

**CFD ANALYSIS OF PUMP CONSORTIUM IMPELLER**

Gary C. Cheng\*, Y.S. Chen†, and R.W. Williams‡

Abstract

Current design of high performance turbopumps for rocket engines requires effective and robust analytical tools to provide design impact in a productive manner. The main goal of this study is to develop a robust and effective computational fluid dynamics (CFD) pump model for general turbopump design and analysis applications. A Navier-Stokes flow solver, FDNS, embedded with the extended k- $\epsilon$  turbulence model and with appropriate moving interface boundary conditions, is developed to analyze turbulent flows in the turbomachinery devices. The FDNS code has been benchmarked with its numerical predictions of the pump consortium inducer, and provides satisfactory results. In the present study, a CFD analysis of the pump consortium impeller will be conducted with the application of the FDNS code. The pump consortium impeller, with partial blades, is the new design concept of the advanced rocket engine. A 3-D flow calculation with 81 x 41 x 41 grid system was conducted for the team base-line impeller. The result shows a massive flow separation occurs between the full-blade pressure surface and the partial-blade suction surface. Similar result was predicted by the other consortium members. A pump consortium optimized impeller, a revision based on the base-line impeller, was then designed by Rocketdyne to remove the flow separation. A 3-D flow analysis, with 103 x 23 x 30 mesh system and with the inlet flow conditions provided by Rocketdyne, was performed for the optimized impeller. The numerical result indicates no flow separation occurs inside the flow passage, which is also consistent with the other consortium members' predictions. However, the flow field inside the optimized impeller as calculated by the team members showed great variations, especially near the exit shroud region. The discrepancy is suspected to be due to different exit boundary conditions used by the consortium members. Therefore, three different exit wall boundary conditions will be further examined by the FDNS code, those are fixed-wall, wall-slip (symmetry), and rotating wall boundary conditions. The computed results will be compared in order to address the effect of exit boundary conditions on the impeller flow field. Meanwhile, two off-design cases of the optimized impeller, 80% and 120% of the design flow, will also be analyzed with a particular exit boundary condition. All CFD analysis of the pump consortium base-line impeller, and the optimized impeller with various exit boundary conditions will be presented in the coming CFD workshop meeting.

---

\* SECA, Inc., 3313 Bob Wallace Ave., Suite 202, Huntsville, AL

† Engineering Sciences, Inc., 4920 Corporate Dr., Suite K, Huntsville, AL

‡ ED 32, NASA/Marshall Space Flight Center, Huntsville, AL

# **CFD ANALYSIS OF PUMP CONSORTIUM IMPELLER**

By

**Gary C. Cheng, SECA, Inc.**

**Y.S. Chen, ESI**

**AND**

**R.W. Williams**

**NASA/Marshall Space Flight Center**

NASA Contract No. NAS8-38868

**TENTH ANNUAL CFD WORKSHOP MEETING, APRIL, 1992**

● **INLET/EXIT WALL B.C. TESTED**

		Exit B.C.		
		Fixed-Wall	Rotating-Wall	Wall-Slip
Inlet B.C.	Fixed-Wall	Case 1	N/A	Case 2
	Rotating-Wall	Case 3	Case 4	Case 5

203

● **CALCULATED MASS FLOW RATE SPLIT**

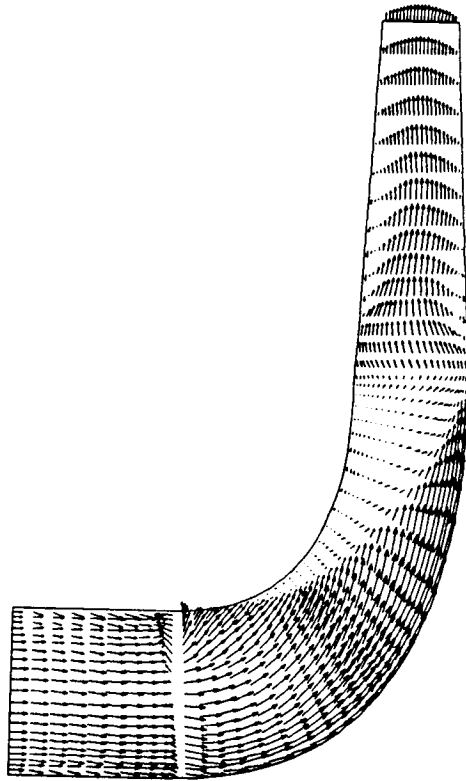
	Case 1	Case 2	Case 3	Case 4	Case 5
S.F. - P.P. /S.P. - P.F.	50.4/49.6	49/51	43.2/56.8	42.4/57.6	40.6/59.4

# DEFINITION OF PERFORMANCE PARAMETERS

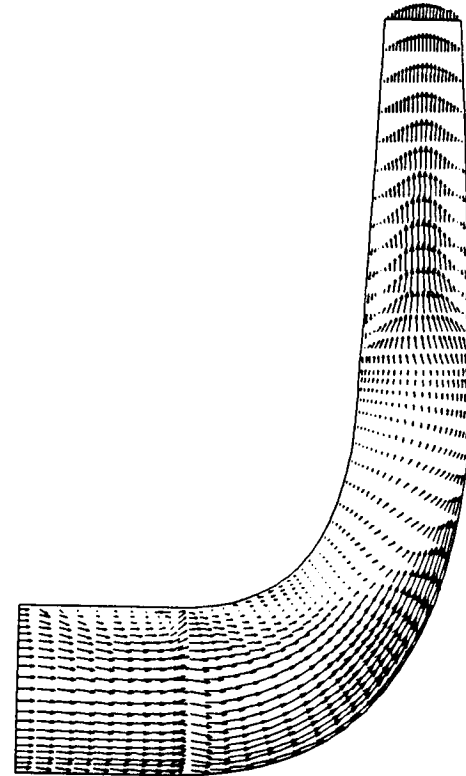
---

---

- $C_u = c_u / U_{tip}$ ;  $C_M = c_M / U_{tip}$  where  $c_u$  = Absolute Tangential Velocity,  
 $c_M$  = Meridional Velocity,  $U_{tip}$  = Wheel Tip Velocity
- $\beta$  = Relative Flow Angle Relative to Tangential Direction
- Relative Radius =  $(R_i - R_{hub}) / (R_{shroud} - R_{hub})$
- Relative X =  $(X_i - X_{shroud}) / (X_{hub} - X_{shroud})$
- Relative Angle =  $(\text{Angle}_i - \text{Angle}_{suction}) / (\text{Angle}_{pressure} - \text{Angle}_{suction})$
- $\Psi$  (Head Coefficient) =  $\Delta H_{tg} / U_{tip}^2$
- $\eta$  (Efficiency) = Head Rise / Euler Head Rise

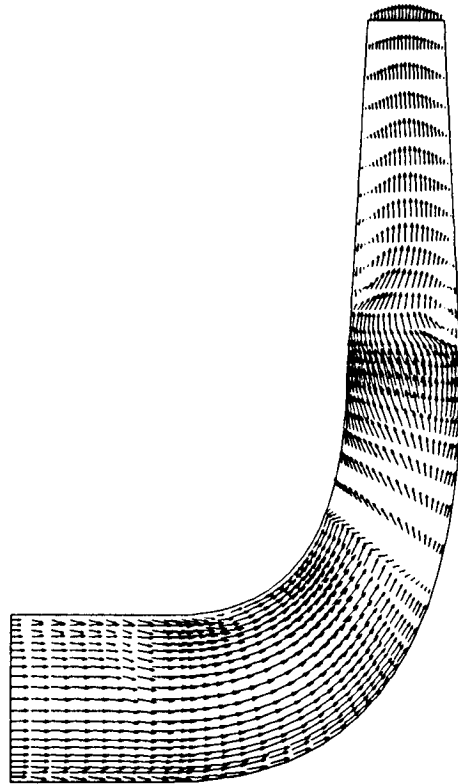


Case 1

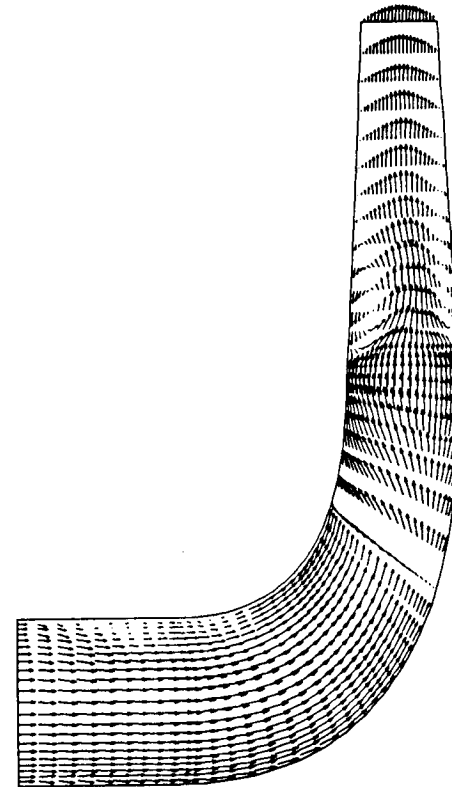


Case 3

**VELOCITY VECTORS NEAR SUCTION SIDE OF BLADE**

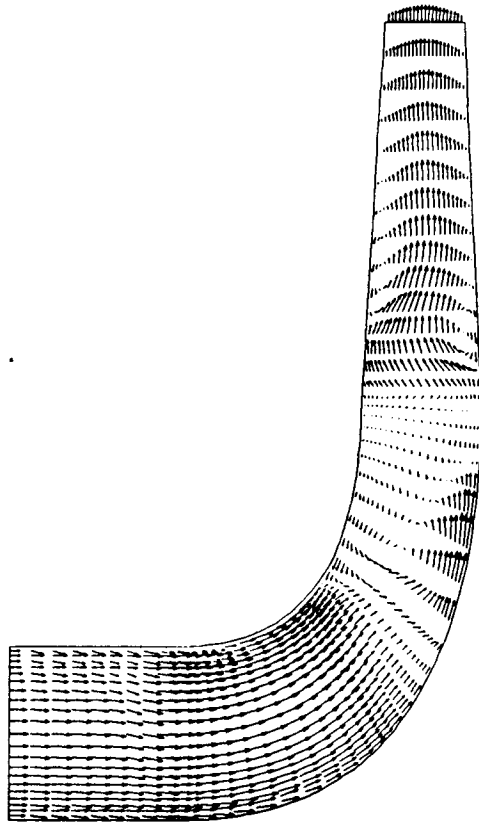


Case 1

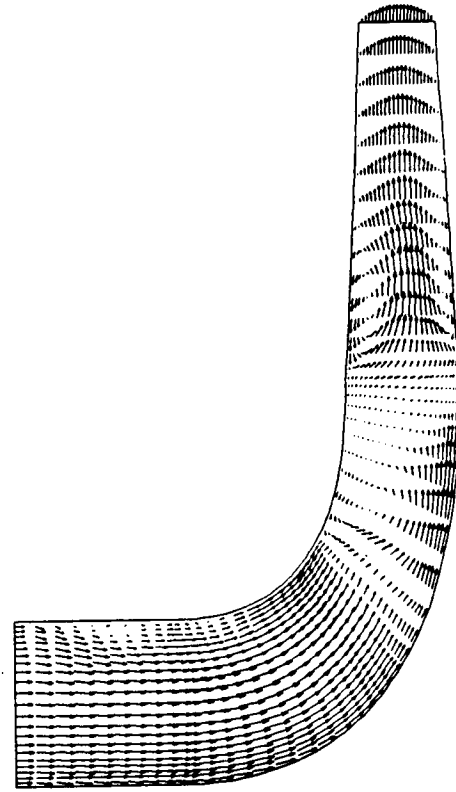


Case 3

**VELOCITY VECTORS NEAR PRESSURE SIDE OF SPLITTER**

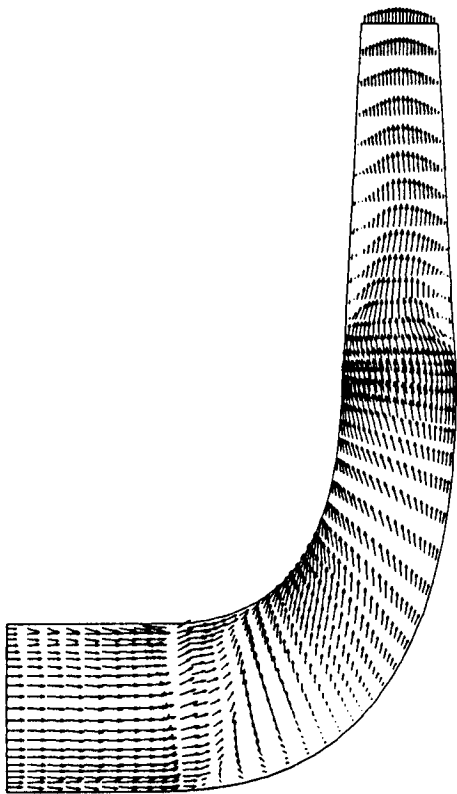


Case 1

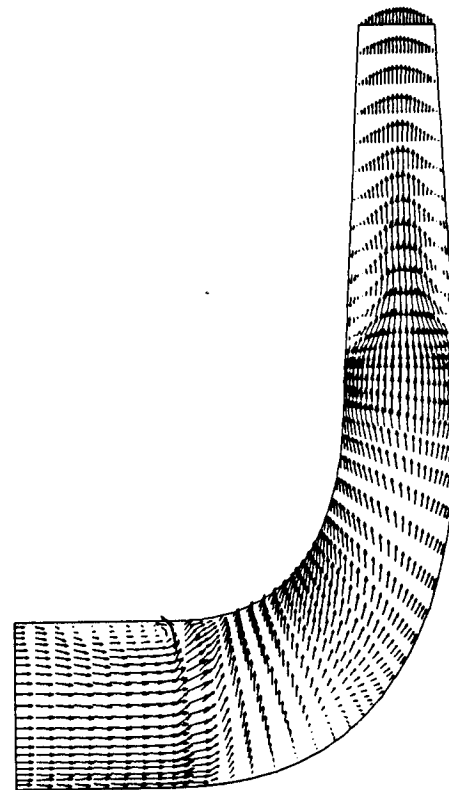


Case 3

**VELOCITY VECTORS NEAR SUCTION SIDE OF SPLITTER**



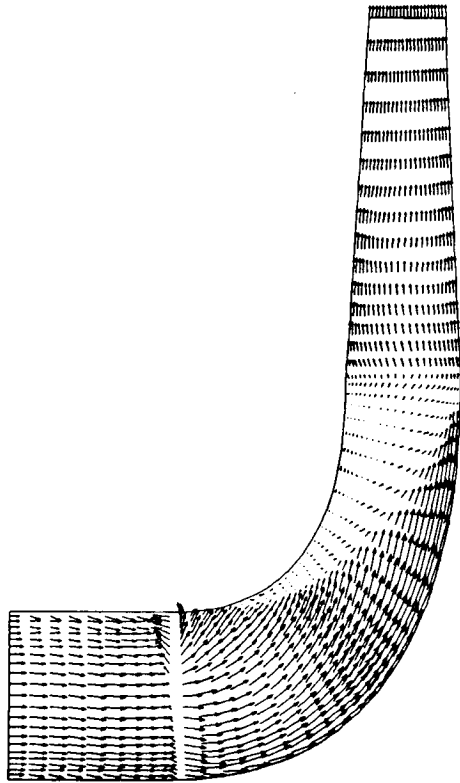
Case 1



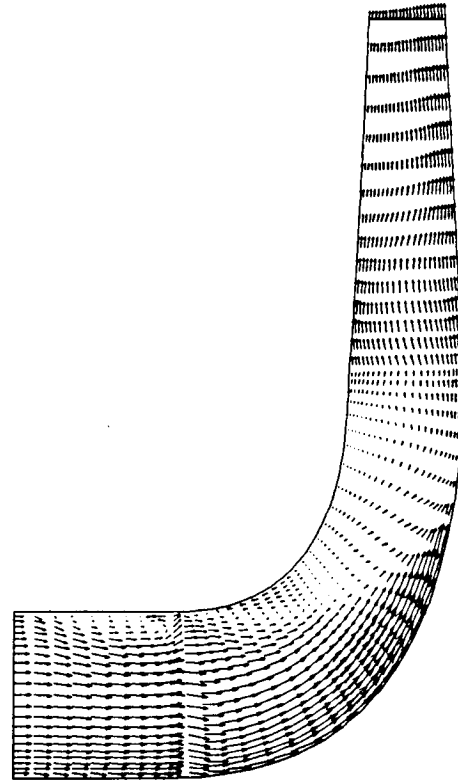
Case 3

**VELOCITY VECTORS NEAR PRESSURE SIDE OF BLADE**



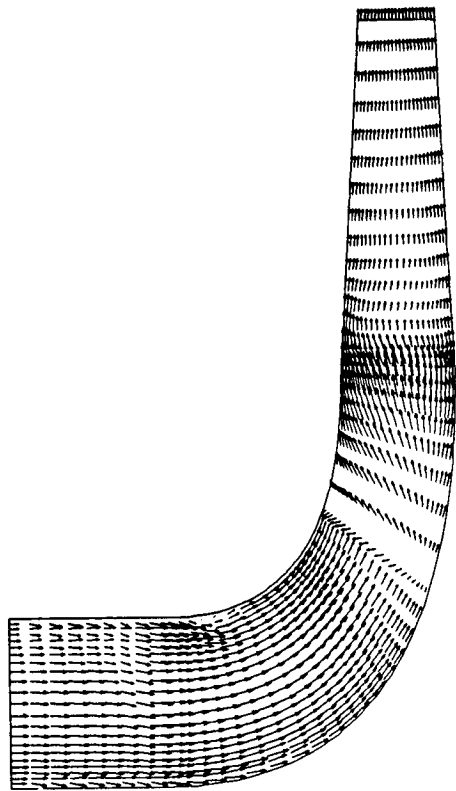


Case 2

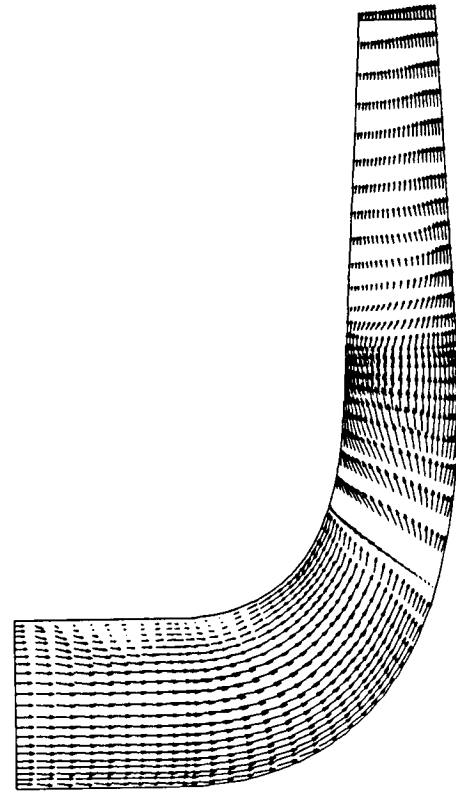


Case 5

**VELOCITY VECTORS NEAR SUCTION SIDE OF BLADE**

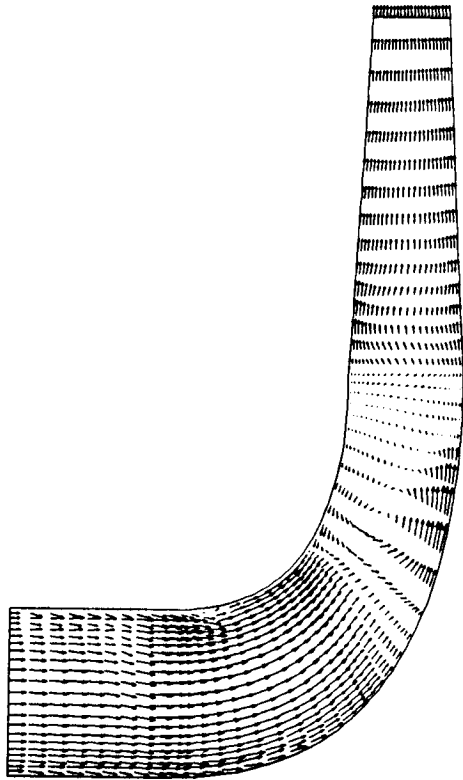


Case 2

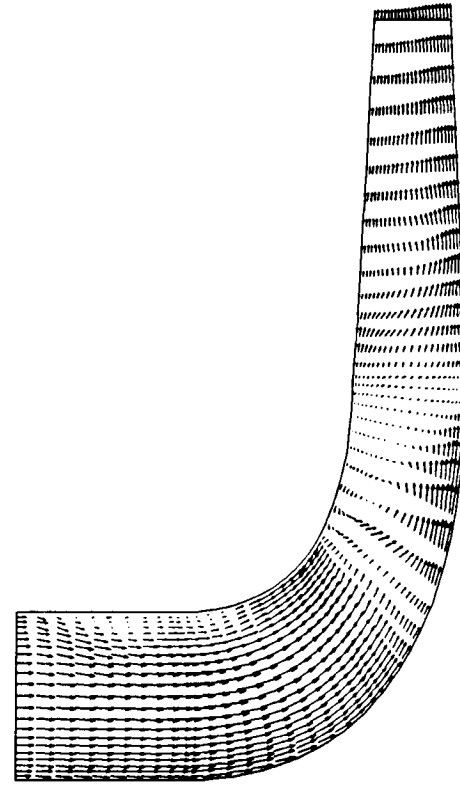


Case 5

**VELOCITY VECTORS NEAR PRESSURE SIDE OF SPLITTER**

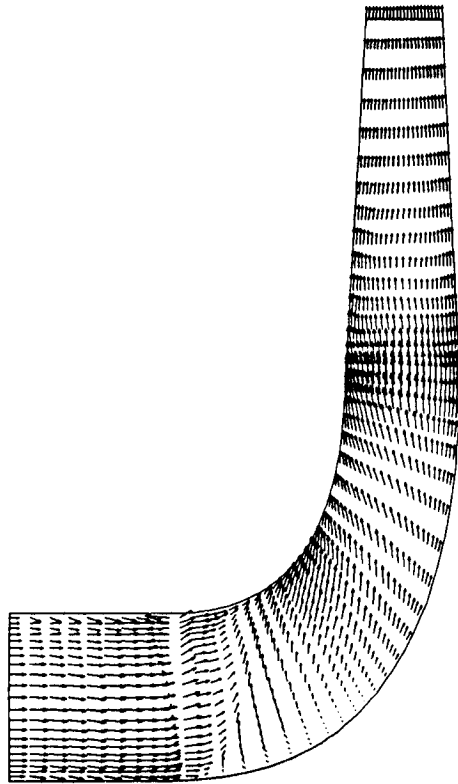


Case 2

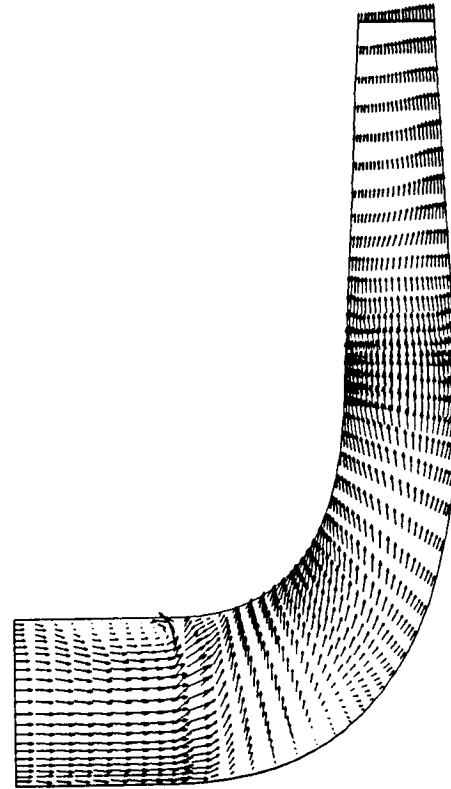


Case 5

**VELOCITY VECTORS NEAR SUCTION SIDE OF SPLITTER**



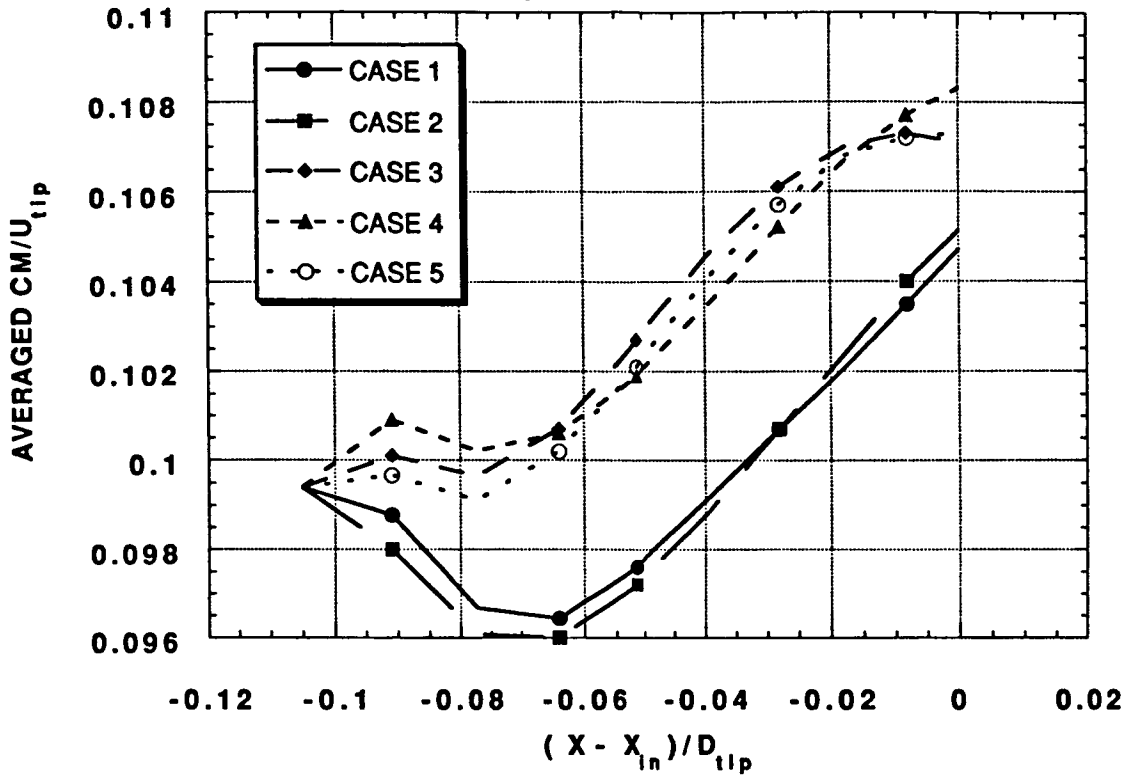
Case 2



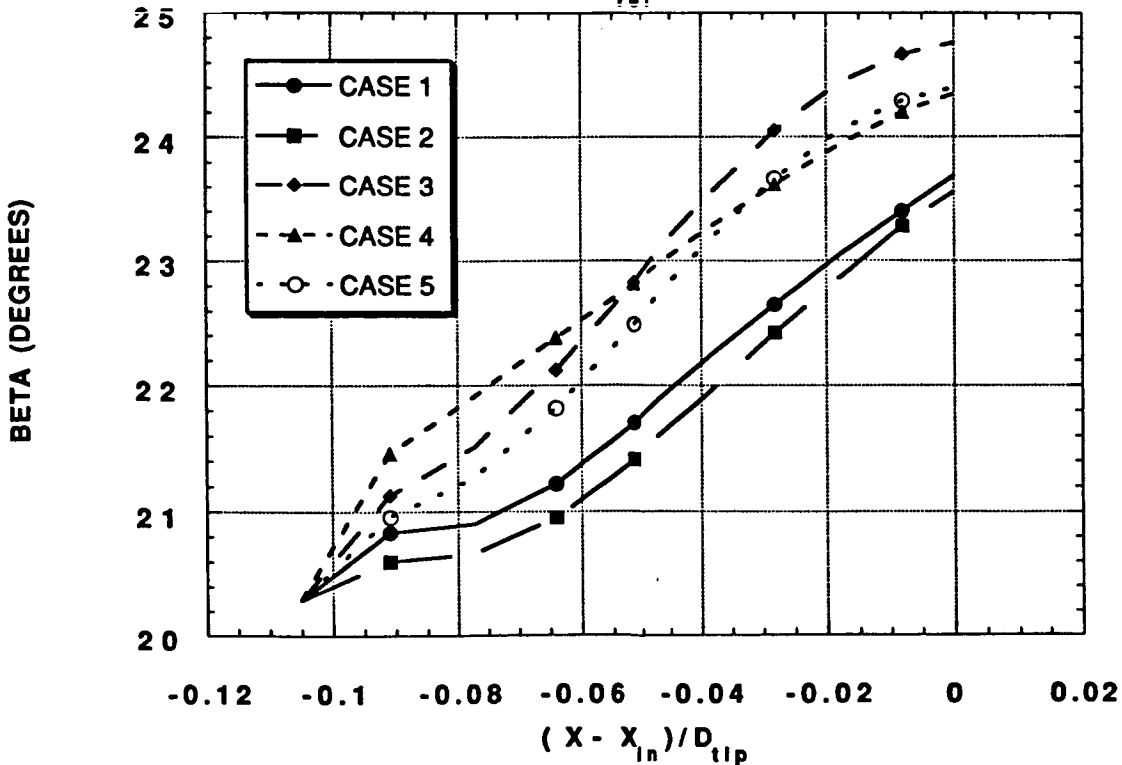
Case 5

**VELOCITY VECTORS NEAR PRESSURE SIDE OF BLADE**

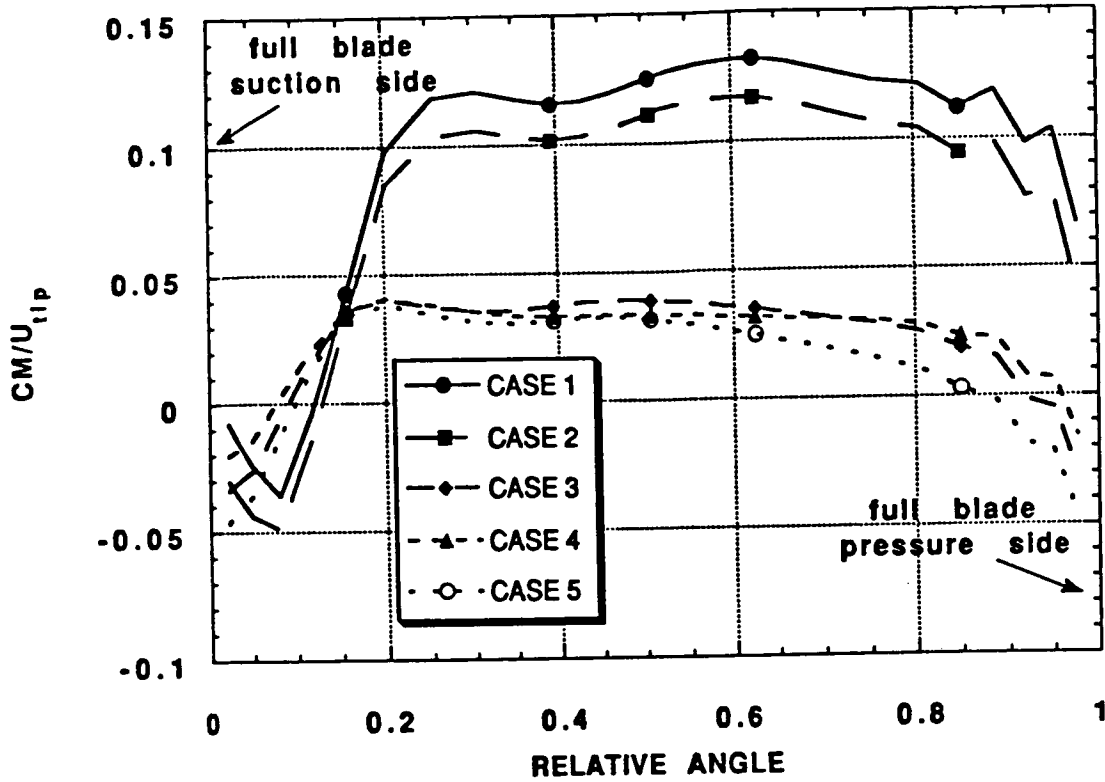
**OPTIMIZED IMPELLER: CM VS. X**  
**FOR  $R_{rel} = 0.05$  (NEAR HUB)**



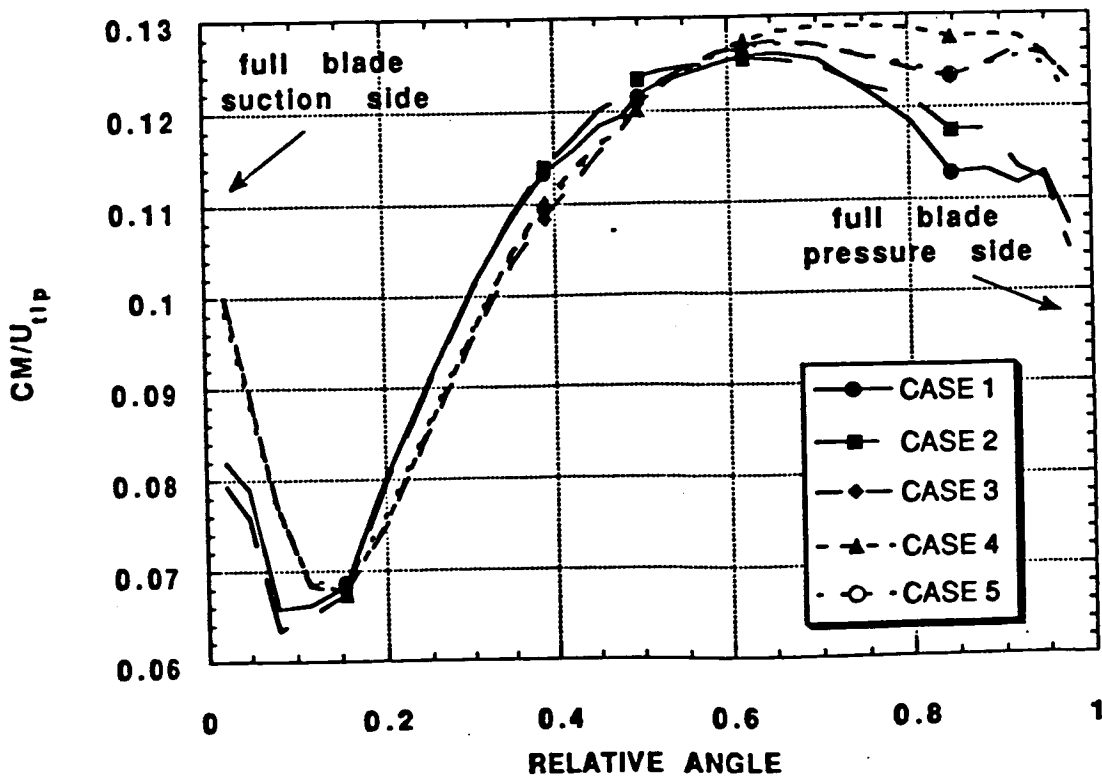
**OPTIMIZED IMPELLER: BETA VS. X**  
**FOR  $R_{rel} = 0.05$**



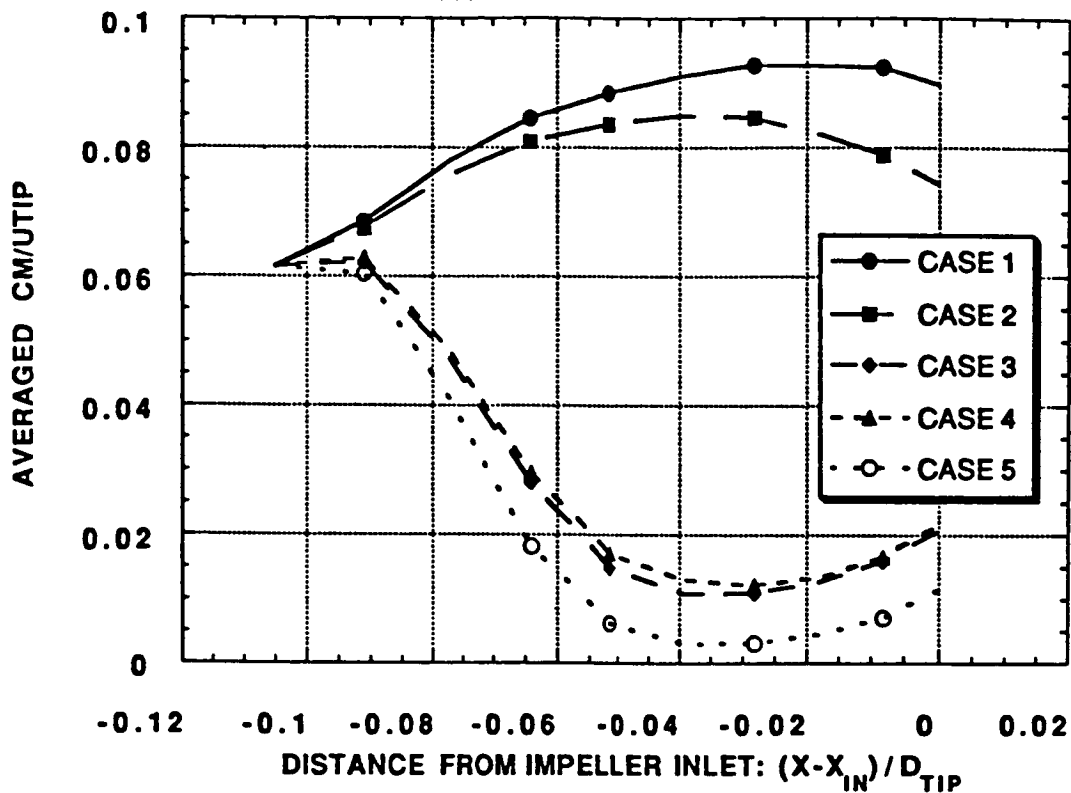
OPTIMIZED IMPELLER: BLADE-TO-BLADE CM  
AT THE IMPELLER INLET FOR  $R_{rel} = 0.95$



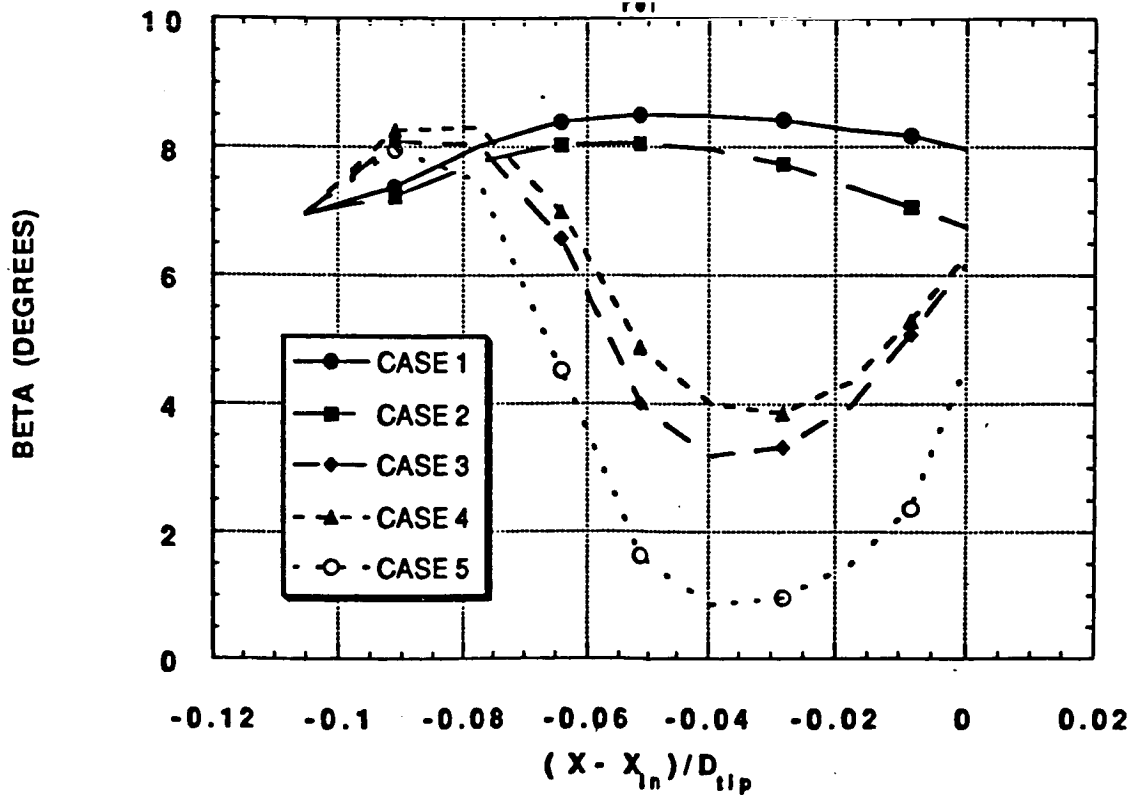
OPTIMIZED IMPELLER: BLADE-TO-BLADE CM  
AT THE IMPELLER INLET FOR  $R_{rel} = 0.05$



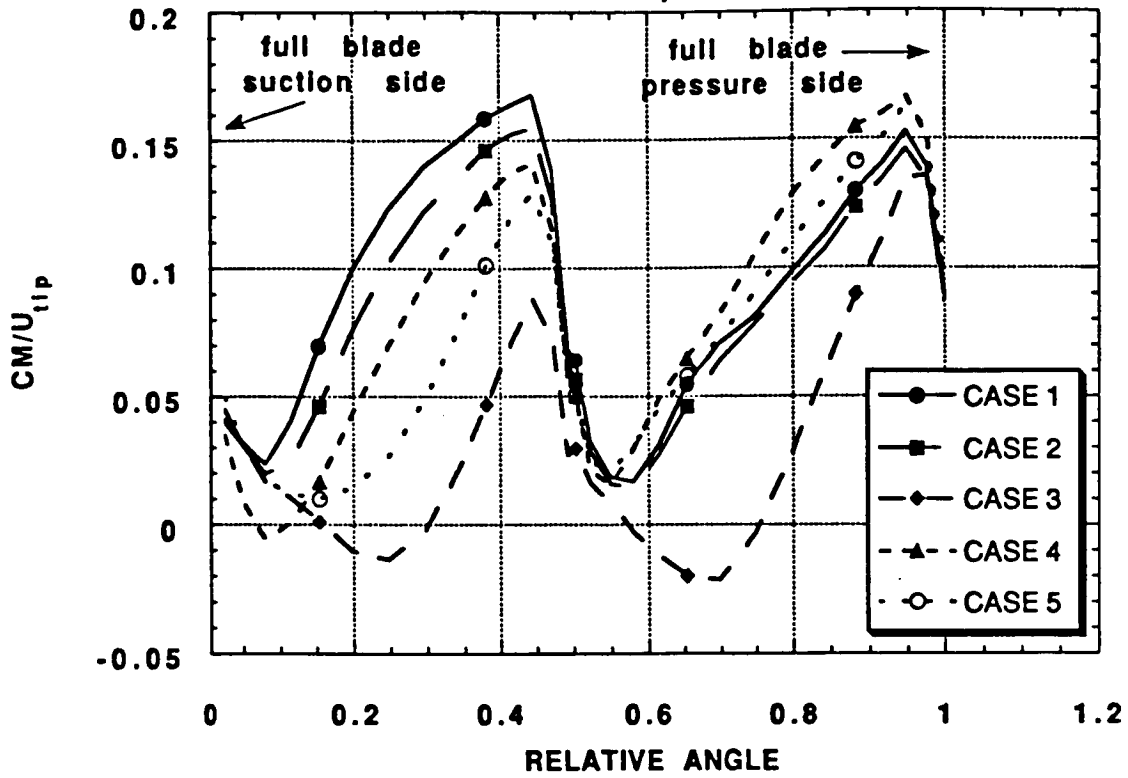
OPTIMIZED IMPELLER: CM VS. X  
FOR  $R_{rel} = 0.95$  (NEAR SHROUD)



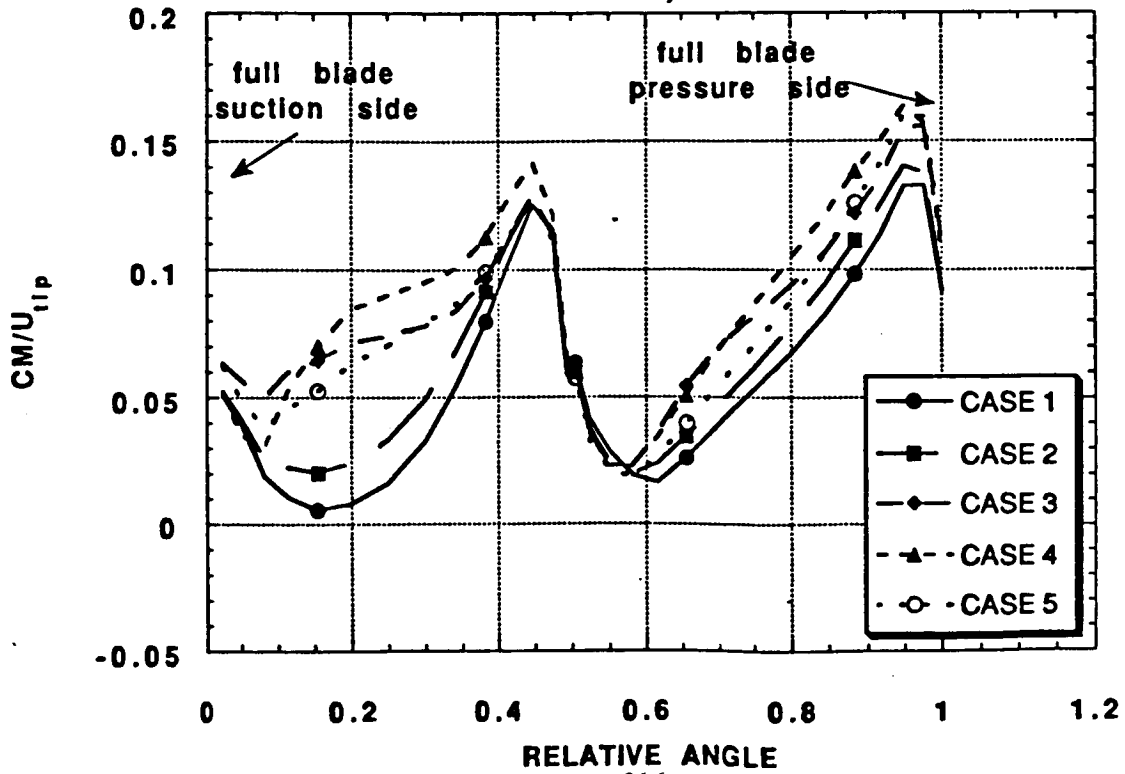
OPTIMIZED IMPELLER: BETA VS. X  
FOR  $R_{rel} = 0.95$



OPTIMIZED IMPELLER: BLADE-TO-BLADE CM  
AT THE IMPELLER EXIT, NEAR THE SHROUD

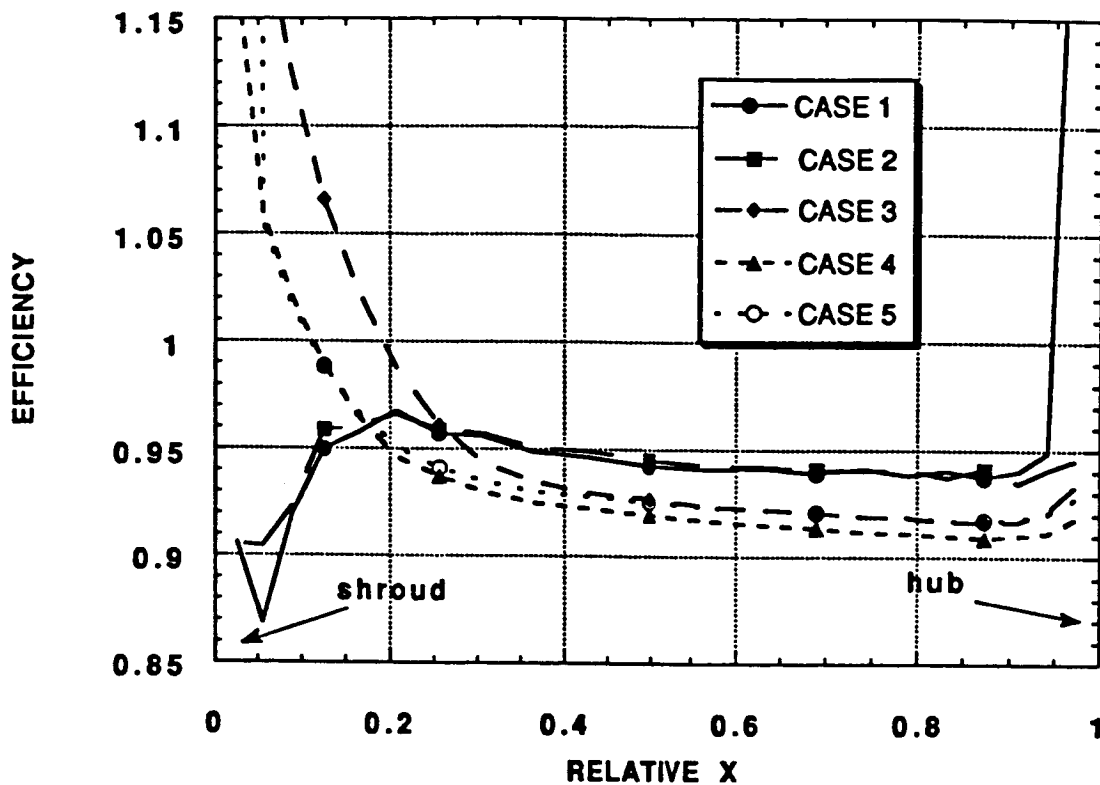


OPTIMIZED IMPELLER: BLADE-TO-BLADE CM  
AT IMPELLER EXIT, NEAR THE HUB

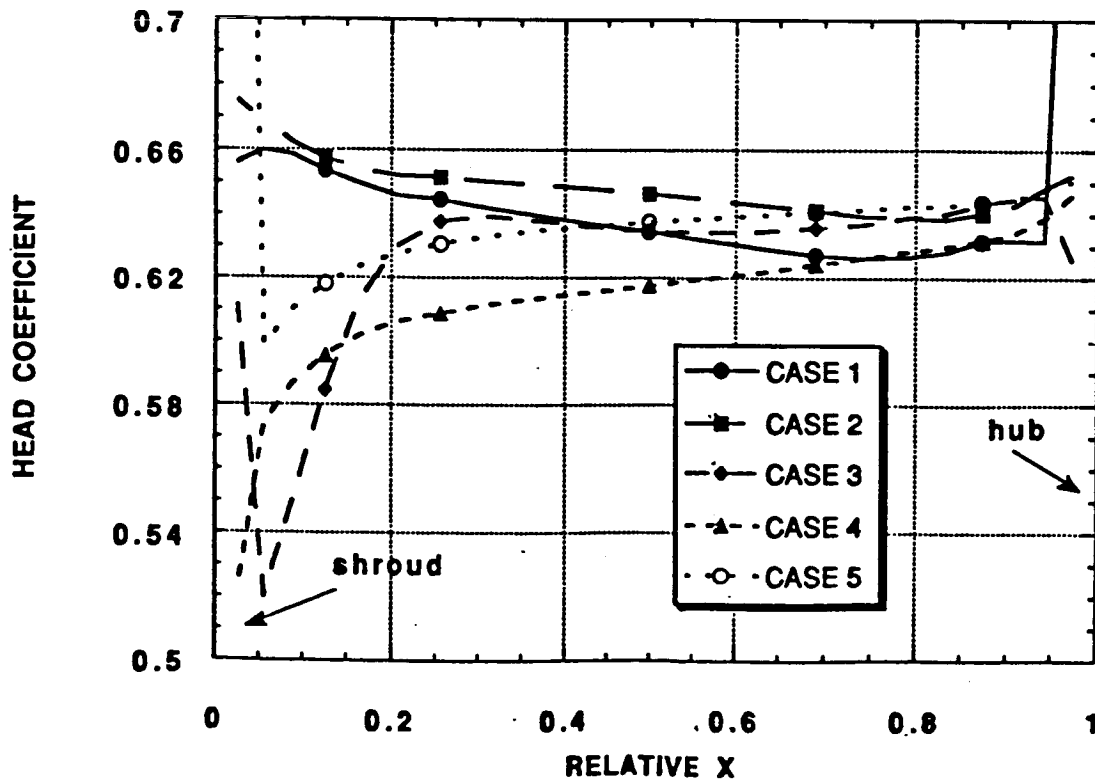




OPTIMIZED IMPELLER:  
EFFICIENCY VS. X



OPTIMIZED IMPELLER:  
HEAD COEFFICIENT VS. X



# CONCLUSIONS

---

- THE PRESENT CFD RESULTS HAVE SHOWN SENSITIVITY OF INLET AND EXIT WALL BOUNDARY CONDITIONS ON THE FLOW STRUCTURE INSIDE THE OPTIMIZED CONSORTIUM IMPELLER DESIGN
- INLET SHROUD WALL BOUNDARY TREATMENTS HAVE SIGNIFICANT EFFECT ON THE FLOW SPLIT AROUND THE PARTIAL BLADE (MORE FLOW THROUGH THE PARTIAL/FULL-PRESSURE PASSAGE WHEN THE INLET SHROUD WALL IS ASSUMED ROTATING)
- ONLY MINOR IMPACT ON THE OVERALL IMPELLER PERFORMANCE DATA WAS REVEALED FOR DIFFERENT BOUNDARY CONDITIONS IMPOSED