynolds stress tensor in a cavitating mixing layer

V. Aeschlimann LEGI
S. Barre LEGI – CNRS

The purpose of this experimental study was to analyze a 2D cavitating shear layer. The global aim of this work was a better understanding and modeling of cavitation phenomena from a 2D turbulent shear flow to rocket engine turbopomp inducers. This 2D mixing layer flow provided us a well documented test case to be used for comparison between the behavior with and without cavitation. Similarities and differences led to characterize effects of the cavitation on the flow dynamic. The run fluid was liquid water. The experimental facility allowed us to set two distinct configurations with different cavitation levels:
- CDM: a mixing layer flow (U1 = 15.8 m/s for the high speed side and U2= 3.5 m/s for the low speed side)
- MD: a downward facing step flow (U1 = 13.5 m/s and U2 = 0 m/s).

The development of Kelvin-Helmholtz instabilities was observed at the interface. Vaporizations and implosions of cavitating structures inside the vortices were also observed.

PIV-LIF(Particle Image Velocimetry – Laser Induced Fluorescence) system was used to measure the velocity of the liquid phase. Instantaneous velocity fields were measured in the whole flow. The Self similarity of the flow was characterized by the dimensionless analysis of the mean and fluctuating velocity fields. Parameters that characterized the flow dynamic were studied and quantified: Vorticity thickness, growth rate and Reynolds tensor components. Turbulent kinetic energy and the anisotropy tensor components were also analyzed and estimated. General behaviors of the two configurations have been observed:
- In the CDM case the mixing area developed along the x-axis a turbulent shear area, growing linearly, showing a constant growth rate over the studied cavitation parameter range.
- The MD case was more complex, presenting a flow separation with a large recirculating area and a quite large positive pressure gradient. The reattachment point moved depending on the cavitation level. The recirculating area seemed to have an unsteady behavior and was certainly pulsing and shedding vortices downstream. Successive vaporizations and condensations of the fluid particles inside the turbulent area have generated additional velocity fluctuations due to the strong density changes associated with the vaporization and condensation processes. However, the mean spatial development of the mixing area was only barely affected over the studied cavitation number range. The main results of this study clearly showed that the turbulence-cavitation relationship inside a mixing layer is not only driven by a simple change of compressibility properties of the fluid in the turbulent field due to the presence of a twophase flow.

40. Analytical investigations of thermodynamic effect on cavitation characteristics of sheet and tip leakage vortex cavitation

S.I Watanabe Department of Mechanical Engineering, Kyushu University
A.I Furukawa Department of Mechanical Engineering, Kyushu University
Y. Yoshida Japan Aerospace Exploration Agency
Y. Tsujimoto Graduate School of Engineering Science, Osaka University

Vapor production in cavitation extracts the latent heat of evaporation from the surrounding liquid, which decreases the local temperature, and hence the local vapor pressure in the vicinity of cavity. This is called thermodynamic/thermal effect of cavitation. In the present study, the thermodynamic effect on cavitation characteristics such as cavitation compliance and mass flow gain factor, which are known to be important parameters for cavitation instabilities appearing in turbopumps, were studied. Main cavitations in turbopumps, blade and tip leakage vortex cavitations were separately analyzed by simple analytical methods developed based on the potential flow theory, taking account of the latent heat extraction and heat transfer between the cavity and the surrounding fluid. The cavitation characteristics were estimated for the partial cavity and the tip leakage vortex cavity, and the thermodynamic effects on those characteristics were discussed.

41. Cavitation Erosion: A critical review of physical mechanisms and erosion risk models

T. Van Terwisga MARIN
P. Fitzsimmons Lloyd's Register
E. Foeth MARIN
Z. Li Delft University of Technology

This work aims at a postprocessing procedure for the assessment of the cavitation erosion risk based on multiphase CFD results an on experimental observations. Existing procedures often use available information, such as the rms value of the vapour fraction in a particular area, without thorough justification of this criterion. This paper aims at linking the available information that comes from multiphase RANS or experimental observations with High Speed Video, with the many publications on fundamental mechanisms of cavitation dynamics. The first objective of this paper is to review physical mechanisms for cavitation erosion loads that have been suggested in the literature. These mechanisms are evaluated with observations that are available from full scale ships where cavitation has lead to erosion damage on the rudder or the propeller. A second objective is to review risk assessment models that use CFD results or experimental results as input for the prediction of the risk of cavitation erosion. A detailed phenomenological description of the process leading to cavitation erosion from sheet cavitation and vortex cavitation is hypothesized. The process is based on the conversion of potential energy contained in the cavity and a focusing of this energy in space and in time that is governed by ring vortices or horseshoe vortices in case the ring vortices attach to a surface. It is concluded from experiments by Kawanami etal. [1] that horseshoe vortices tend to concentrate the vorticity toward the material surface. It is hypothesized in this paper that this concentrated vorticity forms a mechanism to break a monolithic cavity up into bubbles due to instabilities caused by the high fluid velocities and to concentrate all microbubbles in space by centrifuging out the heavier liquid particles. The radiated shockwaves caused by the implosion of one microbubble is then hypothesized to be sufficient to initiate a synchronized implosion of the cloud of microbubbles in the immediate vicinity. A selection of cavitation erosion models available in open literature is reviewed in this paper. It is concluded that the models by Bark et al. [2] and Fortes-Patella et al. [3] appear to offer the best frameworks to be coupled to the mechanisms hypothesized in this paper.

42. Investigation on numerical schemes in the simulation of barotropic cavitating flows

M. Bilanceri	*University of Pisa*
F. Beux	*University of Pisa*
M. Salvetti	*University of Pisa*

A numerical methodology for the simulation of cavitating flows is considered. A homogeneous-flow cavitation model, accounting for thermal effects and active nuclei concentration, is considered, which leads to a barotropic state law. The continuity and momentum equations for compressible inviscid flows are discretized through a finite-volume approach, applicable to unstructured grids. The numerical fluxes are computed by shockcapturing schemes and ad-hoc preconditioning is used to avoid accuracy problems in the low-Mach regime. Second-order accuracy in space is obtained through MUSCL reconstruction. Time advancing is carried out by an implicit linearized scheme. Two different numerical fluxes are investigated here, viz. the Roe and the Rusanov schemes. For the Rusanov flux two different time linearizations are proposed; in the first one the upwind part of the flux function is frozen in time, while in the second one its time variation is taken into account, although in an approximated manner. The different schemes and the different linearizations are appraised for the quasi 1D-flow in a nozzle through comparison against exact solutions and for the flow around a hydrofoil mounted in a wind tunnel through comparison against experimental data. Non-cavitating and cavitating conditions are simulated. It is shown that, for cavitating conditions, the Rusanov scheme together with the more complete time linearization allows time steps much larger than for the Roe scheme to be used. Finally, the results obtained with this scheme are in good agreement with the exact solutions or the experimental data for all the considered test cases.

43. The partial cavity on a 2D foil revisited

M. Hoekstra	*MARIN, The Netherlands*
G. Vaz	*MARIN, The Netherlands*

The partial cavity on a 2D NACA0015 foil at 6 degrees angle-of-attack is studied numerically. Assuming the fluid to be a continuum of variable density, we solve the RANS equations, complemented with turbulence and cavitation models. Some important details of the mathematical model are pointed out first. We study then carefully what occurs in the numerical simulations in and near the cavity from the inception phase to the stage well before serious unsteadiness (cavity shedding) starts. By making the computations on grids of different densities we get an impression of numerical uncertainties. This is important for the interpretation and the subsequent comparison with what experimental investigations have learned us about the physics of these almost steady partial cavities on foils. The results show that close to inception a cavity exists while the boundary layer is non-separating. The liquid-vapour interface turns out not to be a material surface, neither at the front end nor at the tail of the cavity. It also appears that the widely accepted re-entrant jet model as conceived from free-streamline theory is not a good description of the flow at the tail. The confrontation of the numerical results with information from experiments indicates that there is agreement and corroboration in several respects, but also intriguing discrepancies are found which require further elucidation.

45. Cavitating propeller flows predicted by RANS solver with structured grid and small reynolds number turbulence model approach

T. Sipilä	*VTT Technical Research Centre of Finland*
T. Siikonen	*Helsinki University of Technology*
I. Saisto	*VTT Technical Research Centre of Finland*
J. Martio	*VTT Technical Research Centre of Finland*
H. Reksoprodjo	*Helsinki University of Technology*

Within the EU research project VIRTUE, a propeller is investigated in uniform and nonuniform inflow conditions by means of a RANS equation solver, FINFLO. The analyses are made in wetted and cavitating conditions. The propeller analyzed in this paper is the INSEAN E779A propeller. The paper contains calculations at three different grid resolutions in wetted conditions and at the two finest grid resolutions in cavitating conditions in uniform inflow. The mediumsize grid is used for the propeller in nonuniform inflow simulations. The simulations are conducted on a model scale and the results are compared with the measurements and cavitation tests performed by INSEAN. The nonuniform inflow is generated by modeling the geometry of the artificial wake generator used in the cavitation tests in the calculation domain. The experimental results are published in several papers, for example in [1] and [2].

The predicted propeller open water thrust and torque are found to be within 5 % of the measured ones. The pressure peak at the leading edge of a blade is found to be sensitive to the grid resolution. The predicted cavitation behavior of the propeller blades is in reasonable accordance with the cavitation test observations. In uniform inflow the vaporized region is overpredicted. Contrastingly, the vaporized region is Underpredicted in the nonuniform inflow calculations. Side entrant jets could be identified in the cavity region in the nonuniform inflow simulations. The predicted vaporized regions in several blade positions together with photographs of the cavitating propeller are shown for comparison. The cavitation behavior trends seemed to be similar in the simulations and observations in nonuniform inflow, except that the rollup of detached sheet cavitation into a tip vortex could not be captured in the calculations. The total wake is measured between the propeller plane and the wake generator. The predicted wake is found to be too strong, but the width of the wake is relatively close to the measurements. The propeller loading history is shown over one propeller revolution. It shows qualitatively reasonable trends.

The loading histories of the wetted and cavitating propeller are almost the same due to the relative small cavitating region in the investigated conditions. The pressure distributions at several blade positions on the suction side of the propeller are shown in wetted and cavitating conditions for comparison.

46. Modeling and analysis of a cavitating vortex in 2D unsteady viscous flow

J. Bosschers *MARIN*

Two models, of which one analytical and the other computational, have been developed to describe an axisymmetric cavitating vortex in two-dimensional, unsteady, incompressible and viscous flow. The models are used to investigate the influence of viscosity on the flow structure, cavity size and cavity resonance frequency. The analytical formulation is an extension of the Lamb-Oseen vortex and has not been presented before. It is derived under the assumption of small radial velocity equivalent to small temporal changes of the cavity diameter. The computational model solves the equations without simplifications. It is shown that viscous effects have a significant influence on cavity radius and resonance frequency if the cavity size is on the same order or smaller than the viscous core size.

48. Cavity flow with surface tension past a flat plate

Y. Savchenko	*Institute of Hydromechanics*
Y. Semenov	*Institute of Hydromechanics*

The effect of surface tension forces on the cavity flow parameters is considered for steady flow past a rounded-edge plate perpendicular to the incident flow. The fluid is assumed to be ideal, weightless, and incompressible. The problem is solved in a parameter region by finding analytical expressions for the flow potential and the function that conformally maps the parameter region into the flow region in the physical plane. The dynamic boundary condition includes the surface tension force, which is proportional to the free boundary curvature, and allows one to obtain an integral equation in the velocity magnitude on the free boundary. The integral equation is solved numerically by the method of successive approximations. The results of calculations of the effect of the Weber number and the plate edge radius on the geometry of the cavity and the drag coefficient of the plate are presented.

49. RANS simulations of a 3D sheet-vortex cavitation

I. Oprea	*Wärtsilä Netherlands B.V*
N. Bulten	*Wärtsilä Netherlands B.V*

On marine propellers, cavitation appearance and development is critical for performance and erosion considerations. Behind a ship, the propeller experiences all kinds of cavitation types, varying from sheet and bubbles to tip vortex cavitation. When a cavitation analysis is required, two methods are available: experimental or numerical. To find the optimum propeller that fits into different configurations and requirements, designers need accurate predictions within reasonable time. The experimental method is typically used at the end of a design process to verify performance. Therefore, quick and accurate numerical predictions are essential at different stages in the design process, to evaluate performance and cavitation patterns. Mathematical methods range from basic panel codes to the more complex ones, derived from the Navier-Stokes equations. Methods like DES and LES require large meshes and small time steps which makes their usability limited. The most practical viscous numerical method available at the moment in industry is Reynolds Average Navier-Stokes (RANS). The current paper will present the results of a RANS simulation of a 2D sheet cavity and a 3D sheet-tip vortex cavitation. Accurate results of these basic simulations are steps towards the end goal, cavitating propeller simulations. In this method the viscous effects are taken into account with aid of a two equation turbulence model, which results in a reasonably fast approach due to reasonably grids requirements. It is concluded that the RANS method can predict complex 3D sheet-vortex cavitation development and shedding. In addition, it is appropriate for industrial use because it achieves reasonably quick and accurate results. As a next step in the research project, the cavitation development on a propeller will be analyzed with this method.

50. Cause of cavitation instabilities in three-dimensional inducer

D. Kang	*Osaka University*
K. Yonezawa	*Osaka University*
H. Horiguchi	*Osaka University*
Y. Kawata	*Osaka Institute of Technology*
Y. Tsujimoto	*Osaka University*

Alternate blade cavitation, rotating cavitation and cavitation surge in rocket turbopump inducers were simulated by a commercial CFD code. In order to clarify the cause of instabilities, the velocity disturbance caused by cavitation was obtained by subtracting the velocity vector under noncavitating condition from that under cavitating condition. It was found that there exists a disturbance flow towards the trailing edge of the tip cavity. This flow has an axial flow component towards downstream which reduces the incidence angle to the next blade. It was found that all of the cavitation instabilities start to occur when this flow starts to interact with the leading edge of the next blade. The existence of the disturbance flow was validated by experiments.

52. Numerical simulation of three-dimensional unsteady sheet cavitation

A.H. Koop	*University of Twente*
H.W.M. Hoeijmakers	*University of Twente*

The shedding of a sheet cavity is governed by the direction and momentum of re-entrant and side-entrant jets and their impingement on the free surface of the cavity. Therefore, for an accurate prediction of the shedding of the sheet cavity it is important to predict the re-entrant and side-entrant jets accurately. It appears that these jets are inertia driven suggesting that a numerical method based on the Euler equations is able to capture the phenomena associated with unsteady sheet cavitation. Due to the dynamics of sheet cavitation, strong pressure pulses are generated, originating from the collapse of shed vapor structures. To be able to predict the dynamics of the pressure waves the fluid is considered as a compressible medium by adopting appropriate equations of state for the liquid phase, the two-phase mixture and the vapor phase of the fluid.

In this paper a computational method for solving the compressible unsteady Euler equations on unstructured grids is employed to predict the structure and dynamics of threedimensional unsteady sheet cavitation occurring on stationary hydrofoils, placed in a steady uniform flow. In the two-phase flow region an equilibrium cavitation model is employed, which assumes local thermodynamic and mechanical equilibrium. In this model the phase transition does not depend on empirical constants to be specified.

The three-dimensional unsteady cavitating flow about a three-dimensional hydrofoil (Twist11) is calculated. It is shown that the formation of the re-entrant flow and a cavitating horseshoe vortex are captured by the present numerical method. The calculated results agree reasonably well with experimental observations. Furthermore, it is demonstrated that the collapse of the shed vapor structures and the resulting high pressure pulses are captured in the numerical simulations.

53. Influence of shock-bubble and bubble-bubble interactions on the collapse of a cluster of bubbles

K. Kobayashi *Osaka Prefecture University*
Y. Jinbo *Osaka Prefecture University*
H.Takahira *Osaka Prefecture University*

The present work is concerned with direct numerical simulations for the shock-bubble and bubble-bubble interactions using the improved ghost fluid method in which the Riemann solutions are utilized to diminish numerical oscillations near interfaces. The influence of bubble size and bubble-bubble distance on the collapse of in-line two bubbles and in-line three bubbles are investigated to understand the conditions under which the bubble-bubble interactions accelerate or decelerate the bubble collapse. It is shown that when the in-line bubbles collapse by the incident shock wave, the collapse of the downstream bubble can be accelerated by the shock waves due to the collapse of upstream bubbles, while the collapse of the upstream bubbles is decelerated by the expansion wave cased by the reflection of the incident shock wave at the surface of the downstream bubbles. Also, there exists the bubble-bubble distance in which the collapse of the downstream bubble is most accelerated. When the downstream bubble is smaller than the upstream bubble, the downstream bubble collapses more violently than the single bubble at any distance between the bubbles. The phase of the downstream bubble at the impact of the shock waves generated from the upstream bubbles is important in determining the acceleration of the collapse of the downstream bubble. It is also shown that the pressure increase in liquid near the axis of symmetry around the downstream bubble increases with the increase of number of bubbles and with the decrease of the bubble-bubble distance because the pressure increase in liquid is caused by the superposition of the shock waves generated from all bubbles.

56. A numerical study of unsteady cavitation on a hydrofoil

S. Kim *NSWCCD*

The unsteady turbulent cavitation on a hydrofoil with finite span and NACA-0015 section is studied numerically. For the computations, a two-phase flow approach based on homogeneous mixture approximation is adopted in which liquid-vapor mixture is modeled as an inter-penetrating continuum with the phase compositions represented by volume- fraction and the inter-phase mass transfer computed using a finite-rate model derived from bubble dynamics. An implicit, finite-volume based projection algorithm was developed that couples velocity, phase compositions, and pressure. Turbulence is modeled using Reynolds-averaged Navier-Stokes (RANS), large eddy simulation, and RANS/LES hybrid approaches. A suite of multiphase computational fluid dynamics (MCFD) solvers was built using OpenFOAM, an object-oriented, open-source CFD tool-kit, being validated for steady and unsteady cavitating flows on hydrofoils and marine propulsors. The large eddy simulation (LES) and the RANS/LES hybrid results on the hydrofoil closely reproduced the salient features of the unsteady sheet/cloud cavitation such as the breakup of sheet cavity by re-entrant jet, and the formation and collapse of cloud cavity. The lift and drag force predictions in a range of cavitation number were also found to be in good agreement with the experimental data in terms of the mean values, the root-mean-square values, and the spectral contents.

57. On pressure and temperature waves within a cavitation bubble

P. Pelz *Chair of Fluid Systems, University of Technology Darmstadt*
A. Ferber *Chair of Fluid Systems, University of Technology Darmstadt*

The presented work is about the detailed pressure, temperature and velocity distribution within a plane, cylindrical and spherical cavitation bubble. The review of Plesset & Prosperetti (1977) and more recently the review of Feng & Leal (1997) describe the time behavior of the gas within a spherical bubble due to forced harmonic oscillations of the bubble wall. We reconsider and extend those previous works by developing from the conversation laws and the ideal gas law a boundary value problem for the distribution of temperature and velocity amplitude within the bubble. This is done for a plane, cylindrical, or spherical bubble. The consequences due to shape differences are discussed. The results show that an oscillating temperature boundary layer is formed in which the heat conduction takes places. With increasing dimensionless frequency, i.e. Péclet number, the boundary-layer thickness decreases and compression modulus approaches its adiabatic value. This adiabatic behaviour is reached at lower frequencies for the plane geometry in comparison with cylindrical and spherical geometry. This is due to the difference in the volume specific surface, which is 1, 2, 3 times the inverse bubble height/radius for the plane, cylindrical and spherical bubble respectively. For the plane bubble the analysis ends up in an eigenvalue problem with four eigenvalues and modes. The analytical result is not distinguishable from the numerical result for the plane case gained by a finite element solution. Interestingly if the diffusion time for the temperature distribution is of the order of the traveling time of a pressure wave no adiabatic behavior is observed. A parameter map for the different regimes is given. Since only the behavior of the gas within the bubble is considered the analysis is independent of the surface tension coefficient and the inertia of the surrounding liquid. For the plane bubble since there is no curvature there is no pressure change over the free surface. Despite of this a plane bubble is manly academic, since due to inertia the pressure within the fluid would have to be infinity if the liquid volume around the bubble is unbounded.

58. Determination of the tensile strength and the nuclei spectrum by means of the in-situ-nozzle

N. Hamadeh	*Chair of Fluid Systems, University of Technology Darmstadt*
P. Pelz	*Chair of Fluid Systems, University of Technology Darmstadt*
B. Stoffel	*Chair of Fluid Systems, University of Technology Darmstadt*
G. Ludwig	*Chair of Fluid Systems, University of Technology Darmstadt*

In this paper a new method for the determination of the tensile strength in combination with the underlying distribution of free cavitation nuclei is presented. This method is based on the advancement of the in-situ-measuring of the tensile strength by means of the so-called “in-situ-nozzle” originally presented at the CAV2006 Symposium [11]. The in-situ-nozzle works as a cavitation susceptibility meter that is based on the venture principle [8]. In contrast to several previously presented cavitation susceptibility meters the in-situ-nozzle can be used to measure the tensile strength “in-situ” i.e. directly within a hydraulic pipe, without the need for any bypass connections. Consequently the in-situ-nozzle serves as a measuring device for the detection of the tensile strength at basically any position within a hydraulic circuit. The adjacent analysis method that has been developed to evaluate the collected data allows for providing additional information on the freely distributed cavitation nuclei in an operating test installation. For this purpose the measured tensile strength is associated with the critical size of gas bubbles acting as cavitation nuclei. Following this approach it is possible to give an approximation of the currently existing nuclei size distribution respectively the nuclei concentration at the mounting position of the in-itunozzle. By installing two identically constructed in-situ-nozzles in a centrifugal pump test rig the tensile strength and the nuclei concentration at two positions with different boundary conditions are determined. One nozzle is mounted directly upstream of the test pump, whereas the other one is installed in the pressure pipe downstream of the pump. This is done first of all, to examine the capability of the measuring device of being sensitive enough for a change in the boundary conditions in general. If this is verified, the two nozzles can be adopted to particularly investigate the influence of specifically chosen operating points of the test pump on the tensile strength as well as on the underlying nuclei concentration.

59. Transient hydroelastic analysis of surface-piercing propellers

Y. Young	*University of Michigan at Ann Arbor*
B. Savander	*Maritime Research Associates*

A coupled boundary element method – finite element method (BEM-FEM) is presented for the transient hydroelastic analysis of surface-piercing propellers (SPPs). The method is used to help the design and analysis of three different size SPPs that deliver a constant advance speed of 25.72 m/s (50 knots). Numerical validation studies are shown. The mean and unsteady responses of the three SPPs are presented. Finally, limitations of the BEM-FEM method are discussed.

60. Rate-dependent hydroelastic response of self-adaptive composite propellers in fully wetted and cavitating flows

Y. Young	*University of Michigan*
M. Motley	*Princeton University*

The objectives of this work are to investigate the fully wetted and cavitating performance of a self-adaptive composite propeller and its dependence on the propeller rotational frequency (RPM or revolution per minute) in addition to the advance coefficient and ambient pressure. Self-adaptive composite propellers are designed to take advantage of the intrinsic deformation coupling behavior of anisotropic composites to improve propeller performance via automatic, passive blade pitch adjustment in spatially or temporally varying flow. The design methodology, numerical and experimental studies of self-adaptive composite propellers in fully wetted flow can be found in [1-7]. In past studies, the primary focus was the fully wetted performance of the composite propellers operating at the design RPM. However, since the deformations of adaptive composite propellers depend on the hydrodynamic load, which in turn depends on the propeller RPM, the response of adaptive composite propellers depend on both the advance coefficient and RPM. Moreover, at high RPMs, composite propellers may be subject to resonant vibration failure due to the inherent flexibility needed to achieve the desired self-adaptive behavior, and due to the decrease in natural frequency caused by added mass effects. Hence, it is important to evaluate the rate-dependent behavior of self-adaptive composite propellers. It is also important to evaluate the cavitating performance of self-adaptive composite propellers since cavitation can lead to thrust breakdown, decrease in efficiency, as well as erosion and localized impact damage to the composite blades. In this work, a previously validated coupled boundary element method – finite element method (BEM-FEM) is used to analyze the rate-dependent response of self-adaptive composite propellers in fully wetted and cavitating flows. Implications of the rate-dependent behavior on the design and interpretation of experimental studies, particularly cavitation tunnel studies, are discussed.

61. Cavitation patterns on a plano-convex hydrofoil in a high-speed cryogenic cavitation tunnel

Y. Ito *Tokyo Institute of Technology*
T. Nagayama *Toyota Motor Corporation*
T. Nagasaki *Tokyo Institute of Technology*

Cavitation around a plano-convex hydrofoil has been observed using a cryogenic cavitation tunnel of a blowdown type. An approximately 300mm long test section with flow visualization was set between the 100L upper and lower tanks. The working fluids were water and liquid nitrogen. Experiments with emphasis on periodical shedding of cloud cavitation were performed for three channels, 20, 30 and 60 mm in width, and two hydrofoils, 20 and 60mm in chord length L_C. Inlet velocity uin and cavitation number σ were varied between 3.8 and 19.5 m/sec, and −1.83 and 19.35, respectively. Incident angle was fixed at 8º. Observed cavitation patterns are sorted according to the maximum cavitation length L_{max} compared to L_C. Type X is defined as no cavitation, type A as $0<L_{max}\leq L_C$, type B as $L_C<L_{max}\leq 2L_C$, and type C as $L_{max}>2L_C$.

Type B has either the periodical shedding mode or the steady mode, so type PB is defined as type B with periodical shedding and type SB as type B under steady condition. Apparently types A and C are almost steady. Type PB solely occurs in the case that L_{max} is comparable to L_C. The cavitation patterns are mapped in the diagram of the degree of subcooling $P_{in}-P_{sat}$ versus the dynamic pressure $\rho u_{in}^2/2$ because they are not organized only by $\sigma =(P_{in}-P_{sat})/(\rho u_{in}^2/2)$. It is suggested that the maximum cavitation thickness is controlled by the similarity laws of hydrofoils for types SB and C.

62. Numerical investigation of cloud cavitation and cavitation noise on a hydrofoil section

J. Seo *Stanford University*
S. Lele *Stanford University*

Partial cavitating flow and cloud cavitation on a hydrofoil section are numerically investigated. A fully compressible, density based homogeneous equilibrium model is employed along with a RANS turbulence model and high-order numerical methods based on a sixth-order central compact scheme and localized artificial diffusivity scheme are used to resolve the cavitating flow and pressure waves generated by cloud cavitation. Predicted results compare well with the experimental measurements for steady/unsteady partial cavitating flows on a NACA66 hydrofoil at cavitation number, σ=1.0-1.4 and angle of attack 6 and 8 degree. Detailed experimental data from the work of Leroux et al. were provided by Prof. J.-A. Astolfi at Institut de Recherche de l'Ecole Navale, France. Numerical visualizations of cloud cavity evolution and surface pressure signals show relatively good agreement with the experimental data. The re-entrant jet flow and pressure wave generated by collapse of cloud cavity are closely investigated. The mechanism associated with two different unsteady dynamics of cloud cavitation observed in previous numerical/experimental study for angle of attack 6 and 8 degree are further explored using the present computational results. The pressure pulse generated by the collapse of bubble cloud and the flow-blockage effect caused by a large cavity cloud are found to be responsible for the shifting of cloud cavitation dynamics.

63. Development and validation of new cryogenic cavitation model for rocket turbopump inducer

N. Tani *JAXA*
S. Tsuda *JAXA*
N. Yamanishi *JAXA*
Y. Yoshida *JAXA*

A rocket propellant is often used cryogenic fluid such as liquid hydrogen or liquid oxygen, and it is known that a suction performance of turbopump inducer (Fig.1) in cryogenic fluid is improved due to 'Thermodynamic effect'. Thermodynamic effect can be explained by temperature decrease inside a cavity region arising from the latent heat absorption. Although this effect is also to be observed in water cavitation, it becomes more accentuated in cryogenic fluid. In order to properly understand the cavitation performance of the inducer, this thermodynamic effect should not be overlooked.

In the present study, cryogenic cavitation model without energy equation was established. During the design phase of an inducer, wall clock time of CFD simulation becomes an important point, therefore, equation number should be small. The present model considers that temperature decrease due to latent heat absorption is analytically estimated. Validation calculations were carried out for blunt head form and hydrofoil for water cavitation and two-dimensional blunt body, Laval nozzle and inducer for cryogenic fluids.

64. Evaluation of incipient cavitation erosion for pipe wall at downstream of an orifice

Y. Nagaya *Institute of Nuclear Safety System, Incorporated*
M. Murase *Institute of Nuclear Safety System, Incorporated*
S. Mizuyama *The Kansai Electric Power co.inc*
S. Hattori *Department of Mechanical Engineering, University of Fukui*

Cavitation induced vibration and the consequent erosion of pipes are the potential damaging factors in the piping systems. In order to prevent such trouble, it is preferable to develop a detection method for cavitation occurrence. Especially, in power plants, it is necessary to detect cavitation from the outside of the piping during operation. In this paper, in order to evaluate incipience of cavitation erosion, we carried out cavitation erosion experiments using aluminium specimens and we measured impulsive force induced by cavitation bubbles collapse using impact force detectors. In the cavitation erosion experiments, the incipient cavitation numbers, where cavitation erosion pits occured, were 0.8 at 50mm and 75mm downstream from the orifice and 0.7 at 100mm downstream. At those cavitation numbers, the states of cavitation was in a developed state or nearly so. In the measurements of impulsive force, the cavitation number, where impulsive force began to increase, was almost with the same as cavitation numbers at the occurrence of erosion pits.

65. Thermo-fluid dynamic experiment of He II cavitating flow

M. Murakami *University of Tsukuba*
K. Harada *University of Tsukuba*

We have conducted a series of experiments of superfluid helium, He II, cavitating flows. For the purpose of comparison, normal fluid, He I, flows were also examined. In the experiments liquid helium flows driven by a bellows pump were investigated by flow visualization method and PIV, and through the measurements of the temperature depression and the pressure loss. The most noticeable features characterizing superfluid cavitating flows arise from superthermal conduction, the specific heat anomaly and quantized vortices. Due to the superthermal conduction phenomena, latent heat can be supplied to cavitation bubbles from bulk liquid almost without limit, and consequently the development of cavitation is considerably rapid and in large-scale in He II. The specific heat becomes extremely large across the lambda phase transition line from He I to He II, which is known as the specific heat anomaly. In the present study, the experimental results obtained so far were examined with respect to the thermodynamic effect in He II cavitating flows referring to the Brennen thermodynamic parameter Σ. It is seen that in most aspects the thermodynamic effect is negligible in cavitating flows of He II owing to the superthermal conduction. This means He II hardly behaves as a cryogenic fluid in cavitation phenomena. It is, on the other hand, found that the temperature depression is not negligible and there is apparent temperature dependence in its variation. Consequently, He II may, in some aspects, be expected to behave as a cryogenic fluid in cavitation because of extremely large void fraction.

66. Pressure-wave formation and collapses of cavitation clouds impinging on solid wall in a submerged water jet

K. Sato *Kanazawa Institute of Technology*
Y. Sugimoto *Kanazawa Institute of Technology*
S. Ohjimi *Kanazawa Institute of Technology*

A high-speed water jet ejected into water forms a cavitating water jet accompanied with cavitation clouds in a periodic manner. A powerful impulsive force can be caused at the collapse of unsteady cavitation clouds at the same time when the cavitating water jet impinges against a solid wall. It is known that this force can be widely used in an industrial field such as cleaning, cutting, and peening.

In the present experiment, cavitation clouds are observed to investigate the details such as impinging, dividing and collapsing behaviors using a constrained-type test section as well as an open-type test section. The constrained-type test section is used to quasi-two dimensionally observe the behavior of cavitation clouds in the near impinging wall region. The present purpose is to investigate about the behavior of cavitating water jet in the near impinging wall region as well as the relation of cavitation cloud collapse with the formation of pressure wave, the propagation of pressure wave and the cavitation impact.

In order to estimate the high speed phenomena such as rapid and consecutive collapses of cavitation clouds and pressure wave formation, the frame difference method for cavitating flow is used in the present image analysis for cavitation cloud. The usefulness of the method is experimentally verified for the behavior analysis of high speed liquid flow accompanied with growth and collapse of bubbly cloud.

As a result it is experimentally found that 1) the present image analysis method based on the frame difference method makes possible to grasp the motion of pressure wave propagation in cavitation cloud, 2) local cloud collapse causes a pressure wave which propagates toward the surrounding area and as a result causes secondary collapses in a chain-reaction manner, and 3) cavitation clouds on the impinging wall tend to be peripherally located in an annular zone at the final collapsing stage. The existence of the annular cloudy zone can be related to the ring-like cavitation erosion distribution and the chain-reaction-type propagation of cavitation clouds.

67. Numerical prediction of cavitation erosion in cavitating flow

N. Ochiai *Graduate School of Tohoku Univ.*
Y. Iga *Institute of Fluid Science, Tohoku Univ.*
M. Nohmi *EBARA Research Co. Ltd.*
T. Ikohagi *Institute of Fluid Science, Tohoku Univ.*

In this study bubble behavior in cavitating flow is analyzed and prediction of cavitation erosion in 2D cavitating flow around ClarkY 11.7 % hydrofoil at several cavitation is performed by impact pressure induced by bubble collapse. Our numerical method predicts that the impact energy is small if variation of cavitating flow is small and that the position of peak impact energy moves downstream with the decrease in cavitation number until the maximum sheet cavity length becomes larger than chord length. When the maximum sheet cavity length becomes larger than chord length, there are not obvious peak values and relatively weak erosion occurs. And it is found that high impact pressures are mainly induced by bubbles in a cloud and in the vicinity of sheet cavity termination during a cloud collapse. Therefore large impact energy occurs when the cloud cavity collapses near the hydrofoil, the sheet cavity termination is on the hydrofoil and the number of bubble is large in these cavities.

68. Study on unsteady cavitating flow simulation around marine propeller using a RANS CFD code.

K. Kimura *Akishima Laboratories (Mitsui Zosen) Inc.*
T. Kawamura *The University of Tokyo*
Z. Huang *The University of Tokyo*
A. Fujii *Mitsui Engineering & Shipbuilding Co., Ltd.*
T. Taketani *Akishima Laboratories (Mitsui Zosen) Inc.*

The authors have been investigating the possibility of the application of CFD to the propeller performance evaluation and optimization. In these previous papers [1,2], the authors presented CFD simulation of non-cavitating and cavitating flow around a marine propeller using a commercial CFD code. A good agreement with the experiment was confirmed for the non-cavitating flow. Various validations were also carried out for the cavitating flow, and the followings were revealed. First, we confirmed that the cavity shape in a uniform flow was qualitatively well estimated, but the difference between two propellers, of which the blade sections were somewhat different, were not reproduced. Secondary, the cavity pattern in the non-uniform flow was also qualitatively well estimated, but the resulting pressure fluctuation was not validated. In this paper, the systematic experiment was carried out using two propellers, whose dimensions were very similar, to study the above issues, and simulation was carried out for the same cases. In the uniform cavitating flow simulation, the discrepancy of cavity shape around two propellers, whose dimension were very similar, was reproduced, and the quantitative validation of the fluid force such as thrust was done. In the non-uniform cavitating flow simulation, the comparison of the cavity pattern with the experiment and the quantitative validation of the fluctuating pressure on the wall of the cavitation tunnel were done.

70. Rudder Gap Flow Control for Cavitation Suppression

J. Oh	*Inha University*
H. Lee	*Seoul National University*
K. Shin	*Seoul National University*
C. Lee	*Seoul National University*
S. Rhee	*Seoul National University*
J. Suh	*Seoul National University*
H. Kim	*Inha University*

For the suppression of rudder cavitation, especially within and around the gap between the stationary and movable parts, flow control devices were developed. In the present study, both experimental and computational analyses of the flow control devices were carried out. The new rudder system is equipped with cam devices, which effectively close the gap between the stationary horn/pintle and movable flaps. Model scale experiments of surface pressure measurements, flow field visualization near the gap using PIV, and cavitation behavior observation were conducted in a cavitation tunnel. The experiments were simulated using a computational fluid dynamics tool and the results are compared for validation. It is confirmed that the flow control devices effectively suppresses the rudder gap cavitation and, at the same time, augments lift

72. High speed motion in water with supercavitation for sub-, trans-, supersonic Mach Numbers

V. Serebryakov	*Institute of Hydromechanics of NASU, Kiev, Ukraine*
I. Kirschner	*Alion Science and Technology Corporation, RI, U.S.*
G . Schnerr	*Technical University of Munchen, Germany*

The results of research for supercavitating motion in water at very high speeds – comparable with sonic speed ~1500m/s – are presented. At such speeds the water is a compressible fluid and the basic compressible hydrodynamics of supercavitating flows together with practical approaches and experimental data are considered. The theory of ballistic projectiles motion is developed with emphasis on the problems of maximal range, lateral motion prediction and problems of minimal declination, hydro-elastic effects, and resonant oscillation frequencies. One main purpose of the article is an attempt to advance the level of understanding of the problem of very high-speed underwater launch by a comprehensive review of previous research on this topic.

73. Large eddy simulation of cavitation inception in a high speed flow over an open cavity

E. Shams	*Oregon State University*
S. Apte	*Oregon State University*

Large-eddy simulation of flow over an open cavity corresponding to the experimental setup of Liu and Katz [1] is performed. The flow Reynolds number based on the cavity length and the free stream velocity is 170;000. The filtered, incompressible Navier-Stokes equations are solved using a co-located grid finite-volume solver with the dynamic Smagorinsky model for subgrid scale closure. The computational grid consists of around five million grid points with two million points clustered around the shear layer and the wall-layer over the leading edge is resolved. The only input from the experimental data is the mean velocity profile at the inlet condition. The mean flow is superimposed with turbulent velocity fluctuations generated by solving a forced periodic duct flow at free-stream Reynolds number. The flow statistics, including mean and rms velocity fields and pressure coefficients, are compared with the experimental data to show reasonable agreement. Cavitation inception is investigated using two approaches: (i) a discrete bubble model wherein the bubble dynamics is computed by solving the Rayleigh-Plesset and the bubble motion equations using an adaptive time-stepping procedure, and (ii) a scalar transport model for the liquid volume fraction with source and sink terms for phase change. The cavitation inception occurs near the trailing edge similar to that observed in the experiments. A periodic growth and decay of bubble size and liquid vapor fraction is observed above the trailing edge owing to local variations in pressure minima. The dynamic interactions between traveling vortices in the shear layer and the trailing edge affect the value and location of the pressure minima.

74. A hybrid lagrangian-eulerian approach for simulation of bubble dynamics

S. Apte	*Oregon State University*
E. Shams	*Oregon State University*
J. Finn	*Oregon State University*

A mutiscale numerical approach is developed for the investigation of bubbly flows in turbulent environments. This consists of two different numerical approaches capable of capturing the bubble dynamics at different scales depending upon the relative size of the bubbles compared to the grid resolution: (i) fully resolved simulations (FRS) wherein the bubble dynamics and deformation are completely resolved, and (ii) subgrid, discrete bubble model where the bubbles are not resolved by the computational grid. For fully resolved simulations, a novel approach combining a particle-based, mesh-free technique with a finite-volume flow solver, is developed. The approach uses marker points around the interface and advects the signed distance to the interface in a Lagrangian frame. Interpolation kernel based derivative calculations typical of particle methods are used to extract the interface normal and curvature from unordered marker points. Unlike front-tracking methods, connectivity between the marker points is not necessary. For underresolved bubbles, a mixture-theory based Eulerian-Lagrangian approach accounting for volumetric displacements due to bubble motion and size variations is developed.

The bubble dynamics is modeled by Rayleigh-Plesset equations using an adaptive timestepping scheme. A detailed verification and validation study of both approaches is performed to test the accuracy of the method on a variety of single and multiple bubble problems to show good predictive capability. Interaction of bubbles with a traveling vortex tube is simulated and compared with experimental data of Sridhar and Katz [1] to show good agreement.

77. Surrogate-based modeling of cryogenic turbulent cavitating flows

C. Tseng University of Michigan
W. Shyy University of Michigan

The cryogenic cavitation has critical implications on the performance and safety of liquid rocket engines. In this study, a systematic investigation based on the surrogate modeling techniques is conducted to assess and improve the performance of a transport-based cryogenic cavitation model. Based on the surrogate model, global sensitivity analysis is be conducted to assess the role of model parameters regulating the condensation and evaporation rates, and uncertainties in material properties, specifically, the vapor density and latent heat. The surrogate models considered include the response surface, radial basis neural network, Kriging, and a weighted average composite model combining all surrogates. It is revealed that the vapor density and the model parameter controlling the evaporation rate are more critical than latent heat and the model parameter controlling the condensation rate. Based on the recommended model parameter values, better prediction of the cryogenic turbulent cavitation can be attained.

78. Numerical investigation of thermodynamic effect on unsteady cavitation in cascade

Y. Iga Institute of Fluid Science, Tohoku Univ.
N. Ochiai Institute of Fluid Science, Tohoku Univ.
Y. Yoshida Japan Aerospace Exploration Agency (JAXA)
T. Ikohagi Institute of Fluid Science, Tohoku Univ

In the present study, the thermodynamic effect on unsteady cavitation is investigated in cascade in water and liquid nitrogen at different freestream temperatures. Cavitation flowfield is simulated based on self-developed locally homogeneous model of a compressible gas-liquid two-phase medium, which is available to treat unsteady cavitation. For calculation of thermodynamic effect on cavitation, simplified thermodynamic model for the locally homogenous medium is used where local saturated vapor pressure changes with depending on rate of heat transfer by evaporation and condensation. In the result of the numerical analysis, the difference of cavitating flowfields is reproduced numerically in water and liquid nitrogen at different freestream temperature concerning the cavity surface profile and distribution of the evaporation region. Also the thermodynamic effect on cavitation in water and liquid nitrogen is investigated by comparing the cavity volumes. Then, well known thermodynamic effect on cryogenic cavitation can be reproduced numerically in liquid nitrogen, where development of the cavity is suppressed according to increase of freestream temperature. On the other hand, the inverse thermodynamic effect, which is experimentally observed in single hydrofoil in water, is reproduced under the condition of unsteady cavitation in water in the present study.

79. A panel method for trans-cavitating marine propellers

S. Gaggero Univ. of genova
S. Brizzolara Univ. of genova

The extension to super-cavitating propellers of the numerical panel method developed by the Marine CFD Group of the University of Genova is presented and largely validated in the paper. The validation of the theoretical model for the cavity detachment and closure in the wake of the blade profiles is presented first on a typical super-cavitating profile for which theoretical and experimental solutions are known. Then the 3D panel method is applied on the complete series of Newton- Rader trans-cavitating propeller for which experimental measurements in cavitation tunnel and numerical results obtained by other researchers have been recently published. The main dynamic characteristics such as thrust and torque coefficients versus the advance ratio and the cavitation index, but also the cavitation patterns, in terms of bubble length at various radii, bubble volume and extension on the trailing vortex wake are presented and discussed for various propellers of the N-R series, having different pitch and expanded area ratios. Good correlations are in general achieved for what regards not only cavitation patterns, but also thrust and torque breakdown consequent to the cavity inception and growth on propeller blades.

87. Assessment of a central difference finite volume scheme for modeling of cavitating flows using preconditioned multiphase euler equations

K. Hejranfar Department of Aerospace Engineering, Sharif University of Technology
K. Hesary Department of Aerospace Engineering, Sharif University of Technology

A numerical treatment for the prediction of cavitating flows is presented and assessed. The algorithm uses the preconditioned, homogenous, multiphase Euler equations with appropriate mass transfer terms. A cell-centered finite-volume scheme employing the suitable dissipation terms to account for density jumps across the cavity interface is shown to yield an effective method for solving the multiphase Euler equations. The Euler equations are utilized herein for the cavitation modeling, because some certain characteristics of cavitating flows can be obtained using the solution of this system with relative low computational effort. In addition, the Euler equations are appropriate for the assessment of the numerical method used, because of the sensitivity of the solution to the numerical instabilities. For this reason, a sensitivity study is conducted to evaluate the effects of various parameters such as numerical dissipation coefficients and grid size on the accuracy and performance of the solution. The computations are presented for cavitating flows around the NACA0012 and NACA66(MOD) hydrofoils and also an axisymmetric hemispherical fore-body for different conditions and the results are compared with the available numerical and experimental data. The solution procedure presented is shown to be accurate and efficient for predicting different types of cavitating flows over 2D/axisymmetric geometries.

89. Investigation of turbulent modulation by cavitation for subgrid-scale modeling in LES

K. Okabayashi *Osaka University Graduate School*
T. Kajishima *Osaka University*

The two-way interaction between cavitation and turbulence was nvestigated by the direct numerical simulation of a spatially-developing mixing layer. Namely, the vortical structure and Reynolds stress components were compared between cavitating and non-cavitating conditions. Under cavitating condition, cavitation mainly occur in the regions of low pressure which are corresponding to vortices. Under cavitating condition, instability of mixing layer is caused more easily due to disturbance by cavitation to the flow field. Therefore, cavitation excites the instability of shear layer. vortices generating and pairing. As a result of stimulated pairing, the pitch of roll-cell vortices become longer than that in non-cavitating condition. One of the circumferential components of roll-cell vortices is suppressed by the decreasing of the number of roll-cell vortices. Circumferential component of streamwise vortices, on the other hand, tends to increase in comparison with non-cavitating condition. This is explained by volume fluctuation by cavitation. The modulation of Reynolds stress is consistently described by these changes in vortical stractures.

90. Generality of rotating partial cavitation in two-dimensional cascades

B. An *Osaka University Graduate School*
T. Kajishima *Osaka University*
K. Okabayashi *Osaka University Graduate School*

Numerical simulations of 2-dimensional (2D) unsteady cavitating flows were carried out under various conditions of the number of blades, incidence angles and cavitation numbers. When the incidence angle increased or the cavitation number decreased, the steady balanced cavitation transited to unsteady and non-uniform patterns. Typical patterns reported in the previous studies such as rotating, asymmetric and alternating for 3- and 4-blades were successfully reproduced. In this study, cascades of the larger number of blades were dealt with to consider the generality of unsteadiness by reducing the influence of periodicity. The cavitation is basically triggered in the backward next section. However, the period of time for growing causes complexity in the discrimination of propagation. In most cases of rotating partial cavitation, except for 4-blades, the cavity develops in the second passage of backward direction after the decay of largest cavity. In case of many blades, multiple cavities rotate simultaneously and the particular patterns observed in cascades of small even numbers of blades
attenuate.

91. Cavitation in a bulb turbine

J. Necker *Voith Hydro Holding*
T. Aschenbrenner *Voith Hydro Holding*
W. Moser *Voith Hydro Holding*

The flow in a horizontal shaft bulb turbine is calculated as a two-phase flow with a commercial Computational Fluid Dynamics (CFD-)-code including cavitation model. The results are compared with experimental results achieved at a closed loop test rig for model turbines. On the model test rig, for a certain operating point (i.e.volume flow, net head, blade angle, guide vane opening) the pressure behind the turbine is lowered (i.e. the Thomacoefficient s is lowered) and the efficiency of the turbine is recorded. The measured values can be depicted in a so-called s−break curve or h-s−diagram. Usually, the efficiency is independent of the Thoma-coefficient up to a certain value. When lowering the Thoma-coefficient below this value the efficiency will drop rapidly. Visual observations of the different cavitation conditions complete the experiment. In analogy, several calculations are done for different Thoma-coefficients s and the corresponding hydraulic losses of the runner are evaluated quantitatively. Besides, the fraction of water vapour as an indication of the size of the cavitation cavity is analyzed qualitatively.
The experimentally and the numerically obtained results are compared and show a good agreement. Especially the drop in efficiency can be calculated with satisfying accuracy. This drop in efficiency is of high practical importance since it is one criterion to determine the admissible cavitation in a bulbturbine. The visual impression of the cavitation in the CFDanalysis is well in accordance with the observed cavitation bubbles recorded on sketches and/or photographs.

92. Cavitation and flow instabilities in a 3-bladed axial inducer designed by means of a reduced order analytical model

A. Cervone *Alta S.p.A.*
L. Torre *Alta S.p.A.*
A. Pasini *University of Pisa*
L. D'agostino *University of Pisa*

The present paper illustrates the main results of an experimental campaign conducted using the CPRTF (Cavitating Pump Rotordynamic Test Facility) at Alta S.p.A. The tests were carried out on the DAPAMITO inducer, a three-bladed axial pump designed and manufactured by Alta S.p.A. using a simplified analytical model for the prediction of geometry and noncavitating performance of typical space rocket inducers. The transparent inlet section of the facility was instrumented with several piezoelectric pressure transducers located at three axial stations: inducer inlet, outlet and the middle of the axial chord of the blades. At each axial station at least two transducers were mounted with given angular spacing in order to cross-correlate their signals for amplitude, phase and coherence analysis. However, probably because of the high value of the blade tip clearance, very few flow instabilities have been detected on the inducer, including: steady asymmetric cavitation caused by the different extension of the cavitating regions on the blades; cavitation surge at a frequency equal to 0.16 times the inducer rotational frequency; a higher-order axial phenomenon at 7.2 times the rotational frequency.

94. Prediction of cavitating flow around 3-D straight/swept hydrofoils

S. Singh	*The University of Texas at Austin*
S. Kinnas	*The University of Texas at Austin*

A boundary element method (BEM) model is applied for prediction of cavitating flow around 3-D straight/swept hydrofoils between slip (zero shear) walls. The governing equation and boundary conditions are formulated and solved by assuming piecewise constant distribution of sources and dipoles on the hydrofoil and cavity surfaces, and piecewise constant distribution of dipoles on the trailing wake sheet. Cavity shape determination is initiated with a guessed cavity planform, and the cavity extent and thickness are determined iteratively until the dynamic and kinematic boundary conditions are satisfied on the cavity surface. To account for no-normal flow through the side walls, the method of images is used. For the fully-wetted case, the attached flow results obtained are compared with results from a full-fledged Reynolds-Averaged Navier-Stokes (RANS) solver. The cavitating results for a straight wing between slip walls are compared with results from an existing 2-D BEM solver for cavitating flow around hydrofoils. The RANS solver is also used to study separated flow characteristics around 2-D/3-D hydrofoils at high loading.

97. A boundary element method for the strongly nonlinear analysis of surface-piercing hydrofoils

V. Vinayan	*The University of Texas at Austin*
S. Kinnas	*The University of Texas at Austin*

A two-dimensional BEM based scheme is presented for the numerical modeling of the ventilated flow past a surfacepiercing hydrofoil. Fully nonlinear boundary conditions are applied on the free-surface allowing for the modeling of the jets generated as a result of the passage of the hydrofoil through the air-water interface. The scheme is validated through a comparison with self-similar solutions in the case of non-ventilating entry and with experiments in the case of ventilating entry. Results are presented for fully wetted and ventilating cases with and without the effects of gravity. Preliminary results are presented for the case of a hydrofoil in rotational motion, simulating the ventilation characteristics of a typical surface-piercing propeller. The fully non-linear scheme is a step towards quantifying the errors associated with some of the linearizing assumptions made in a 3-D boundary-element tool (PROPCAV) for the modeling of surface-piercing propellers..

98. Removal of an obstruction from a tube by a collapsing bubble

S. Ohl	*Institute of High Performance Computing*
D. Pavard	*ENSEEIHT*
E. Klaseboer	*Institute of High Performance Computing*
B. Khoo	*National University of Singapore*

The use of a collapsing bubble to clear an obstruction (in the form of a steel ball) near a tube, submerged in water, is studied with high speed photography. Tubes in horizontal and vertical configurations are studied. The bubble is generated via an electric spark discharge. The flow in the tubes resulting from the expansion of the bubble, or the high speed jet from the collapsing bubble pushes the ball away from the tubes and therefore clears the obstructions. In a case where air-backed tube is used, the bubble jets away from the tube. The resulting water plum at the hole (water-air interface) removes the blockage. The speed of the ball can be as high as 1 m/s shortly after the collapse of the bubble. Further studies are required to translate the phenomena observed to clinical applications such as the removal of blood clots in vessels or the clearing of blocked transplanted tubes..

99. Multiphase flow analysis of cylinder using a new cavitation model

W. Park	*Pusan National University*
C. Ha	*Pusan National University*
C. Merkle	*Purdue University*

Cavitating flow simulation is of practical importance for many engineering systems, such as marine propellers, pump impellers, nozzles, injectors, torpedoes, etc. The present work is to test a new cavitation model. The governing equation is the Navier-Stokes equation based on an homogeneous mixture model. The solver employs an implicit preconditioning algorithm in curvilinear coordinates. The computations have been carried out for the cylinders with 0-, 1/2- and 1-caliber forebody and then compared with experiments and other numerical results. Fairly good agreement with experiments and numerical results has been achieved.

100. Shallow angle water entry of ballistic projectiles

T. Truscott	*Massachusetts Institute of Technology*
A. Techet	*Massachusetts Institute of Technology*
D. Beal	*Naval Underwater Warfare Center*

The water-entry of ballistic projectiles is investigated using high-speed digital imaging to capture the subsurface cavity dynamics. Specially designed 0.22 caliber projectiles are fired into water at shallow angles to the free surface (5Âº to 15Âº) at Mach numbers between 0.3 and 1.0. Redesigned projectile tip geometries allowed projectiles to successfully enter the water and travel large distances underwater, due to the subsurface vapor-cavity that forms after impact, dramatically decreasing drag on the projectile. Projectile dynamics, critical entry angle and cavity formation are discussed for various bullet geometries, and results show that successful water-entry is a function of tip shape and length-to-diameter ratio. The data conclusively show that bullets with lower length-to-diameter ratios tumble inside the vapor cavity, while higher length-to-diameter ratios can lean against the cavity walls inducing a planing force pushing them back inside the cavity and mitigating the tumbling behavior. Experimental cavity observations of vapor-cavity formation is compared to a modified version of Logvinovich's [1] theoretical model, which includes an updated formulation of the model and an angle of attack correction. Despite the unsteady nature of this problem, this improved steady state model fits well with experimental data and serves as an accurate design tool for naval engineers.

102. A multi-scale study on the bubble dynamics of cryogenic cavitation

SS. Tsuda	*Japan Aerospace Exploration Agency*
S. Takeuchi	*The University of Tokyo*
Y. Matsumoto	*The University of Tokyo*
M. Koshi	*The University of Tokyo*
N. Yamanishi	*Japan Aerospace Exploration Agency*

This study aims to construct a multi-scale cavitation model for unsteady cryogenic cavitation CFD. Many elementary physical processes of bubbles (i.e, nucleation, growth/shrink, evaporation/condensation, coalescence/fission, collapse, bubble-bubble interaction, bubble-turbulence interaction, and so on) emerge in cryogenic cavitation where some of the processes have not been understood well. In this paper, we mainly focused the molecular processes in homogeneous liquid-vapor nucleation with non-condensable gas solution by using Molecular Dynamics (MD) method. Bubble nucleation in liquid oxygen including helium, nitrogen, or argon was simulated. Molecular interaction was given by Lennard-Jones potential, and basically, each potential parameter was defined so that a saturation curve obtained by MD data was consistent with an experimental value. In the case that helium was impurity, a bubble nucleus was formed by density fluctuation at a lower concentration while a cluster constituted with helium molecules formed a bubble nucleus at a higher concentration, and the nucleation point becomes closer to the saturation point of pure oxygen when helium molecules form clusters. On the other hand, in the case that nitrogen or argon was the impurity, the above-mentioned clustering was not observed clearly at a concentration where helium made clusters, and these impurities have weaker action to make clusters and cavitation bubble nuclei compared with helium.

103. Cavitation as a Microfluidic Tool

C. Ohl	*Nanyang Technological University Singapore*
P. Quinto-Su	*Nanyang Technological University Singapore*
R. Dijkink	*University of Twente, Enschede, The Netherlands*
R. Gonzalez	*Nanyang Technological University Singapore*
F. Prabowo	*Nanyang Technological University Singapore*
X. Huang	*Nanyang Technological University Singapore*
T. Wu	*Nanyang Technological University Singapore*
V. Venugopalan	*University of California, Irvine, CA,*

Cavitation in confined geometries in particular in narrow gaps prevalent in microfluidic geometries allows for novel applications. Here we will give an overview of successful demonstrations of cavitation as a microfluidic tool. Cavitation can pump and mix liquids very rapidly, move objects such as cells, rupture plasma membranes, probe elastic properties in micro-rheology, study coalescence, and even create arbitrary superpositions of shock waves. In all areas, bubbles are created with a focused laser which allows precise temporal and spatial control. With the usage of digital holography arbitrary configurations of bubbles can be created such as bubble clusters, squarish, toroidal, or even linear cavitation bubbles.

Interestingly, even in very narrow gaps of a few tens of microns most of the bubble dynamics can be described with potential flow. This presentation will summarize published work and show current research under progress.

104. Bubble shock wave interaction near biomaterials

S. Ohl	*Institute of High Performance Computing*
E. Klaserboer	*Institute of High Performance Computing*
B. Khoo	*National University of Singapore*

The interaction of bubbles, both oscillating and stationary near bio-materials is of interest for the development of various medical treatment involving ultrasound and shock waves. This is because cavitation bubbles often nucleate in the bodily fluid under pressure waves, and their dynamics directly influence the success of the treatment and the collateral damages sustained. For example, in the treatment of Extracorporeal Shock Wave Lithotripsy (ESWL), cavitation bubbles are created when the shock wave is administered. These bubbles oscillate and collapse near the kidney stones and the body tissues. They are responsible both for the breaking up of the stones as well as the collateral damages to the nearby tissues. We study the interaction of an oscillating bubble near various bio-materials. The bio-materials are modeled as elastic fluids with similar physical properties such as elastic modulus, Poisson ratio, and density. The bubble dynamics are summarized based on biomaterial physical properties. We also study the interaction of a stationary bubble with the nearby bio-materials when hit by a lithotripter shock wave. High speed jets and splitting of bubbles are observed due to the influence of the nearby biomaterials.

107. A modified SST k-ω turbulence model to predict the steady and unsteady sheet cavitation on 2D and 3D hydrofoils

D. Li	*SSPA Sweden AB, Göteborg, Sweden*
M. Grekula	*SSPA Sweden AB, Göteborg, Sweden*
P. Lindell	*SSPA Sweden AB, Göteborg, Sweden*

The paper presents a study of using a modified SST (Shear-Stress Transport) k-ω model with a multi-phase mixture flow RANS solver to predict the steady and unsteady cavitating flows around 2D and 3D hydrofoils. Based on Reboud et al [6]'s idea of modifying turbulent viscosity for a RNG k-▯ model, a modification is applied to a SST k-ω model in the present work. The cavitation is modeled by Schnerr-Sauer's cavitation model [16]. First, results of 2D NACA0015 foil at two cavitation numbers, ω =1.6 (stable sheet cavitation) and ω=1.0 (unsteady with shedding) are compared for different grids and with available experiment data. Then, the problem of the standard SST model in predicting unsteady cavitation is discussed. Finally the results for a 3D twisted hydrofoil are compared with the experiment by Foeth and Terwisga [3]. It is found that with the modified SST k-ω model the RANS solver is able to predict the essential features like development of re-entrant jets, the pinch-off, the shedding of vortex and cloud cavities for the 2D NACA0015 foil at ω =1.0. For the case at ω =1.6, the model predicts a high frequency fluctuating sheet cavity with minor shedding at its closure. Compared with the standard SST model, the global quantities like lift, drag, and shedding frequency predicted by the modified model are closer to the experimental data, although considerable discrepancy with the experiment data is noted for the unsteady case at ω =1.0. In addition, a special type of secondary cavities, developed downstream an upstream-moving collapse cavity and termed as “vortex group cavitation” by Bark et al [1], appears to be observable in the simulation at this condition. The existence of this type of cavity has been reconfirmed in a recent experiment in the SSPA’s cavitation tunnel..

109. Microbubble disruption by ultrasound and induced cavitation phenomena

Y. Tomita	*Hokkaido University of Education, Hakodate*
R. Uchikoshi	*Graduate School, Hokkaido University of Education*
T. Inaba	*Graduate School, Hokkaido University of Education*
T. Kodama	*Graduate School of Biomedical Engineering, Tohoku University*

Aiming at the enhancement of in vitro sonoporation effect to cells in drug delivery, cavitation phenomena ultrasonically generated in a cylindrical vessel, which is one of a 24 wellplate, were observed with long-term photography by using a digital video camera synchronized with a repeatable flash light with the duration of 1 μs as well as by taking a number of snapshots. A suspension consisting of distilled water and microbubbles (Sonazoid ultrasound contrast agent containing C4F10 gas) was used as a test liquid in the present experiment. It was found that microbubbles were rapidly destroyed after the ultrasound irradiation and their survival numbers were reduced by half at the exposure time of 100 ms. A maximum number of cavitation bubbles, sometimes more than sixty cavitation bubbles detected in the observation volume, were generated at a certain exposure time less than one second. Cavitation bubbles were mainly observed near the second products consisting of the fragments of the shell material and the flowing gas out of the interior of individual microbubbles. By exploring the free surface oscillation of a Sonazoid suspension, we found out that the (1,1) mode with the frequency of about 5 Hz was prominent. This seems important because the flow induced by this surface oscillation as well as by the acoustic radiation pressure can provide a higher possibility to convey the second products everywhere in the liquid. Consequently cavitation bubbles can be generated at a relatively long time after the ultrasound irradiation even though the majority of microbubbles are destroyed during an earlier period. In fact plenty of cavitation bubbles were occasionally generated even at the ultrasound exposure time of five seconds.

110. A dynamic test platform for evaluating control algorithms for a supercavitating vehicle

A. Hjartarson	*University of Minnesota*
H. Mokhtarzadeh	*University of Minnesota*
E. Kawakami	*University of Minnesota*
G. Balas	*University of Minnesota*
R. Arndt	*University of Minnesota*

The use of supercavitation to enable marine vehicles to travel at extraordinary speeds is a topic of considerable interest. The control of these vehicles poses new challenges not faced with fully wetted vehicles due to a complex interaction between the vehicle and the cavity that it rides in. Some of the existing models make assumptions that may not be valid for a maneuverable vehicle. Furthermore, since there are various models being suggested for planing forces as well as different ways of obtaining fin and cavitator forces, there is a lack of unity among the equations used to calculate the hydrodynamic forces imparted on such a vehicle. Experimental test platforms have been developed at St. Anthony Falls Laboratory to enable testing and validation of control algorithms and hydrodynamic models. Previous efforts have revealed the destabilization of marginal supercavities by control surfaces, especially when a cavity is being maintained with ventilation [1]. Our latest water tunnel test platform is a body of revolution with an actuated cavitator on the model forebody, actuated fins that protrude through the cavity surface, and variable pitch of the model body, all supported by a six-axis force balance. In this paper we will present a brief description of the forces present in our mathematical model of a supercavitating vehicle, and then present the new experimental test platform that will be used to validate, and expand on this model.

111. Investigation of the behavior of ventilated supercavities

E. Kawakami	*University of Minnesota*
M. Williams	*UC Berkeley*
R. Arndt	*University of Minnesota*

The topic of supercavitation is of considerable interest to drag reduction and/or speed augmentation in marine vehicles. Supercavitating vehicles need to be supplied with an artificial cavity through ventilation until they accelerate to conditions at which a natural supercavity can be sustained. A study has been carried out in the high-speed water tunnel at St. Anthony Falls Laboratory to investigate some aspects of the flow physics of such a supercavitating vehicle. During the present experimental work, the ventilated supercavity formed behind a sharp-edged disk was investigated utilizing several different configurations. Results regarding cavity shape, cavity closure and ventilation requirements versus cavitation number and Froude number are presented. Additionally, effects related to flow choking in a water tunnel test section are discussed. Results obtained are similar in character to previously reported results, but differ significantly in measured values. Cavity shape, particularly aft of the maximum cavity diameter, is found to be a strong function of the model support scheme chosen.

112. Effects of surface characteristics on hydrofoil cavitation

M. Williams	*UC Berkeley*
E. Kawakami	*University of Minnesota*
E. Amromin	*Mechmath, LLC*
R. Arndt	*University of Minnesota*

This was an exploratory research project aimed at capitalizing on our recent research experience with unsteady partially cavitating flows. Earlier work identified the significant and unexpected effect of surface properties and water quality on the dynamics of these flows. The aim of this study was to explore the possibility of using hydrophobic surfaces to control or minimize unwanted vibration and unstable operation in the partially cavitating regime. A candidate shape, denoted as the Cav2003 hydrofoil, was selected on the basis of theoretical analysis for a given range of contact angle. We manufactured three hydrofoils of identical cross section, but different surface characteristics. Three different surfaces were studied: anodized aluminium (hydrophilic), Teflon (hydrophobic), and highly polished stainless steel (hydrophobic). Contact angle was measured with a photographic technique developed by three of the undergraduates working on the project. Studies were made in both weak and strong water. Significant surface effects were found, but were unexpected in the sense that they did not correlate with measured contact angles.

113. Blade load dynamics in cavitating and two phase flows

M. Kjeldsen *Norwegian University of Science and Technology*
R. Arndt *University of Minnesota*

A comparative study of lift dynamics on a hydrofoil and inlet pressure dynamics on a pump impeller vane is described in this paper. The hydrofoil, a rectangular planform NACA 0015 with a chord length of c=0.081m, fitted with a special arrangement that allowed the injection of gas downstream of the minimum pressure point, was tested in the St Anthony Falls Laboratory (SAFL) closed loop water tunnel at the University of Minnesota. The SAFL water tunnel is specially suited for gas injection type measurements due to high gas removal capabilities. The tests on the hydrofoil also included a full range of cavitation experiments. The pump tests were made at the Waterpower Laboratory at the Norwegian University of Science and Technology (NTNU). Upstream of the pump inlet a special bubble injection device was located. This arrangement allows a controlled amount of gas to enter the flow. The water and gas flow rates were measured separately. Lift measurements from the hydrofoil study display a striking similarity between gas loaded and cavitation lift dynamics. The pump dynamics data show a maximum for a moderate gas void fraction. It is also observed that a more pronounced low frequency dynamics is present for the gas-loaded systems.

114. Blade section design of marine propellers with maximum inception speed

Z. Zeng *China Ship Scientific Research Center*
G. Kuiper *Consultant*

Kuiper and Jessup (1993)) developed a design method for propellers in a wake. This method is based on the use of the Eppler foil design method. The optimized section is transformed into the three-dimensional propeller flow using the approach of the effective blade sections. Effective blade sections are two-dimensional sections in two-dimensional flow which have the same chordwise loading distribution as the three-dimensional blade sections of a propeller. However, the design procedure is laborious in two aspects: finding an optimum blade section using the Eppler program requires much skill of the designer, and transforming the two-dimensional blade section into a propeller blade section in three-dimensional flow is complex. In this paper, these two problems are dealt with. A blade section design procedure is presented using an optimization technique and an alternative procedure for the effective blade section is developed using a lifting surface design method. To validate the method a benchmark model of a naval ship was used. This benchmark model was extended by new appendices and a reference propeller, designed using conventional design methods. This reference propeller was optimized using the new design procedure and model tests were carried out Special attention was given to the data of the model and the reference propeller, to make the configuration suitable for RANS calculations.

116. Numerical modeling of cavity flow on bottom of a stepped planing hull

M. Makasyeyev *National Technical University of Ukraine "Kyiv Polytechnic Institute"*

To reduce the friction drag of the hulls of high-speed craft, seaplanes, and hydrofoil boats, steps are made on their undersurfaces. The effect is achieved due to the formation of a gas cavity aft of the step, which reduces the wetted area. The wetted area can also be reduced by striking a compromise between the cavity size and the wetted lengths of the planning surfaces by changing the center-of-mass position of the planning boat and the geometry of the stepped hull. The proper choice of the shape of the stepped bottom and the design parameters may also offer other useful effects, for example, the effect of surface wave energy regeneration on the system of planing surfaces. This paper presents a method of solution of the twodimensional mathematical problem on the cavity planing of the stepped hull of a high-speed craft, which allows one to predict and quantify the above-mentioned effects. The key feature of the proposed approach is that the problem is solved in natural physical formulation. All the required characteristics – the cavity shape and length, the free boundary shape, the wetted lengths of the planing surfaces, and the trim angles are determined from a specified cavitation number, Froude number, and center-of-mass position.

The Froude number is determined from the displacement. When, in addition, the center-of-mass position is specified, additional unknowns appear in the problem – the wetted lengths and the trim angles. In this case, to the singular integral equation of planing must be added the force and the moment balance equations. As a result, a logically closed system of integral equations is obtained. However, the system features parametric nonlinearity in the form of the unknown limits of integration – the cavity length and the wetted lengths. The nonlinear problem is solved by sequential minimization of the residual of the system using nonlinear-programming techniques.

The calculations have shown that the wave amplitude in the wake of the planing boat depends of the cavitation number and the design factors of the step and the planing boat. At negative cavitation numbers, an additional lift develops due to artificial air injection into the cavity under the bottom, which changes the draft and the trim angles. This lowers the wave amplitude in the wake. The calculated data suggest that the wake amplitude can be minimized by optimizing the cavitation number and the design and setting angles of the steps.

117. Interaction of red blood cells with arrays of laser-induced cavitation bubbles

P. Quinto-Su	*Nanyang Technological University Singapore*
R. Dijkink	*University of Twente, Enschede,The Netherlands*
F. Prabowo	*Nanyang Technological University Singapore*
K. Gunalan	*Nanyang Technological University Singapore*
P. Preiser	*Nanyang Technological University Singapore*
C. Ohl	*Nanyang Technological University Singapore*

We use a spatial light modulator (SLM) to simultaneously create several laser-induced cavitation bubbles. The geometry of the bubble array results in different flow patterns during the expansion and collapse of the bubbles. Hence, the induced shear stresses that affect cells are also modified by the geometry. The events are imaged using strobe photography and high speed cameras. In particular we study the deformability of red blood cells due to the shear stresses, since RBC deformability is a potential indicator for several diseases including malaria [1].

120. The classical multicomponent nucleation theory for cavitation in water with dissolved gases

T. Němec	*Institute of Thermomechanics ASCR, Czech Republic*
F. Maršík	*Institute of Thermomechanics ASCR, Czech Republic*

The Classical Nucleation Theory (CNT) in its multicomponent form is presented as a reliable tool for the investigation of homogeneous nucleation in cavitation processes in aqueous systems. Several cases are considered, starting form the most simple case of a void cavity emerging at negative pressure, then treating the case of a bubble composed of water vapor only, and finally investigating the influence of various gases dissolved in water on the nucleation rate and the composition of the critical cluster. Aqueous systems related to cavitation in hydraulic machinery and to cavitation in carbonated beverages are discussed.

122. Simulation of Cavitation Instabilities in Inducers

A. Hosangadi	*Combustion Research and Flow Technology, Inc.*
V. Ahuja	*Combustion Research and Flow Technology, Inc.*
R. Ungewitter	*Combustion Research and Flow Technology, Inc.*

The cavitating performance of a sub-scale configuration of the SSME low pressure fuel pump (LPFP) has been simulated at off-design flow conditions where a back-flow vortex is generated at the leading edge. The numerical simulations have been compared with measured experimental data both for velocity profiles upstream of the inducer as well dynamic pressure traces on the shroud at the leading edge. Velocity profiles in the back-flow vortex for flow rates down to 70 percent of design were quantified; the swirl velocity comparisons were good while the axial velocity profile were reasonable but slightly over predicted the core velocity. Dynamic cavitating performance was modeled at a moderate Nss number of 20000 for 90 percent of design flow coefficient where rotational cavitation modes are present The source of this instability resulted from the interaction of the cavity with the neighboring blade leading to the detachment of the cavity that rotates relative to the blade and generates an asymmetric cavity pattern. The asymmetrical cavities generate a large radial load on the shaft which rotates at the fundamental mode of the rotational cavitation. For the sub-scale configuration the radial force amplitude was 186 lb-f which gives a non-dimensional force factor of 0.0116.

Spectral analyses of the dynamic pressure traces on the shroud, at the leading edge plane, were compared with experimental measurements. The fundamental rotational cavitation mode was observed to be 125 Hz which is approximately 1.29 N (rotational frequency is 96.6. Hz); both the frequency and relative amplitude compared well with the unsteady measurements. In addition to the fundamental rotation cavitation mode the data shows substantial energy with multiple peaks in the 5 -7.5 N range. This range was reasonably represented in the numerical results although the spectrum was not as rich. A helical pressure wave at the fundamental mode is found to propagate upstream and a potential for interaction with structural elements was identified.

124. Cavitation of JP-8 fuel in a converging-diverging nozzle: experiments and modeling

I. Dorofeeva	*University of Notre Dame*
F. Thomas	*University of Notre Dame*
P. Dunn	*University of Notre Dame*

This paper presents the results of an experimental investigation and an attempt to model cavitation of gas turbine aviation fuel. The work is motivated by the need to predict cavitation behavior for the design of modern aircraft fuel systems. Fuel cavitation can lead to unexpected degradation in system performance due to the effective compressibility associated with the formation of a two-phase mixture and/or damage of fuel system components due to subsequent bubble collapse. The primary working fluid for the experiments reported in this paper is JP-8, which is the gas turbine fuel most typically used by the United States military. JP-8 consists of over 228 hydrocarbons and is closely related to Jet A-1, which is the most common commercial gas turbine fuel. Experiments are also performed with dodecane and decane which are two of the primary constituents of JP-8 by weight but have disparate vapor pressures. Following the experimental study by Davis [7,8], a two-dimensional converging-diverging (C-D) nozzle geometry was selected for this experimental investigation. This relatively simple geometry is nonetheless capable of producing many of the essential features of fuel cavitation, including compressibility, choking, bubbly shock formation, and bubble collapse. In this paper, streamwise nozzle pressure distributions are presented for choked cavitating nozzle flows with water, JP-8, dodecane and decane the working fluids. An analytical model is developed which is shown capable of duplicating many essential features of the measured nozzle pressure distributions including streamwise location of a bubbly shock and the associated pressure jump.

125. The influence of aerodynamic pressure on the water-entry cavities formed by high-speed projectiles

J. Aristoff *Massachusetts Institute of Technology*

We present the results of a theoretical investigation of the vertical impact of high-speed projectiles onto a water surface. A model is developed to describe the evolution of the resulting air cavity. Expressions for the cavity profile and pinch-off time are obtained in the limit where collapse is caused primarily by aerodynamic pressure. Theoretical predictions compare favorably with experimental observations reported in the literature.

127. Numerical simulation of three-dimensional cavitation bubble oscillations by boundary element method

K. Afanasiev *Kemerovo State University, Kemerovo, Russia*
I. Grigorieva *Kemerovo State University, Kemerovo, Russia*

This work is devoted to a numerical investigation of threedimensional cavitation bubble. Bubble oscillations in ambient unbounded fluid are investigated numerically. The fluid is assumed inviscid, incompressible and unbounded and the flow is irrotational. The boundary integral method is used as an instrument of numerical investigation. Much attention is paid to the description of a numerical algorithm.

129. Imaging the effect of acoustically induced cavitation bubbles on the generation of shear-waves by ultrasonic radiation force

J. Gateau *Institut langevin INSERM*
M. Pernot *Institut langevin - ESPCI - CNRS-UMR 7587 - UNIV. PARIS 7 - INSERM*
M. Tanter *Institut langevin - ESPCI - CNRS-UMR 7587 - INSERM*
M. Fink *Institut langevin - ESPCI - CNRS-UMR 7587 - UNIV. PARIS 7 – INSERM*

In soft solids, the acoustic radiation force on bubbles generates a shear wave. This bubble-based shear wave can be imaged using high frame rate ultrasound imaging. We report here an experiment where cavitation is induced in a tissue mimicking material by an ultrasonic tone-burst excitation, which also pushes the bubbles. The generated shear wave was imaged and the energy backscattered by the bubbles measured. The tone burst excitation was iterated at the same location and the decrease of both the amplitude of the particle velocity induced by the shear wave and the backscattered energy was shown. Data treatment to extract the bubbles' contribution to this two quantities, and a simple theoretical model allowed us to point out their linear dependence.

130. Controlled supercavitation formed by a ring type wing

V. Makhrov *Moscow Aviation Institute*

The paper present the something results of theoretical and experimental research axisymmetric supercavity flow formed by a ring type wing. This flow is named “Lighthill-Shushpanov flow”. It has been solved by distributing vortices singularities on combination the “body-ring wing-cavity” surface. Numerical solutions of system integral-differential equations were obtained using a spline function for the cavity shape with positive and negative numbers of cavity. The results of the cavity experimental testing has been cited as an example of the new method formed cavity flow.

131. Modeling collapse aggresiveness of cavitation bubbles in hydromachinery

P. Zima *Department of Thermodynamics, Institute of Thermomechanics, Academy of Sciences Czech Republic*
M. Sedlář *SIGMA Research and Development InstituteCzech Republic*
M. Müller *Faculty of Mechanical Engineering, Czech Republic*

A simple model of assessment of collapse aggressiveness of cavitation bubbles suitable for application at pump design stage is proposed. The model is focused on quantifying the energetic effects of single bubble collapses and the emphasis is placed on computational efficiency. The objective of the model is to provide a rapid estimation of the erosion risk for steadystate or near-steady-state flow with traveling bubble cavitation. The proprietary 3-dimensional Navier-Stokes code for turbulent flow in hydrodynamic machinery is coupled with the unreduced Rayleigh-Plesset equation for incompressible liquid by virtue of the iterations of continuity and momentum equations to account for density changes in the bubbly regions (two-way coupling). The model of collapse aggressiveness of the cavitation bubbles is based on the estimation of the energy dissipated between two successive bubble rebounds. The model is tested for a 2-dimensional hydrofoil in the cavitation tunnel of SIGMA Research and Development Institute in Lutín. The closed-loop tunnel is equipped with the acoustic bubble spectrometer to measure the nuclei population in the test section inlet. The erosion pattern of the hydrofoil surface is monitored using optical profilometry. The results indicate the dominant effect of the first (most energetic) collapses, however, the model overestimates the importance of smaller nuclei mainly due to their large number in the spectra. Introduction of a threshold for the minimum collapse energy required to form any erosive potential seems necessary to rectify this deficiency. The model, although aimed to achieve efficiency and simplicity by relying on the single-bubble dynamics, shows good agreement with the experimental evidence. An attempt to apply the model to the 3-dimensional geometry of a mixed-flow pump impeller is also presented. This paper is a report on part one of work in progress. In the second part the results of the ongoing pitting tests will be used to develop a model of erosive potential.

132. Dynamics of a vapour bubble near a thin elastic plate

M. Shervani-Tabar	*Department of Mechanical Engineering, University of Tabriz*
M. Shabgard	*Department of Mechanical Engineering, University of Tabriz*
M. Rezaee	*Department of Mechanical Engineering, University of Tabriz*
R. Zabihyan	*Department of Mechanical Engineering, Tabriz Azad University*

Numerical and experimental results show that during the collapse phase of a vapor bubble near a rigid boundary, in the absence of strong buoyancy forces, a liquid micro jet is developed on the side of the bubble far from the rigid surface and directed towards it. Numerical and experimental results also show that, in the case of a bubble near a free surface, during the collapse phase of the bubble and in the absence of strong buoyancy forces, the vapor bubble is repelled by the free surface. In this case a liquid micro jet is developed on the closest side of the bubble to the free surface and is directed away from it. The dynamic behavior of a vapor bubble near a free surface leads to the idea that a vapor bubble during its growth and collapse phases near a deformable diaphragm may have a behavior similar to its behavior near a free surface.

In this paper dynamics of a vapor bubble during its growth and collapse phases near a thin elastic plate is investigated. It has been shown that the growth and collapse of a vapor bubble generated due to a high local energy input causes considerable deformation on the nearby thin elastic plate.

Different thin elastic plates with different thicknesses and different flexural rigidities are assumed and the dynamic behavior of a vapor bubble near each of these plates is investigated. Results show that during the growth and collapse of a vapor bubble near a thin elastic plate with a proper thickness and flexural rigidity, in the absence of strong buoyancy forces, a liquid micro jet may develop on the closest side of the bubble to the thin elastic plate and directed away from it.

134. Acoustically induced and controlled micro-cavitation bubbles as active source for transcranial adaptive focusing

J. Gateau	*Institut Langevin -ESPCI - CNRS-UMR 7587 - UNIV. PARIS 7 - INSERM*
L. Marsac	*supersonic imagine*
M. Pernot	*Institut Langevin -ESPCI - CNRS-UMR 7587 - UNIV. PARIS 7 - INSERM*
J. Aubry	*Institut Langevin -ESPCI - CNRS-UMR 7587 - UNIV. PARIS 7 - INSERM*
M. Tanter	*Institut Langevin -ESPCI - CNRS-UMR 7587 - UNIV. PARIS 7 - INSERM*
M. Fink	*Institut Langevin -ESPCI - CNRS-UMR 7587 - UNIV. PARIS 7 – INSERM*

The skull bone is a strong aberrating medium for ultrasound in the low MHz range. Brain treatment with High Intensity Focused Ultrasound (HIFU) can however be achieved through the skull by multichannel arrays using an adaptive focusing technique. Time-reversal is a robust adaptive technique for correction of aberrations. It achieves moreover a matched filter and then allows the optimal energy concentration for thermal therapy. Nevertheless, this method requires a reference signal sent by a source embedded in brain tissues. Acoustically generated cavitation bubbles are active acoustic sources which can be remotely generated. Therefore, they are suited for this non-invasive time reversal aberration correction. We report here in vitro experiments where micro-cavitation was induced transcranially in agar gel at targeted positions using a coarse aberration correction either obtained from CT-scan based simulations or conventional steering. The bubbles' ultrasonic signature received by the array were then successfully used to optimally focus at the designated locations.

135. Cavitation analysis of a double acting podded drive during ice milling

R. Sampson	*Emerson Cavitation Tunnel, UK*
M. Atalr	*University of Newcastle, UK*
N. Sasaki	*National Maritime Research Institute, Japan*

Propeller ice interaction is a complex phenomenon, which relies on innovative and complex experimental research in dedicated ice tank facilities. Whilst the ice tanks model the contact forces with good agreement, the hydrodynamic loading is often only assumed due to the inability to scale atmospheric pressure during these experiments. A small amount of research is however being conducted in cavitation tunnels using innovative methods to represent full scale ice milling conditions. Propulsor ice interaction tests in a cavitation tunnel are therefore both novel and uncommon due to their complexity. The Emerson Cavitation Tunnel at the University of Newcastle (ECT) has pioneered a series of ice milling tests within a cavitation tunnel using model ice. These ice milling tests allow an ice propulsor to experience correctly scaled cavitation numbers as a propeller interacts with ice. In all of the conditions tested, the current research observed presence and influence of cavitation and showed it to be a significant factor, something that is missing from standard ice tank tests. The work published in this paper forms part of a PhD research into the topic by the principal author [1].

136. Air entrainment mechanisms from artificial supercavities: Insight based on numerical simulations

M. Kinzel	*Penn State University - Applied Research Lab.*
J. Lindau	*Penn State University - Applied Research Lab.*
R. Kunz	*Penn State University - Applied Research Lab.*

Using multiphase computational simulations based on the Navier-Stokes equations, we examine the internal gaseous flows of artificially ventilated supercavities. These simulations indicate that air shear layers that develop on the cavity-wall (the air-liquid interface surrounding the cavity) are an important mechanism of air entrainment. This corroborates previous theory developed for toroidal cavities, and indicates that similar mechanisms occur in twin-vortex cavities and cavities closing on bodies. The importance of these shear layers on the cavity behavior potentially impacts computational simulations, experiments, and design-level models. Lastly, a more inclusive, semi-empirical air entrainment model is presented that attempts to accommodate the observed processes.

137. An examination of thermal modeling affects to the numerical prediction of large-scale cavitating fluid flow

M. Kinzel *Penn State University - Applied Research Lab.*
J. Lindau *Penn State University - Applied Research Lab.*
R. Kunz *Penn State University - Applied Research Lab.*

The importance of modeling thermal effects in cavitatingfluid is examined in the context of computational fluid dynamics. Simulations of cavitation in water are used to study the effects of thermal versus and pressure variations in the fluid properties, and their impact on predictions. These studies are extended to evaluate energy-conserving approaches compared to isothermal ones, to assess the underlying thermal models influence on the predicted cavities occurring in water. Results indicate that the thermal effects remain important, but only for specific applications that need high-frequency phenomena from the numerical simulation. Low-frequency measures, needed for loading analysis, appear to be relatively insensitive to thermal effects. Lastly, various thermally driven cavitation problems requiring energy-equation conservation are presented to display applications requiring such a formulation.

138. A dual-time implicit preconditioned Navier-Stokes method for solving 2D steady/unsteady laminar cavitating/noncavitating flows using a Barotropic model

K. Hejranfar *Sharif University of Technology, Tehran, Iran*
E. Ezzatneshan *Sharif University of Technology, Tehran, Iran*
K. Hesary *Sharif University of Technology, Tehran, Iran*

A two-dimensional, time-accurate, homogeneous multiphase, preconditioned Navier-Stokes method is applied to solve steady and unsteady cavitating laminar flows over 2D hydrofoils. A cell-centered finite-volume scheme employing the suitable dissipation terms to account for density jumps across the cavity interface is shown to yield an effective method for solving the multiphase Navier-Stokes equations. This numerical resolution is coupled to a single-fluid model of cavitation that the evolution of the density is governed by a barotropic sate law. A preconditioning strategy is used to prevent the system of equations to be stiff. A dual-time implicit procedure is applied for time accurate computation of unsteady cavitating flows. A sensitivity study is conducted to evaluate the effects of various parameters such as numerical dissipation coefficients and preconditioning on the accuracy and performance of the solution. The computations are presented for steady and unsteady laminar cavitating flows around the NACA0012 hydrofoil for different conditions. The solution procedure presented is shown to be accurate and efficient for predicting steady/unsteady laminar cavitating/noncavitating flows over 2D hydrofoils.

139. Vorticity Confinement methods for cavitating flows

T. Hachmann *University of Technology Hamburg-Harburg, Germany*
U. Lantermann *University of Technology Hamburg-Harburg, Germany*
M. Abdel-Maksoud *University of Technology Hamburg-Harburg, Germany*
D. Hänel *University of Duisburg-Essen*

Present work deals with investigations of numerical aspects of cavitating vortex dominated flows. Computations of the viscous flow on realistic, technical configurations require efficient methods and high grid resolution, which is not sufficient in most cases to capture important details of the flow. Insufficient resolution increases the numerical dissipation of vortices generated at the tip region of lifting surfaces. One possible solution to reduce the unphysical decay of the strength of the vortices (despite of moderate resolution) is the application of vorticity confinement methods. Aim of the paper is the development and the comparison of Vorticity Confinement (VC) methods for cavitating flows on unstructured grids. Applications are proposed to control devices and marine propulsion systems. The numerical dissipation of vortices is compared for different VC formulations. Especially the influence of the source terms on cavitating flows is investigated. The numerical computations are carried out by the finite volume solution method FreSCo on arbitrary grids. In the study vorticity confinement techniques are combined with different cavitation models available in the applied numerical method to investigate tip vortex flow. The cavitation models are based on Volume-of-Fluid (VoF). A NACA16020 elliptical wing is selected as a validation case. The combination of vorticity confinement formulations and cavitation models enables a better and a more detailed study of cavitation effects.

141. Numerical prediction of cavitation and pressure fluctuation around marine propeller

K. Sato *Mitsubishi Heavy Industries, Ltd.*
A. Ohshima *Mitsubishi Heavy Industries, Ltd.*
H. Egashira *Mitsubishi Heavy Industries, Ltd.*
S. Takano *Mitsubishi Heavy Industries, Ltd.*

The applicability of numerical prediction method for cavitation around marine propeller was studied. A commercial CFD code was applied for computation of 10 different propellers. The computed cavitation patterns and pressure fluctuations were compared with model test. As the result, it's shown that this method can be used for the prediction of the behavior of sheet cavitation and the pressure fluctuation of the 1st order of blade frequency component.

142. High-speed photography of supercavitation and multiphase flows in water entry

H. Shi *College of Mechanical Engineering and Automation, Zhejiang Sci-Tech University, China*
M. Itoh *Department of Mechanical Engineering, Nagoya Institute of Technology, Japan*

The supercavitation in water entry and associated multiphase flows were studied using a high-speed camera and single-shot optical device. The formation, growth and collapse of supercavities induced by underwater high-speed projectiles were revealed. The unsteady fluid dynamic processes of the splash, surface deformation, supercavity twisting, down jet in the cavity, etc. were also studied. Both axisymmetrical and three-dimensional supercavities were found in the experiment. The shape of the axisymmetrical supercavity has been comparied with Logvinovich's model. The three-dimensional supercavity is caused by the trajectory deflection of the underwater projectile, which is also related unsteadiness and turbulence of the flow field. When the supercavity is twisted, a grain-like cavitating bubble is formed after the upper and lower pinch-offs. It is newly found that during the surface closure (surface seal), the down-jet is generated simultaneously with the formation of upwards splash dome.

143. Prediction of tip vortex cavitation inception on marine propellers at an early design stage

J. Hundemer *Hamburg University of Technology*
M. Abdel-Maksoud *Hamburg University of Technology*

The inception of vortex cavitation at an early design stage is still difficult to forecast. The most reliable prediction of the full scale performance is achieved by means of model tests, which are possible for few designs only. A simplified model to calculate the inception of tip vortex cavitation is developed and tested. The model is based on results obtained from potential flow theory, using a boundary element method. The developed tip vortex cavitation inception model and also the panel method are described, after a short introduction to vortex cavitation. The numerical behaviour of the model is investigated for an elliptic wing at different angles of attack and two marine propellers in homogenous and not axially symmetric inflow. The cavitation model's properties concerning different Reynolds numbers are studied and the scale effects on calculated model- and full-scale tip vortices are discussed.

145. A simple approach to estimating three-dimensional supercavitating flow fields

II. Kirschner *Alion Science and Technology Corporation*
R. Chamberlin *Alion Science and Technology Corporation*
S. Arzoumanian *Alion Science and Technology Corporation*

A simple method is formulated for predicting three-dimensional supercavitating flow behind cavitators subject to gravitational acceleration and motion of the cavitator. The method applies slenderbody theory in the context of matched asymptotic expansions to pose an inner problem for the cavity evolution downstream from the locus of cavity detachment. This inner problem is solved by means of a coupled set of equations for the Fourier coefficients characterizing the cavity radius and the velocity potential as a function of downstream location and circumferential location, thus resulting in a two-dimensional multipole solution at each station. For the lowest-order term in the Fourier expansion, it is necessary to match the parabolic inner solution to a fully elliptic outer solution. This step allows the application of any one of a number of methods to solve the axisymmetric problem, which serves as the base solution that is perturbed by the three-dimensional effects. The method is an attempt to formalize the Logvinovich principle of independent cavity section evolution. Results flow past a circular disk cavitator are presented for severalvalues of the cavity Froude number.

146. Control experiments with a semi-axisymmetric supercavity and a supercavity-piercing fin

M. Wosnik *University of New Hampshire*
R. Arndt *University of Minnesota*

Supercavitation can significantly reduce skin-friction drag on an underwater body, thus enabling a dramatic increase in attainable velocity. The control of a High-Speed Supercavitating Vehicle (HSSV) poses unique challenges, since only small regions at the nose (cavitator) and on the afterbody (fins) are in contact with water and can be used as control surfaces. The interaction between supercavity dynamics and control surface actuation is complex and nonlinear. Experiments were conducted with a semi-axisymmetric, ventilated supercavity and a single wedge-shaped, 45 degree swept, cavity-piercing fin in the high-speed water tunnel at St. Anthony Falls Laboratory. Motion control was combined with water tunnel testing to create a "hardware-in-the-loop" system that can (a) provide critical hydrodynamic parameters for control models and (b) serve as a test bed for fin control strategies. Through a series of experiments, control surfacecavity interaction, cavity stability and hysteresis effects were studied. Fin torque (lift) was measured for different angles of attack with varying cavitation numbers. Closed-loop fin control experiments simulating simple maneuvers were carried out.

149. Mechanism and scalability of tip vortex cavitation suppression by water and polymer injection

N. Chang *University of Michigan, Ann Arbor*
R. Yakushiji *University of Michigan, Ann Arbor*
H. Ganesh *University of Michigan, Ann Arbor*
S. Ceccio *University of Michigan Ann Arbor*

Tip vortex cavitation (TVC) is typically the first form of cavitation observed in propellers; therefore a delay in its onset is sought. TVC suppression via mass injection in the core of the vortex was studied with an elliptical plan-form hydrofoil NACA-66 modified in a re-circulating water tunnel with known nuclei distribution. The solutions injected were water and Polyox WSR 301 solution with concentration ranging from 10 to 500pmm. It was observed that TVC was suppressed in all cases where mass was injected. Higher polymer concentration solutions and higher flux rate provided a drop in cavitation desinence of 1.8. Water injection at the lowest flux rate

provided the smallest drop in cavitation desinence, 0.03. A saturation effect for the TVC suppression was also observed for both the polymer concentration (125ppm) and volume flux rate (Q_{jet} / Q_{core} = 0.48). The mechanisms and scalability that lead to TVC suppression via mass injection will be investigated.

151. Numerical investigation of cavitating flow through the cascade of arbitrary foil

A. Terentiev	*Chuvash Polytechnic Institute of Moscow State Open University*

Two methods are considered for computer modeling of cavitating flow through a cascade of any foils. The first method consists of numerical modeling of non-circulation flows of a cascade of foils with subsequent analytical solution of platecascades. The second method provides direct computer modeling using numerical algorithms for an isolated foil. It is shown, that both methods yield identical numerical results but the second one is more convenient for numerical algorithms and computing.

152. Experimental study of the effects of viscosity and viscoelasticity on a line vortex cavitation

C. Barbier	*Dynaflow, Inc.*
G. Chahine	*Dynaflow, Inc.*

This paper investigates the influence of viscosity and viscoelasticity on the structure of the flow in a line vortex in view of understanding their effects on cavitation inception. Experiments were conducted in a vortex chamber where the fluid injection speed and the liquid properties can be easily controlled. Measurements of the velocities, pressures, and thus the cavitation number were conducted using a PIV system, pressure gauges, and Pitot tubes. Experiments were performed using water, different dilute concentrations of polymer (POLYOX WSR 301) solutions, and solutions with different concentrations of corn syrup for a large range of Reynolds numbers. The measurements and observations showed that cavitation inception at the vortex center was delayed when polymer and corn syrup solutions are used as compared to the experiments in water. However, contrary to reported observations with tip vortices, here the large scale vortex was found to rotate faster in the polymer and corn syrup solutions. This did not match with our observations of cavitation inception delay in the case of polymers and the conventional thinking about the relationship between pressures and velocities in a vortex line. This may be due to the observations that the velocity fluctuations and the turbulent kinetic energy in the viscous core region increased significantly in the polymer and corn syrup solutions and could question the validity of a pressure computation based on a single vortex.

153. Numerical study of vortex cavitation supression with polymer injection

C. Hsiao	*Dynaflow, Inc.*
Q. Zhang	*Dynaflow, Inc.*
G. Chahine	*Dynaflow, Inc.*

A FENE-P model was implemented in the Navier-Stokes solver, 3DYNAFS-VIS, to simulate polymer solutions viscoelastic effects on tip vortex cavitation. Two problems encountered in tip vortex cavitation dynamics were studied numerically. The first problem is that of bubble growth/collapse in a line vortex. The second problem is that of the dynamics a propeller tip vortex in water and in polymer solutions. Bubble growth/collapse in the polymer solution was found to have a shorter period and smaller volume oscillation amplitude than in water. The bubble shape also differs from that in water. Concerning the propeller flow, a RANS solution was first obtained for the full flow field, and then improved by conducting direct Navier Stokes simulations within a reduced domain encompassing the tip vortex with a much finer grid mesh. The viscoelastic effects were only considered in the reduced domain, which is an approximation of the case of local polymer injection such as from the propeller tip. The pressure along the vortex centerline was found to be higher for the polymer solution than for water. The maximum tangential velocity along the vortex for polymer solutions was found to be reduced, and the axial velocity component increased. In the two problems investigated, the presence of polymer makes cavitation more difficult to occur, which is consistent with the experimental observations.

154. Influence of propeller presence and cavitation on a liquid nuclei population

R. Raju	*Dynaflow, Inc.*
C. Hsiao	*Dynaflow, Inc.*
G. Chahine	*Dynaflow, Inc.*

The modification of the bubble nuclei population due to the presence of a rotating propeller was modeled numerically using a combined Eulerian - Lagrangian approach. The liquid flow field was solved using a Reynolds Averaged Navier-Stokes (RANS) solver and was improved further using Direct Navier-Stokes Simulations (DNSS) to better capture the wake rollup in the tip-vortex. Bubbles were propagated in the resulting flow field using a Lagrangian approach and computing bubble motion and volume change. Resulting nuclei distribution modification downstream of the propeller for various advance coefficients was studied. The results show that different advance coefficients can result in very different cavitation zones near the propeller as the cavitation number is reduced below the cavitation inception limit. This has a strong effect on the nuclei population downstream of the propeller.

We further examine the effect of non-uniformity in the upstream distribution of nuclei, and the effects of gravity and propeller scale on the results. Increasing the scale appears to play a major role in increasing the void fraction downstream.

156. Development of measurement techniques for studying propeller erosion damage in severe wake fields

W. Pfitsch *NSWCCD*
S. Gowing *NSWCCD*
D. Fry *NSWCCD*
M. Donnelly *NSWCCD*
S. Jessup *NSWCCD*

Preliminary propeller erosion tests have been conducted at the Naval Surface Warfare Center Carderock Division 24 inch variable pressure water tunnel (VPWT), shown in Figure 1, to establish testing procedures for evaluating various coatings to minimize cavitation erosion damage to marine propellers. A severe wake field was produced using a two dimensional, thick foil ahead of a downstream driven propeller model. This approach was derived from similar tests conducted by Miller [11]. Conventional cavitation viewing was performed with cameras viewing through the tunnel side window. Images were acquired using high speed (up to 6000 fps) and high resolution (2K x 2K) cameras. In addition, a waterproof camera was mounted inside the foil looking directly downstream at the suction face of the blade. Two propellers were tested, a 16 inch (0.406 m) diameter propeller 5388 and a 12 inch (0.305 m) diameter propeller 4119 [8]. The foil wake field was measured with LDV surveys. Accelerometers were mounted in the water tunnel test section to measure acoustic emissions of cavitation activity.

Cavitation erosion was observed at the tip of the 16 inch diameter propeller due to excessive tip vortex, and complicated vortex collapse. Moderate erosion was also observed at the inner radii, where leading edge sheet cavitation collapsed. Scanning techniques for quantifying propeller erosion damage were evaluated. These studies will transition to the 36-inch VPWT where a number of geosym propellers of different materials and coating will be assessed in a similar wake field.

157. Model for the oscillations of the shell of a contrast agent, liquid and solid cases

J. Naude *Facultad de Ingenieria Universidad acional Autonoma de Mexico*
F. Mendez *Facultad de Ingenieria Universidad acional Autonoma de Mexico*

In the present theoretical formulation we study the oscillations of a solid bubble (shell), surrounding a perfect gas, under an ultrasound drive pressure. The model comprehends a Rayleigh-Plesset kind equation for the liquid radius that surrounds the shell together with an equation for the oscillating pressure in the liquid and the exact solution for an elastic solid that represents the shell. We state as main parameters b which represent the quasi steady state deformation of the solid, pX is the ratio of the ambient pressure and the elasticity modulus, and bA which represents the ratio of the driving pressure and the dynamic pressure. As main results we give the oscillations of the liquid in time and the pressure also. For the solid we obtained a weak oscillatory and irregular behavior which predicts the final collapse of the solid.

161. Prediction research on cavitation performance for centrifugal pumps

W. Yong *Technical and Research Center of Fluid Machinery Engineering ,jiangsu University 212013*
L. Houlin *Technical and Research Center of Fluid Machinery Engineering ,jiangsu University 212013*
Y. Shouqi *Technical and Research Center of Fluid Machinery Engineering ,jiangsu University 212013*
T. Minggao *Technical and Research Center of Fluid Machinery Engineering ,jiangsu University 212013*
W. Kai *Technical and Research Center of Fluid Machinery Engineering ,jiangsu University 212013*

The present situation about cavitation performance prediction of centrifugal pump is introduced. The primary methods of cavitation performance prediction for centrifugal pumps are summarized, including numerical simulation method and artificial neural network method. Based on the neutral network toolbox of MATLAB7.0, topological structures of artificial neural networks are determined and network models for predicting cavitation performance of centrifugal pumps are established by analyzing the relations between geometric parameters of centrifugal pumps and net positive suction head at designed flow rate, The BP and RBF neural networks are trained by 60 example data, which are obtained from engineering practice and normalized by using neural network toolbox function. The cavitation flow in centrifugal pumps is simulated by using the commercial CFD code FLUENT6.2. A moving reference frame technique is applied to take into account the impeller-volute interaction. The standard k-ε turbulence model, mixture multiphase model and SIMPLEC algorithm are used. Velocity inlet and pressure-outlet are set as boundary conditions.The cavitation performance curves at design condition are predicted by calculating the head under different net positive suction head. The cavitation performances of 3 pumps with the different specific speeds are predicted by using numerical simulation method and neural network method respectively. The predicted values are compared with the tested values, the results show that the predictions by two methods are satisfied, the relative declination of BP and RBF for 3 pumps are 2.87％, 2.55％, 5％ and 3.71％, 3.27％, 4.62％ respectively. The absolute declinations of numerical simulation method are 0.17m, 0.08m and 0.16m. The advantage and disadvantage of those two methods are compared, The numerical simulation method will take a lot of time to modeling and calculating, but the law of cavitation flow in the centrifugal pumps can be obtained, which are helpful to disclosing the mechanism of cavitation characteristic; The artificial neural networks method needs a great deal of training examples, which are necessary and important to the prediction accuracy, but the math relation between input variables and output variables can be set up by using artificial neural network method, which is useful to optimize the structure of pumps.

164. Observations and numerical simulations of unsteady partial cavitation on 2-D hydrofoil

X. Peng China Ship Scientific Research Center

In present paper, experimental observations and numerical simulations are used to understand some flow mechanism of partial cavity, such as the instability of attached sheet cavitation and the relations between cloud cavitation and erosion. NACA0012 and NACA16012 hydrofoils were employed both as the test model in the cavitation tunnel experiments and the model for numerical simulations. The high-speed video camera and TR-PIV system was used in the experiments. The re-entrant jet was captured successively in experiment and the mechanism that the re-entrant jet induced the break-off of cavity is validated. In numerical simulation, Reynolds-Averaged Navier-Stokes equations with the cavitation model of Transport Equation Model (TEM) were developed. The cavity morphology, flow structure, instability and developing process, were investigated. The numerical results showed that re-entrant jet is not absolutely constituted by liquid but vapor/liquid mixture, and the break-off process of sheet cavitation is directly induced by re-entrant jet closely along the wall. The calculating shedding frequencies are very close to the experimental data.

166. Precursor luminescence near the collapse of laser-induced bubbles in alkali-salt solutions

H. Chu UCLA
S. Vo UCLA
T. Hsieh UCLA

A precursor luminescence pulse consisting of atomic line emission is observed as much as 150 nanoseconds prior to the collapse point of laser-induced bubbles in alkali-metal solutions. The timing of the emission from neutral Na, Li, and K atoms is strongly dependent on the salt concentration, which appears to result from resonant radiation trapping by the alkali atoms in the bubble. The alkali emission ends at the onset of the blackbody luminescence pulse at the bubble collapse point, and the duration of the blackbody pulse is found to be reduced by up to 30% as the alkali salt concentration is increased.

167. Gas bubble growth dynamics in a supersaturated solution: Henry's and Sievert's solubility laws

A. Kuchma Saint-Petersburg State University
G. Gor Saint-Petersburg State University
F. Kuni Saint-Petersburg State University

Theoretical description of diffusion growth of a gas bubble after its nucleation in supersaturated liquid solution is presented. We study the influence of Laplace pressure on the bubble growth. We consider two different solubility laws: Henry's law, which is fulfilled for the systems where no gas molecules dissociation takes place and Sievert's law, which is fulfilled for the systems where gas molecules completely dissociate in the solvent into two parts. We show that the difference between Henry's and Sievert's laws for chemical equilibriumconditions causes the difference in bubble growth dynamics. Assuming that diffusion flux of dissolved gas molecules to the bubble is steady we obtain differential equations on bubble radius for both solubility laws. For the case of homogeneous nucleation of a bubble, which takes place at a significant pressure drop bubble dynamics equations for Henry's and Sievert's laws are solved analytically. For both solubility laws three characteristic stages of bubble growth are marked out. Intervals of bubble size change and time intervals of these stages are found. We also obtain conditions of diffusion flux steadiness corresponding to consecutive stages. The fulfillment of these conditions is discussed for the case of nucleation of water vapor bubbles in magmatic melts. For Sievert's law the analytical treatment of the problem of bubble dissolution in a pure solvent is also presented.

169. Physical mathematical bases of the principle of independence of cavity expansion

V.Serebryakov Institute of Hydromechanics of NASU, Kiev, Ukraine

For the first steps of cavitation researches the very important general peculiarity of supercavitating flow was found which discovered practical independence of cavity sections expansion in motionless fluid. This peculiarity gave the possibility for practical estimation of the cavities in the most part of applications. The paper presents the system of the simple dependencies for practical calculations of axisymmeric and near to one supecavitation flows with account of its perfection on the base of modern achievements of the theoretic and experimental research which based on the property of independence of the cavity section expansion. Main attention is paid to asymptotic dependencies on the base of Slender Body Theory and heuristic models. The calculations examples of steady and unsteady cavities for motion under gravity, axelration, harmonic oscillation of pressure are given. The problems of ventilated cavities and possible ways of drag reduction for motion with supercavitation are considered.

172. Shock propagation in polydisperse bubbly flows

K. Ando California Institute of Technology
T. Colonius California Institute of Technology
C. Brennen California Institute of Technology

The effect of distributed bubble nuclei sizes on shock propagation in dilute bubbly liquids is computed using a continuum two-phase model. An ensemble-averaging technique is employed to derive the statistically averaged equations and a finitevolume method is used to solve the model equations. The bubble dynamics are incorporated using a Rayleigh-Plesset-type equation which includes the effects of heat transfer, liquid viscosity and compressibility. The numerical model is verified by computing linear wave propagation and comparing to the acoustic theory of dilute bubbly liquids. It is known that for the case of monodisperse mixtures, relaxation oscillations occur behind the shock due to the bubble dynamics. The present computations show that bubble size distributions lead to additional damping of the average shock dynamics. If the distribution is sufficiently broad, the effect of polydispersity dominates over the singlebubble-dynamic damping and the shock profile is smoothed out. The size distribution effect on bubble screen problems is also discussed.

173. Numerical study on the surface stability of an encapsulated microbubble in the ultrasound field

Y. Liu — *Department of Mechanical Engineering, The University of Tokyo*
K. Sugiyama — *Department of Mechanical Engineering, The University of Tokyo*
S. Takagi — *Organ and Body Scale Team, RIKEN; Department of Mechanical Engineering, The University of Tokyo*
Y. Matsumoto — *Department of Mechanical Engineering, The University of Tokyo*

The surface stability problem of an encapsulated microbubble in an ultrasound field is numerically addressed. To predict the nonlinear process, the continuity equation and Navier-Stokes equation are directly solved by means of a boundary-fitted finitevolume method on an orthogonal curvilinear coordinate system. The bubble is insonified by an ultrasound pulse consisting of a burst of 10 cycles, of which the first and last two periods are modified by a Gaussian envelope. The simulation code reproduces a shape oscillation of a gas bubble with an initial radius of 30μm at a pressure frequency of 130kHz as shown in experimental and theoretical studies [1]. The effects of the membrane on the shape oscillation are investigated through simulations of a micrometersized bubble encapsulated with a neo-Hookean membrane at an ultrasonic frequency of 1MHz. The encapsulated bubble presents a second-order shape instability, while the gas bubble of the same size keeps spherical because the surface tension significantly suppresses the shape oscillation. The strain-softening features with increasing the oscillation amplitude are characterized by a larger expansion and the higher harmonics when the bubble contracts.

175. Design of cavitation-free hydrofoils by a given pressure envelope

D. Maklakov — *Professor of Kazan State Univercity of Architecture and Engineering*
F. Avkhadiev — *Professor of Kazan State University*

In this paper we shortly describe basic aspects of the theory of pressure envelopes which in the frame work of potential flows allows one to design a wing section shape that generates exactly a specified pressure envelope. By means of this theory we analyze and modify a series of hydrofoils designed by Eppler. The modifications based on shifts and proportional stretches of the dependence of the maximum velocity on the angle of attack. Besides, applying the theory, we solve an optimal problem and design a series of optimal hydrofoils which have a maximal width of the pressure bucket. We present accurate estimates of the maximal width as a function of the cavitation number and angle of attack.

177. Damage potential of the shock-induced collapse of a gas bubble

E. Johnsen — *Stanford University*
T. Colonius — *California Institute of Technology*
R. Cleveland — *Boston University*

Numerical simulations are used to evaluate the damage potential of the shock-induced collapse of a pre-existing gas bubble near a rigid surface. In the context of shock wave lithotripsy, a medical procedure where focused shock waves are used to pulverize kidney stones, shock-induced bubble collapse represents a potential mechanism by which the shock energy directed at the stone may be amplified and concentrated. First the bubble dynamics of shock-induced collapse are discussed. As an indication of the damage potential, the wall pressure is considered. It is found that, for bubbles initially close to the wall, local pressures greater than 1 GPa are achieved. For larger stand-off distances, the wall pressure is inversely proportional to the location of bubble collapse. From this relationship, it is found that bubbles within a certain initial stand-off distance from the wall amplify the pressure of the incoming shock. Furthermore, the extent along the wall over which the pressure due to bubble collapse is higher than that of the pulse is estimated. In addition, the present computational fluid dynamics simulations are used as input into an elastic waves propagation code, in order to investigate the stresses generated within kidney stone in the context of shock wave lithotripsy. The present work shows that the shock-induced collapse of a gas bubble has potential not only for erosion along the stone surface, but also for structural damage within the stone due to internal wave reflection and interference.

179. Cavitation phenomena in a stagnation point flow

Y. Lu — *Dept. of Mech. Eng., JHU*
B. Gopalan — *Dept. of Mech. Eng., JHU*
E. Celik — *Dept. of Mech. Eng., JHU*
J. Katz — *Dept. of Mech. Eng., JHU*
D. Van Wie — *APL, JHU*

Cavitation phenomena inherently occur in regions with low pressure. Consequently, it seems unlikely that cavitation would develop near the stagnation point of a blunt body flow. However, in recent experiments involving a high-speed bubbly jet impinging on a blunt body, we have observed substantial rapid growth and stretching of bubbles near the stagnation point over a wide range of flow parameters. Using a high-speed camera we observe that bubbles with initial diameters of tens of microns located very close to the blunt body are being stretched into long “strings” that are generally aligned parallel to the body surface. In-line Digital Holographic Microscopy (DHM) measurements show that the bubble strings are located far from the walls. High resolution 3-D holographic Particle Image Velocimetry (DHM-PIV) is performed to quantify the 3-D flow field near the leading edge of the blunt body. Instantaneous data show vortices being stretched by the local strain field close to the blunt body in an orientation consistent with the appearance of cavitation. These vortices are originated from the turbulent jet upstream. An estimate based on the measured vortex strength and strain field shows that stretching rapidly decreases the pressure in the vortex core below the vapor pressure, explaining the occurrence of cavitation.

180. On some physics to consider in numerical simulation of erosive cavitation

G. Bark *Chalmers University*
M. Grekula *SSPA Sweden*
R. Bensow *Chalmers University*
N. Berchiche *Chalmers University*

This paper discusses several mechanisms in erosive cavitation, which are all important to capture, and study, when assessing the risk of erosion. In particular we introduce the concept of primary and secondary cavitation in order to put emphasis on a particular class of mechanisms: cavitation created in the secondary flow field governed by, e.g., a shedding or collapse of a primary created cavity. These secondary cavities are almost always erosive and have previously not been well described in the literature. The role of cloud cavitation is partly reconsidered and a hypothesis for development of vortex group cavitation, a type of secondary cavitation, is presented. An underlying part of the discussion is how the described cavitation mechanisms influence numerical simulation of cavitation nuisance.

182. Experimental investigations on the flow structure and turbulence of the propeller tip vortex at different cavitation states

Y. Chow *National Taiwan Ocean University*
W. Tsai *National Taiwan Ocean University*
Y. Lee *National Taiwan Ocean University*

In a previous series of open water tests for a certain marine propeller, we kept decreasing the advance coefficient and observed the propeller tip vortex going through a process from non-cavitating to cavitating and back to non-cavitating but micro-bubble-filled conditions. This process provides an excellent opportunity for investigating the flow structure and the turbulence associated with the tip vortex under different statuses of cavitation. Particle image velocimetry technique is used to measure the instantaneous velocity distributions at six experimental conditions parameterized with the advance coefficient and the cavitation number. The results show that the mean circulation (or mean kinetic energy) of either the cavitating or the micro-bubble-filled tip vortex is greater than that of the single-phase (liquid) counterpart, whereas the turbulent kinetic energy associated with the tip vortex has an opposite trend. These phenomena imply that extraordinary kinetic energy sources/transfers within the flow exist due to interactions between the vapor and the liquid phases of the tip vortex fluid.

183. Incepting cavitation acoustic emissions due to vortex stretching

N. Chang *University of Michigan, Ann Arbor*
S. Ceccio *University of Michigan Ann Arbor*

The acoustic signature of the vortex cavitation bubbles can be characterized during inception, growth, and collapse. Growing and collapsing bubbles produced a sharp, broadband, popping sound. However, some elongated cavitation bubbles produce a short tone burst, or chirp, with frequencies on the order of 1 to 6 kHz. The frequency content of the acoustic signal during bubble inception and growth were related to the volumetric oscillations of the bubble and vortex dynamics coupling. A relationship was also found between the frequency of the oscillations and the flow and water quality conditions.

Paper Number	*Title*	*Page Number*
5	Interfacial cavitation nuclei studied by scanning probe microscopy techniques	1
7	Weakly nonlinear analysis of dispersive waves in mixtures of liquid and gas bubbles based on a two-fluid model	1
8	Numerical analysis of hydrofoil ventilated cavitation under wave impact	1
9	Unsteady dynamics of cloud cavitating flows around a hydrofoil	1
11	A water test facility for liquid rocket engine turbopump cavitation testing	2
12	Enhancement of cavitation aggressivity around a cavitating jet by injecting low-speed water jet for cavitation shotless peening	2
13	Detection of cavitation erosion through acoustic emissions techniques	2
15	Prediction of impeller speed dependence of cavitation intensity in centrifugal pump using cavitating flow simulation with bubble flow model	2
16	Slip effects in vortical structure behind cavitating propeller wake	3
17	Inertia controlled instability and small scale structures of sheet and cloud cavitation	3
18	Unsteady bubbly cavitating nozzle flows	3
21	Supercavitating motion of a wedge in a jet	3
22	Numerical prediction and experimental verification of cavitation of globe type control valves.	4
27	Study on correlation between cavitation and pressure fluctuation signal using high-speed camera system	4
30	Numerical study on cavitation erosion risk of marine propellers operating in wake flow	4
32	Comparison of cavitation erosion rate with liquid impingement erosion rate	4
33	Prediction of cavitation erosion based on the measurement of bubble collapse impact loads	5
35	Application of computer vision techniques to measure cavitation bubble volume and cavitating tip vortex diameter.	5
36	Thermodynamic effects on cryogenic cavitating flow in an orifice	5
37	Development of ballast water treatment technology by mechanochemical cavitations	5
38	PIV-LIV determination of mean velocity field and reynolds stress tensor in a cavitating mixing layer	6
40	Analytical investigations of thermodynamic effect on cavitation characteristics of sheet and tip leakage vortex cavitation	6
41	Cavitation Erosion - A critical review of physical mechanisms and erosion risk models	6
42	Investigation on numerical schemes in the simulation of barotropic cavitating flows	7
43	The partial cavity on a 2D foil revisited	7
45	Cavitating propeller flows predicted by RANS solver with structured grid and small reynolds number turbulence model approach	7
46	Modeling and analysis of a cavitating vortex in 2d unsteady viscous flow	8
48	Cavity flow with surface tension past a flat plate	8
49	RANS simulations of a 3D sheet-vortex cavitation	8
50	Discussions on the cause of cavitation instabilities in three-dimensional inducer based on cfd results of alternate blade cavitation	8
52	Numerical simulation of three-dimensional unsteady sheet cavitation	8
53	Influence of shock-bubble and bubble-bubble interactions on the collapse of a cluster of bubbles	9
56	A numerical study of unsteady cavitation on a hydrofoil	9
57	On pressure and temperature waves within a cavitation bubble	9
58	Determination of the tensile strength and the nuclei spectrum by means of the in-situ-nozzle	10
59	Transient hydroelastic analysis of surface-piercing propellers	10
60	Rate-dependent hydroelastic response of self-adaptive composite propellers in fully wetted and cavitating flows	10
61	Cavitation patterns on a plano-convex hydrofoil in a high-speed cryogenic cavitation tunnel	11
62	Numerical investigation of cloud cavitation and cavitation noise on a hydrofoil section	11

63	Development and validation of new cryogenic cavitation model for rocket turbopump inducer	11
64	Evaluation of incipient cavitation erosion for pipe wall at downstream of an orifice	11
65	Thermo-fluid dynamic experiment of He II cavitating flow	12
66	Pressure-wave formation and collapses of cavitation clouds impinging on solid wall in a submerged water jet	12
67	Numerical prediction of cavitation erosion in cavitating flow	12
68	Study on unsteady cavitating flow simulation around marine propeller using a rans cfd code	12
70	Rudder gap flow control for cavitation suppression	13
72	High speed motion in water with supercavitation for sub-, trans-, supersonic mach numbers	13
73	Large eddy simulation of cavitation inception in a high speed flow over an open cavity	13
74	A hybrid lagrangian-eulerian approach for simulation of bubble dynamics	13
77	Surrogate-based modeling of cryogenic turbulent cavitating flows	14
78	Numerical investigation of thermodynamic effect on unsteady cavitation in cascade	14
79	A panel method for trans-cavitating marine propellers	14
87	Assessment of a central difference finite volume scheme for modeling of cavitating flows using preconditioned multiphase euler equations	14
89	Investigation of turbulent modulation by cavitation for subgrid-scale modeling in LES	15
90	Generality of rotating partial cavitation in two-dimensional cascades	15
91	Cavitation in a bulb turbine	15
92	Cavitation and flow instabilities in a 3-bladed axial inducer designed by means of a reduced order analytical model	15
94	Prediction of cavitating flow around 3-D straight/swept hydrofoils	16
97	A boundary element method for the strongly nonlinear analysis of surface-piercing hydrofoils	16
98	Removal of an obstruction from a tube by a collapsing bubble	16
99	Multiphase flow analysis of cylinder using a new cavitation model	16
100	Shallow angle water entry of ballistic projectiles	16
102	A multi-scale study on the bubble dynamics of cryogenic cavitation	17
103	Cavitation as a microfluidic tool	17
104	Bubble shock wave interaction near biomaterials	17
107	A modified sst k-omega turbulence model to predict the steady and unsteady sheet cavitation on 2d and 3d hydrofoils	17
109	Microbubble disruption by ultrasound and induced cavitation phenomena	18
110	A dynamic test platform for evaluating control algorithms for a supercavitating vehicle	18
111	Investigation of the behavior of ventilated supercavities	18
112	Effects of surface characteristics on hydrofoil cavitation	18
113	Blade load dynamics in cavitating and two phase flows	19
114	Blade section design of marine propellers with maximum inception speed	19
116	Numerical modeling of cavity flow on bottom of a stepped planing hull	19
117	Interaction of red blood cells with arrays of laser-induced cavitation bubbles	20
120	The classical multicomponent nucleation theory for cavitation in water with dissolved gases	20
122	Simulation of cavitation instabilities in inducers	20
124	Cavitation of JP-8 fuel in a converging-diverging nozzle: experiments and modelling	20
125	The influence of aerodynamic pressure on the water-entry cavities formed by high-speed projectiles	21
127	Numerical simulation of three-dimensional cavitation bubble oscillations by boundary element method	21
129	Imaging the effect of acoustically induced cavitation bubbles on the generation of shear-waves by ultrasonic radiation force	21

130 Controlled supercavitation formed by a ring type wing 21
131 Modeling collapse aggresiveness of cavitation bubbles in hydromachinery 21
132 Dynamics of a vapour bubble near a thin elastic plate 22
134 Acoustically induced and controlled micro-cavitation bubbles as active source for transcranial adaptive focusing 22
135 Cavitation analysis of a double acting podded drive during ice milling 22
136 Air entrainment mechanisms from artificial supercavities: insight based on numerical simulations 22
137 An examination of thermal modeling affects to the numerical prediction of large-scale cavitating fluid flow 23
A dual-time implicit preconditioned navier-stokes method for solving 2D steady/unsteady laminar cavitating/noncavitating flows using a barotropic model 23
139 Vorticity confinement methods for cavitating flows 23
141 Numerical prediction of cavitation and pressure fluctuation around marine propeller 23
142 High-speed photography of supercavitation and multiphase flows in water entry 24
143 Prediction of tip vortex cavitation inception on marine propellers at an early design stage 24
145 A simple approach to estimating three-dimensional supercavitating flow fields 24
146 Control experiments with a semi-axisymmetric supercavity and a supercavity-piercing fin 24
149 Mechanism and scalability of tip vortex cavitation suppression by water and polymer injection 24
151 Numerical investigation of cavitating flow through the cascade of arbitrary foil 25
152 Experimental study of the effects of viscosity and viscoelasticity on a line vortex cavitation 25
153 Numerical modeling of tip vortex cavitation modification using polymer solutions 25
154 Influence of propeller presence and cavitation on a liquid nuclei population 25
155 Cavitation in metastable nanoconfined fluids 25
156 Development of measurement techniques for studying propeller erosion damage in severe wake fields 26
157 Model for the oscillations of the shell of a contrast agent, liquid and solid cases 26
161 Prediction research on cavitation performance for centrifugal pumps 26
164 Observations and numerical simulations of unsteady partial cavitation on 2-D hydrofoil 27
166 Precursor luminescence near the collapse of laser-induced bubbles in alkali-salt solutions 27
167 Gas bubble growth dynamics in a supersaturated solution: henry's and sievert's solubility laws 27
169 Physical - mathematical bases of the principle of independence of cavity expansion 27
172 Shock propagation in polydisperse bubbly flows 27
173 Numerical study on the surface stability of an encapsulated microbubble in the ultrasound field 28
175 Design of cavitation-free hydrofoils by a given pressure envelope 28
177 Damage potential of the shock-induced collapse of a gas bubble 28
179 Cavitation phenomena in a stagnation point flow 28
180 On some physics to consider in numerical simulation of erosive cavitation 29
182 Experimental investigations of flow structure and turbulence associated with vanishing propeller tip vortex cavitation 29

183 Incepting cavitation acoustic emissions due to vortex stretching 29

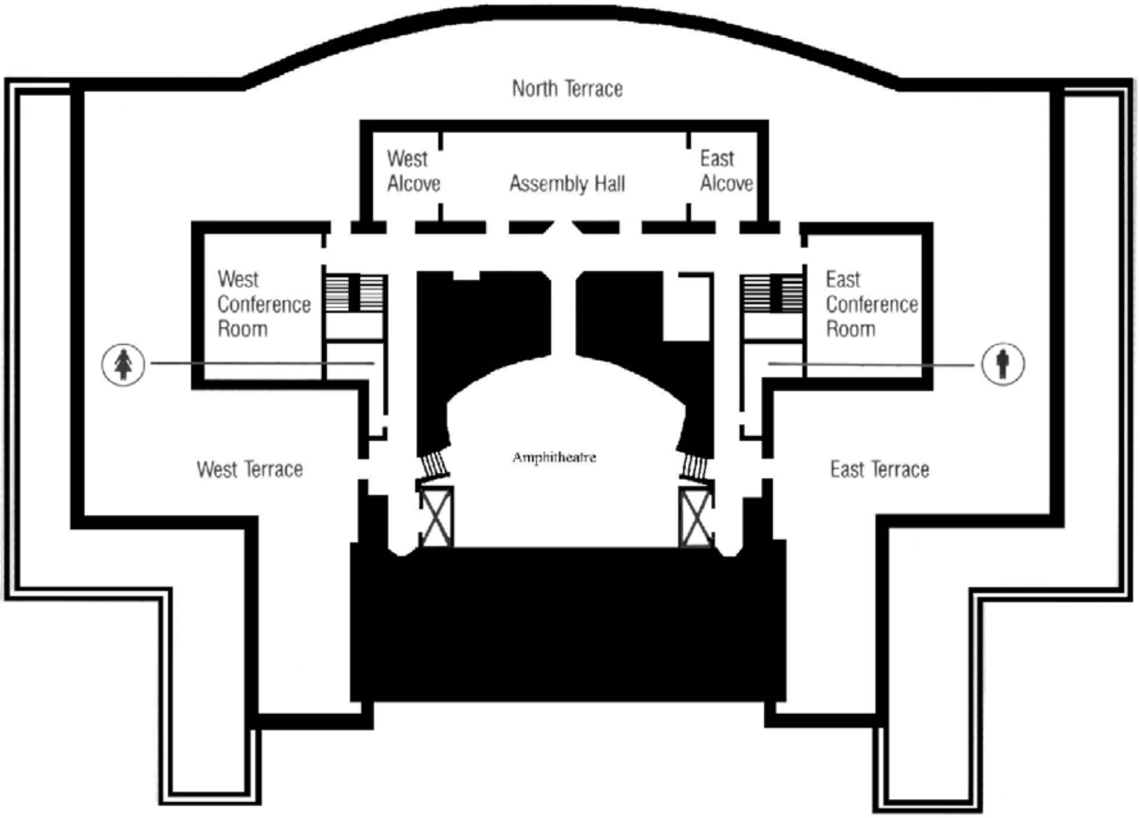
Name	Paper Number	Page Number
Abdel-Maksoud	139, 143	23, 24
Aeschlimann	38	6
Afanasiev	127	21
Ahuja	122	20
Amromin	8,112	1,18
An	90	15
Ando, J	30	4
Ando, K	172	27
Antipov	21	3
Apte	73, 74	13
Aristoff	125	21
Arndt	110,111,112,113,146	18,19,24
Arzoumanian	145	24
Aschenbrenner	91	15
Atalr	135	22
Aubry	134	22
Avkhadiev	175	28
Bakaya	18	3
Balas	110	18
Barbier	152	25
Bark	180	29
Barre	38	6
Beal	100	16
Bensow	180	29
Berchiche	180	29
Beux	42	7
Bilanceri	42	7
Boorsma	13	2
Bosschers	46	8
Brennen	172	27
Brizzolara	79	14
Bulten	49	8
Ceccio	149, 183	24, 29
Celik	179	28
Cervone	92	15
Chahine	152, 153, 154	25
Challier	29	4
Chamberlin	145	24
Chang	149, 183	24,29
Chow	182	29
Chu	166	27
Cleveland	177	28
Colonius	172, 177	27, 28
Conti	35	5
D'agostino	92	15
Delale	18	3
Dijkink	103, 117	17,20
Donnelly	156	26
Dorofeeva	124	20
Dunn	124	20
Egashira	141	23
Ehrlich	11	2
Ezzatneshan	138	23
Ferber	57	9
Ferrando	35	5
Fink	129, 134	21, 22
Finn	74	13
Fitzsimmons	13,41	2,6
Foeth	41	6
Fry	156	26
Fu	71	13
Fujii	68	12
Fujikawa	7	1
Fukaya	15	2
Furukawa	40	6
Gaggero	79	14
Ganesh	149	24
Gateau	129, 134	21, 22
Goncalves	29	4
Gonzalez	103	17
Gopalan	179	28
Gor	167	27
Gowing	156	26
Grekula	107, 180	17, 29
Grigorieva	127	21
Gunalan	117	20

Ha	**99**	*16*
Hachmann	**139**	*23*
Hamadeh	**58**	*10*
Han	**27**	*4*
Hänel	**139**	*23*
Harada	**65**	*12*
Hasegawa	**36**	*5*
Hasuike	**30**	*4*
Hattori	**32, 33, 64**	*4, 5,11*
Hejranfar	**87, 138**	*14, 23*
Hesary	**87, 138**	*14, 23*
Hirose	**33**	*5*
Hjartarson	**110**	*18*
Hoeijmakers	**52**	*8*
Hoekstra	**43**	*7*
Horiguchi	**50**	*8*
Hosangadi	**122**	*20*
Houlin	**161**	*26*
Hsiao	**153,154**	*25*
Hsieh	**166**	*27*
Huang, B	**9**	*1*
Huang, X	**103**	*17*
Huang, Z	**68**	*12*
Hundemer	**143**	*24*
Iga	**67, 78**	*12,14*
Ikohagi	**67, 78**	*12,14*
Inaba	**109**	*18*
Ito	**61**	*11*
Itoh	**142**	*24*
Jessup	**156**	*26*
Jinbo	**53**	*9*
Johnsen	**177**	*28*
Jung	**27**	*4*
Kai	**161**	*26*
Kajishima	**89, 90**	*15*
Kanagawa	**7**	*1*
Kang	**50**	*8*
Katz	**179**	*28*
Kawakami	**110, 111, 112**	*18*
Kawamura	**68**	*12*
Kawata	**50**	*8*
Khoo	**98,104**	*16,17*
Kim, H	**70**	*13*
Kim, S	**56**	*9*
Kimura	**68**	*12*
Kinnas	**94,97**	*16*
Kinzel	**136, 137**	*22, 23*
Kirschner	**72, 145**	*13, 24*
Kjeldsen	**113**	*19*
Klaseboer	**98, 104**	*16, 17*
Kobayashi	**53**	*9*
Kodama	**109**	*18*
Koop	**52**	*8*
Koshi	**102**	*17*
Kubota	**37**	*5*
Kuchma	**167**	*27*
Kuiper	**114**	*19*
Kumaraswamy	**22**	*4*
Kuni	**167**	*27*
Kunz	**136, 137**	*22, 23*
Lantermann	**139**	*23*
Lee, C	**70**	*13*
Lee, H	**70**	*13*
Lee, S	**27**	*4*
Lee, Y	**182**	*29*
Lele	**62**	*11*
Li, D	**107**	*17*
Li,Z	**41**	*6*
Lindau	**136, 137**	*22, 23*
Lindell	**107**	*17*
Liu	**173**	*28*
Lu	**179**	*28*
Ludwig	**58**	*10*
Makasyeyev	**116**	*19*
Makhrov	**130**	*21*
Maklakov	**175**	*28*
Marsac	**134**	*22*
Maršík	**120**	*20*
Martio	**45**	*7*
Matsumoto	**15, 102, 73**	*2, 17,28*
Mendez	**157**	*26*
Merkle	**99**	*16*

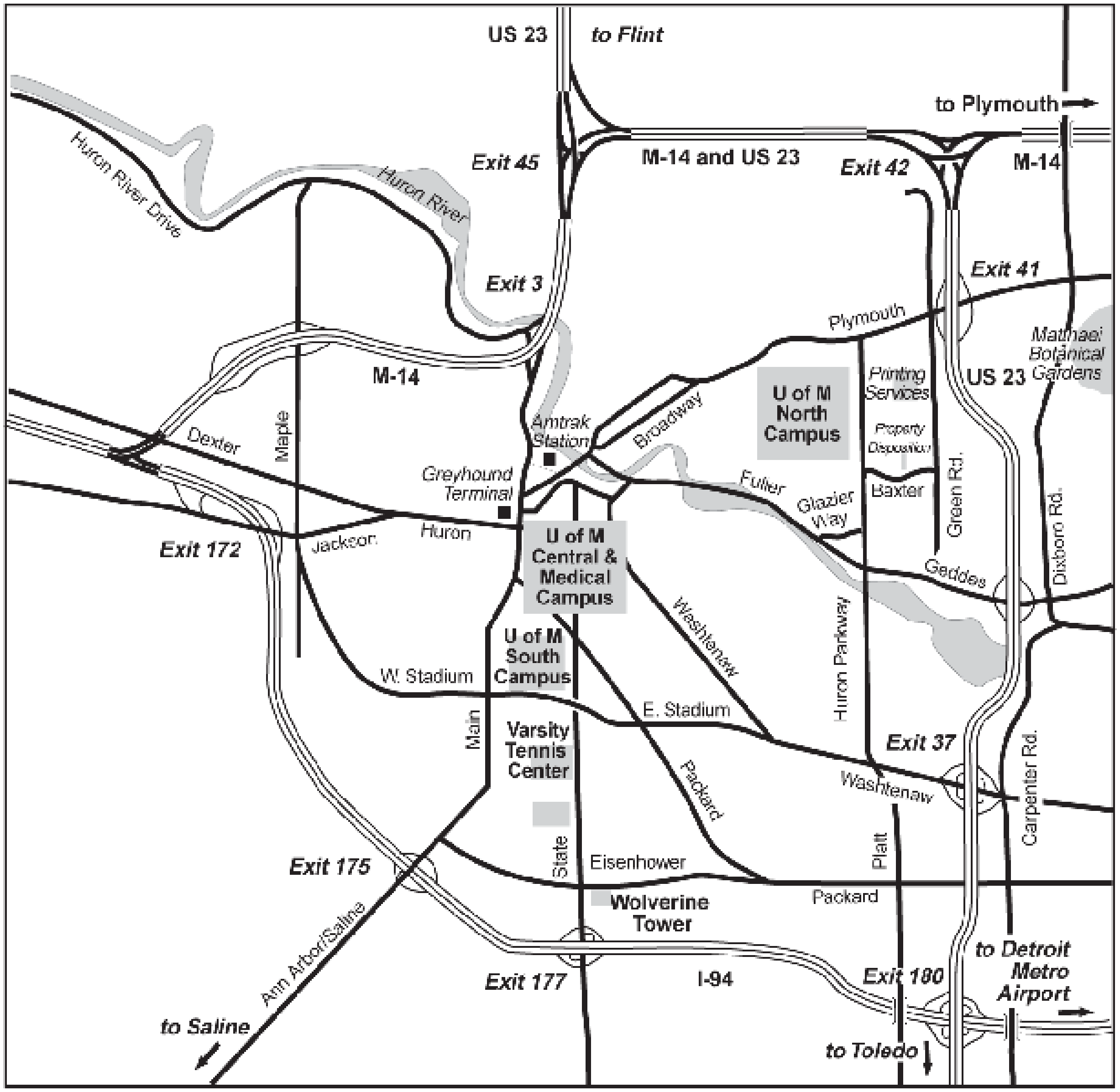
Mikami	**12**	*2*
Minggao	**161**	*26*
Mizuyama	**64**	*11*
Mokhtarzadeh	**110**	*18*
Mørch	**5**	*1*
Moser	**91**	*15*
Motley	**60**	*10*
Müller	**131**	*21*
Murakami	**65**	*12*
Murase	**64**	*11*
Němec	**120**	*20*
Nagasaki	**61**	*11*
Nagaya	**64**	*11*
Nagayama	**61**	*11*
Naude	**157**	*26*
Necker	**91**	*15*
Nesteruk	**86**	*14*
Niiyama	**36**	*5*
Nishizawa	**12**	*2*
Nohmi	**67**	*12*
Ochiai	**67,78**	*12,14*
Oh	**70**	*13*
Ohjimi	**66**	*12*
Ohl, C	**103,117**	*17,20*
Ohl, S	**98, 104**	*16, 17*
Ohshima	**141**	*23*
Oike	**36**	*5*
Okabayashi	**89, 90**	*15*
Oprea	**49**	*8*
Otani	**32**	*4*
Paik	**16**	*3*
Park	**99**	*16*
Pasini	**92**	*15*
Patella	**29**	*4*
Pavard	**98**	*16*
Pelz	**57, 58**	*9,10*
Peng	**164**	*27*
Pernot	**129, 134**	*21, 22*
Pfitsch	**156**	*26*
Pouffary	**29**	*4*
Prabowo	**103, 117**	*17, 20*
Preiser	**117**	*20*
Quinto-Su	**103, 117**	*17, 20*
Raju	**154**	*25*
Rammohan	**22**	*4*
Reksoprodjo	**45**	*7*
Rezaee	**132**	*22*
Rhee	**70**	*13*
Rolland	**29**	*4*
Saisto	**45**	*7*
Salvetti	**42**	*7*
Sampson	**135**	*22*
Sasaki	**135**	*22*
Saseendran	**22**	*4*
Sato	**37, 66, 141**	*5,12,23*
Savander	**59**	*10*
Savchenko	**48**	*8*
Savio	**35**	*5*
Schmidt	**17,18**	*3*
Schnerr	**17, 18,72**	*3, 13*
Schwille	**11**	*2*
M. Sedlář	**131**	*21*
Semenov	**48**	*8*
Seo, J	**62**	*11*
Seo, T	**37**	*5*
Serebryakov	**72, 169**	*13, 27*
Shabgard	**132**	*22*
Shams	**73, 74**	*13*
Shervani-Tabar	**132**	*22*
Shi	**142**	*24*
Shin	**70**	*13*
Shouqi	**161**	*26*
Shyy	**77**	*14*
Siikonen	**45**	*7*
Singh	**94**	*16*
Sipilä	**45**	*7*
Soyama	**12**	*2*
Stoffel	**58**	*10*
Sugimoto	**66**	*12*
Sugiyama, K	**33**	*5*
Sugiyama, K	**173**	*28*
Suh	**70**	*13*

Takagi	**173**	*28*
Takahira	**53**	*9*
Takano	**141**	*23*
Taketani	**68**	*12*
Takeuchi	**102**	*17*
Takinami	**32**	*4*
Tamura, T	**36**	*5*
Tamura, Y	**15**	*2*
Tani	**63**	*11*
Tanter	**129, 134**	*21, 22*
Techet	**100**	*16*
Terentiev	**151**	*25*
Thalhamer	**17**	*3*
Thomas	**124**	*20*
Tomita	**109**	*18*
Torre	**92**	*15*
Truscott	**100**	*16*
Tsai	**182**	*29*
Tseng	**77**	*14*
Tsuda	**36,63,102**	*5,11,17*
Tsujimoto	**40, 50**	*6, 8*
Uchikoshi	**109**	*18*
Ungewitter	**122**	*20*
Van Terwisga	**41**	*6*
Van Wie	**179**	*28*
Vaz	**43**	*7*
Venugopalan	**103**	*17*
Vinayan	**97**	*16*
Viviani	**35**	*5*
Vo	**166**	*27*
Wang	**9**	*1*
Watanabe	**7**	*1*
Watanabe	**40**	*6*
Williams	**111, 112**	*18*
Wosnik	**146**	*24*
Wu	**103**	*17*
Yakushiji	**149**	*24*
Yamanishi	**63, 102**	*11, 17*
Yamasaki	**30**	*4*
Yano	**7**	*1*
Yonezawa	**50**	*8*
Yong	**161**	*26*
Yoshida	**36, 40, 63, 78**	*5,6,11,14*
Yoshimura	**37**	*5*
Young	**59, 60**	*10*
Zabihyan	**132**	*22*
Zemlyanova	**21**	*3*
Zeng	**114**	*19*
Zhang, B	**9**	*1*
Zhang, L	**71**	*13*
Zhang, M	**9**	*1*
Zhang, Q	**153**	*25*
Zima	**131**	*21*

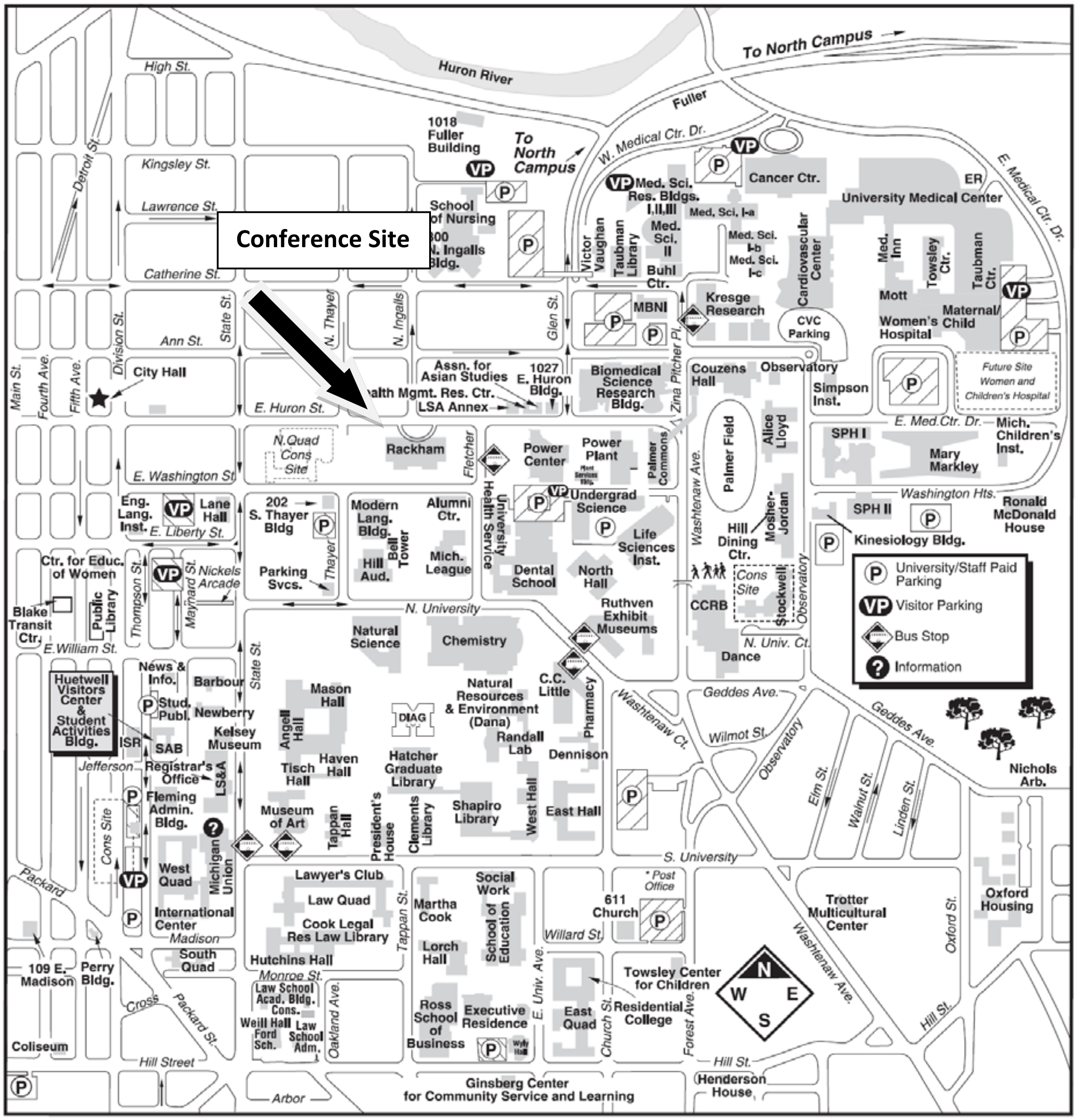
Rackham Building 4th floor



Ann Arbor


US 23
to Flint
to Plymouth
Exit 45
M-14 and US 23
Exit 42
M-14
Huron River Drive
Huron River
Exit 3
Exit 41
Plymouth
Matthaei Botanical Gardens
M-14
U of M North Campus
Printing Services
US 23
Property Disposition
Amtrak Station
Broadway
Dexter
Maple
Greyhound Terminal
Fuller
Glazier Way
Baxter
Green Rd.
Dixboro Rd.
Exit 172
Jackson
Huron
U of M Central & Medical Campus
Geddes
Washtenaw
Huron Parkway
U of M South Campus
W. Stadium
E. Stadium
Exit 37
Main
Varsity Tennis Center
Packard
Washtenaw
Carpenter Rd.
Exit 175
State
Eisenhower
Platt
Packard
Wolverine Tower
Ann Arbor/Saline
Exit 177
I-94
Exit 180
to Detroit Metro Airport
to Saline
to Toledo

University of Michigan
Central Campus

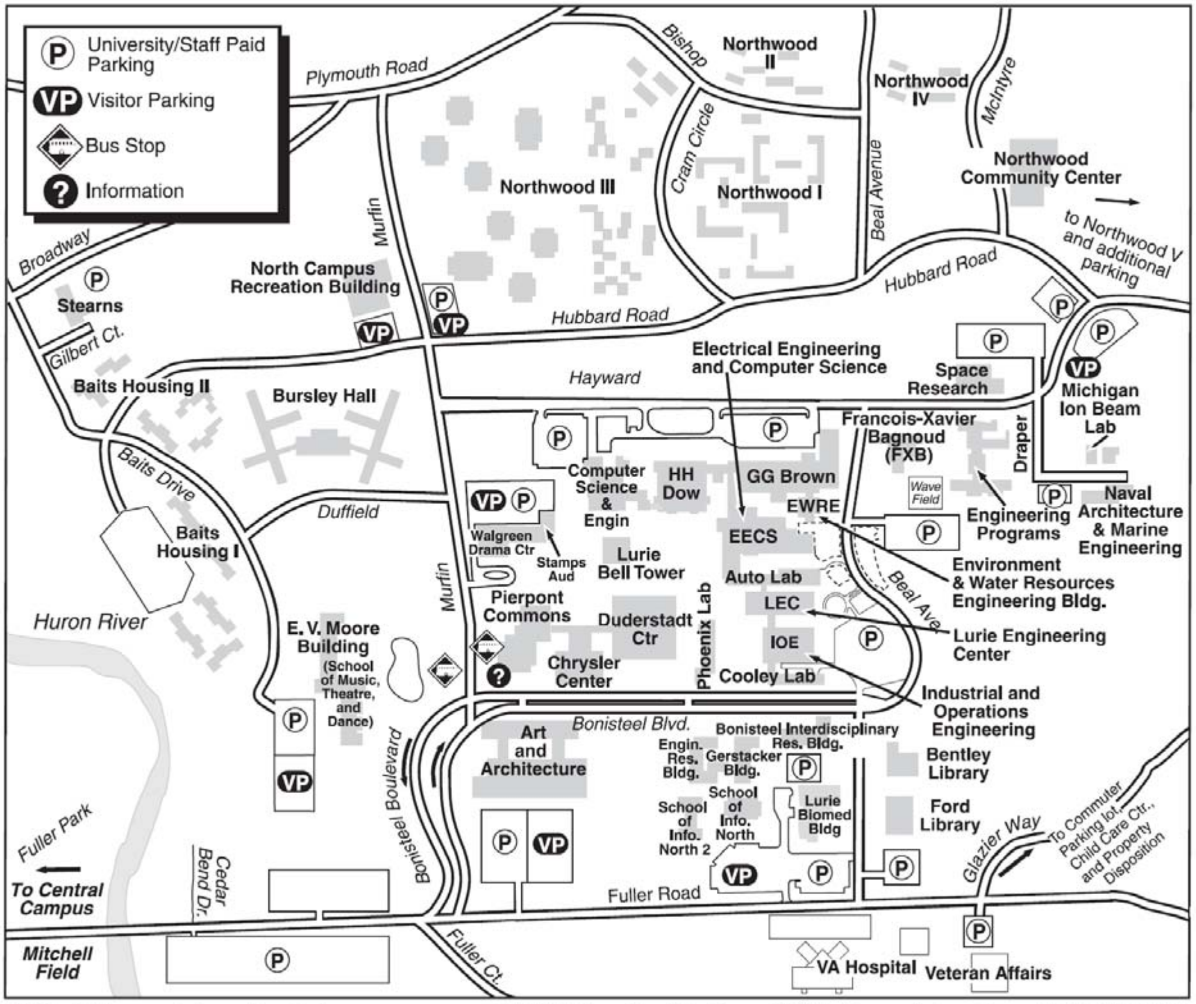


To Wolverine Tower and I-94 ↓

? Campus Information Center, (First floor, Michigan Union, 764-INFO)

© 2009 University of Michigan, MM&D.
For use information call 734-764-9270.

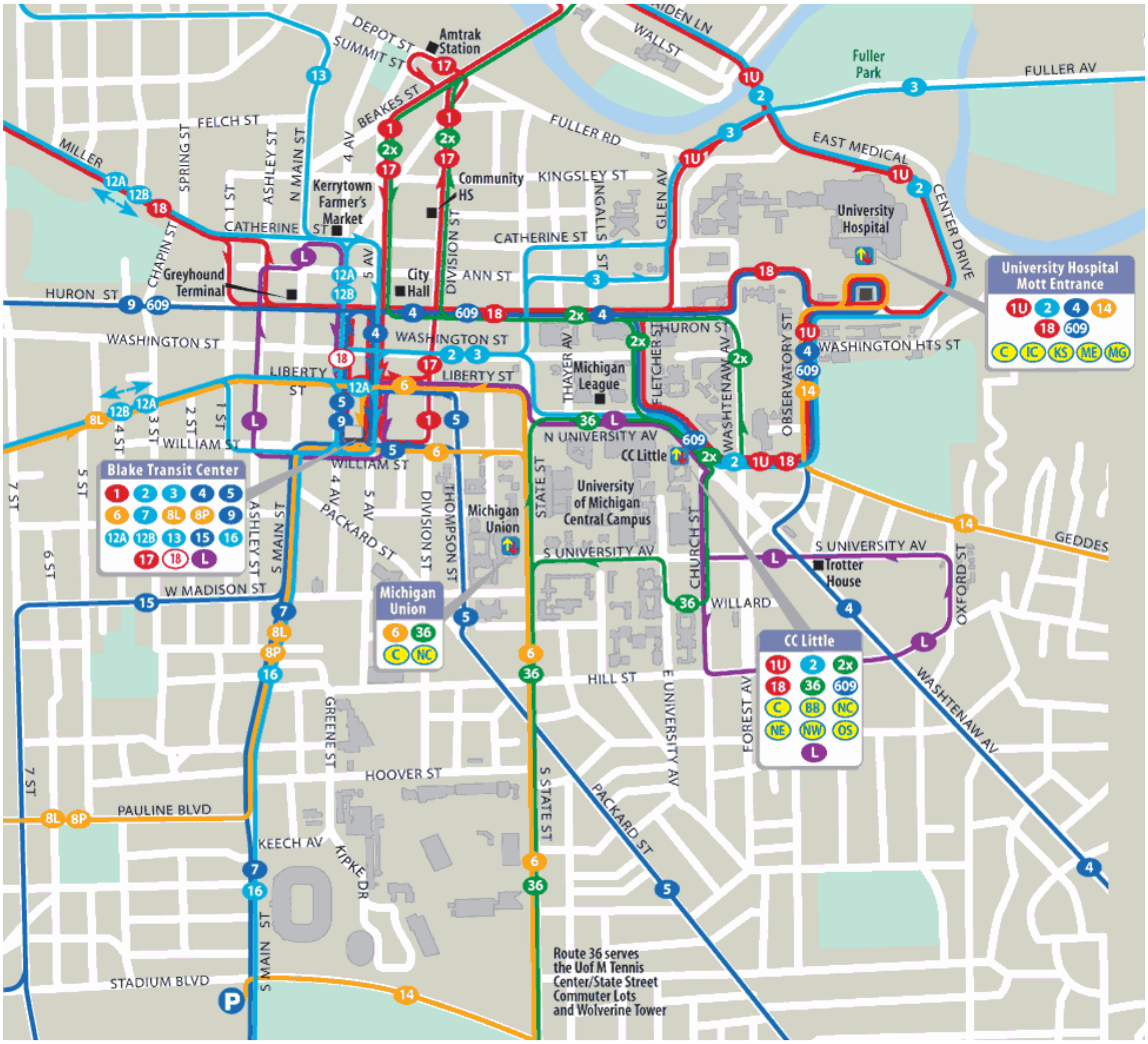
University of Michigan
North Campus



❓ *North Campus Information Center, (Lobby, Pierpont Commons, 764-INFO)*

Parking and Commuting

Rackham building is located at 915 E Washington Street, Ann Arbor. There are many public parking structures near Rackham building. Please refer to the central campus map attached above. Commuting can also be done by public transport if your hotel is accessible by Public transport. Attached is the map of Ann Arbor transportation authority (AATA.org) route map. The cost is $1 per ride.



Contact Organising Committee

In case of emergencies or concern please contact the organizing committee.

Harish Ganesh
2010, Walter E Lay Auto Lab,
1231, Beal Avenue, Ann Arbor
48109

Tel 734 709 7280
gharish@umich.edu