N91-10880

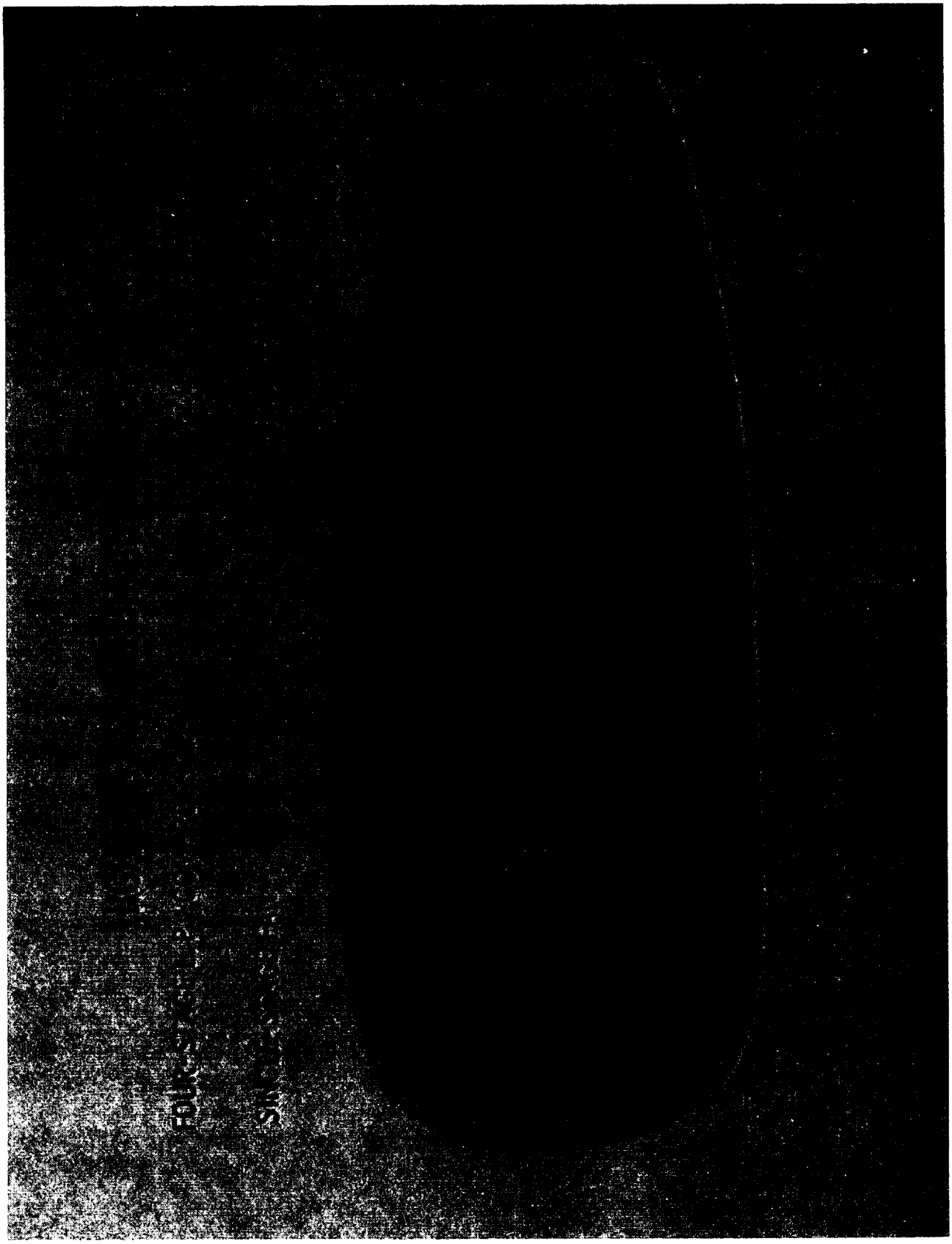
SIMULATION OF TURBOMACHINERY FLOWS

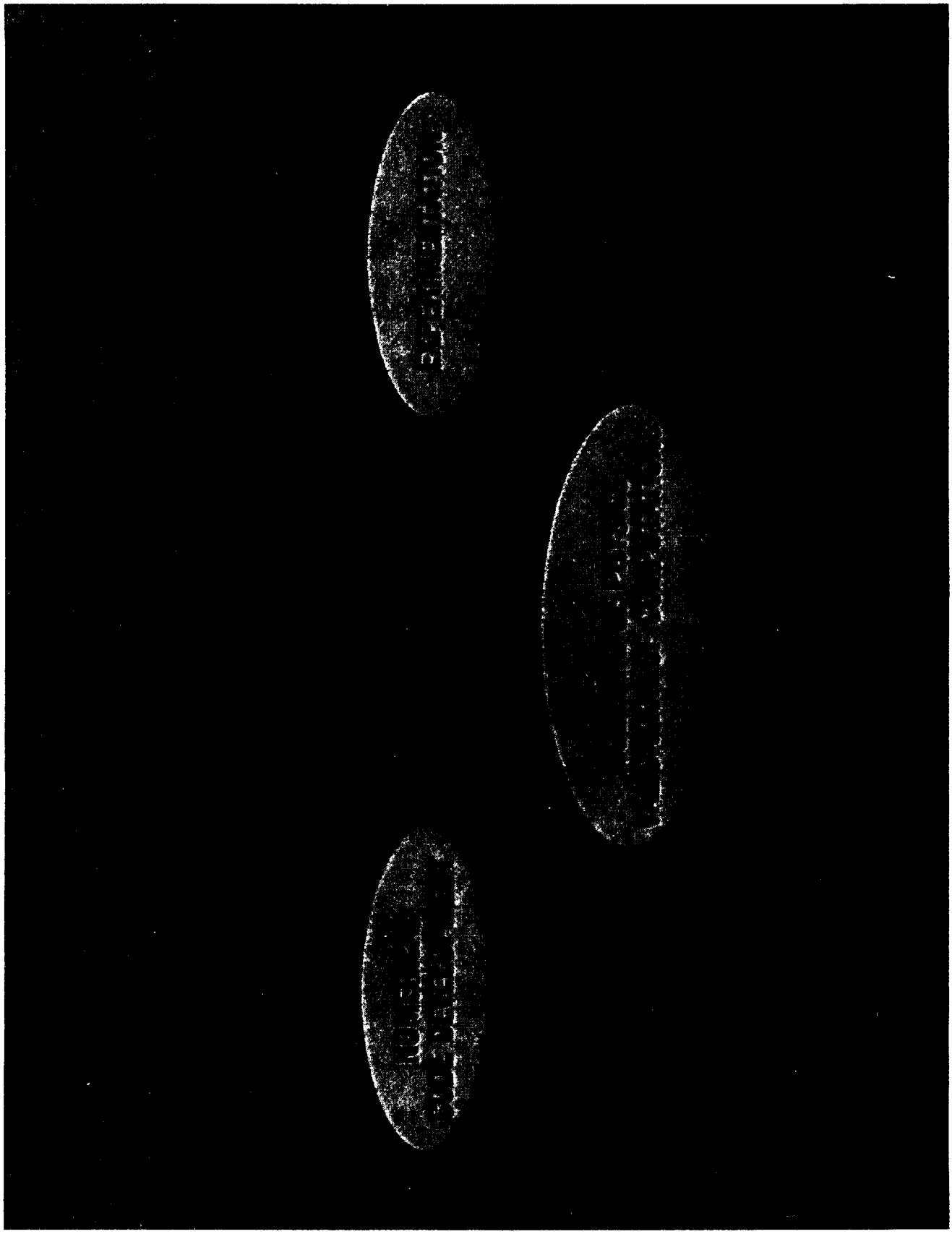
by

John J. Adamczyk
NASA Lewis Research Center

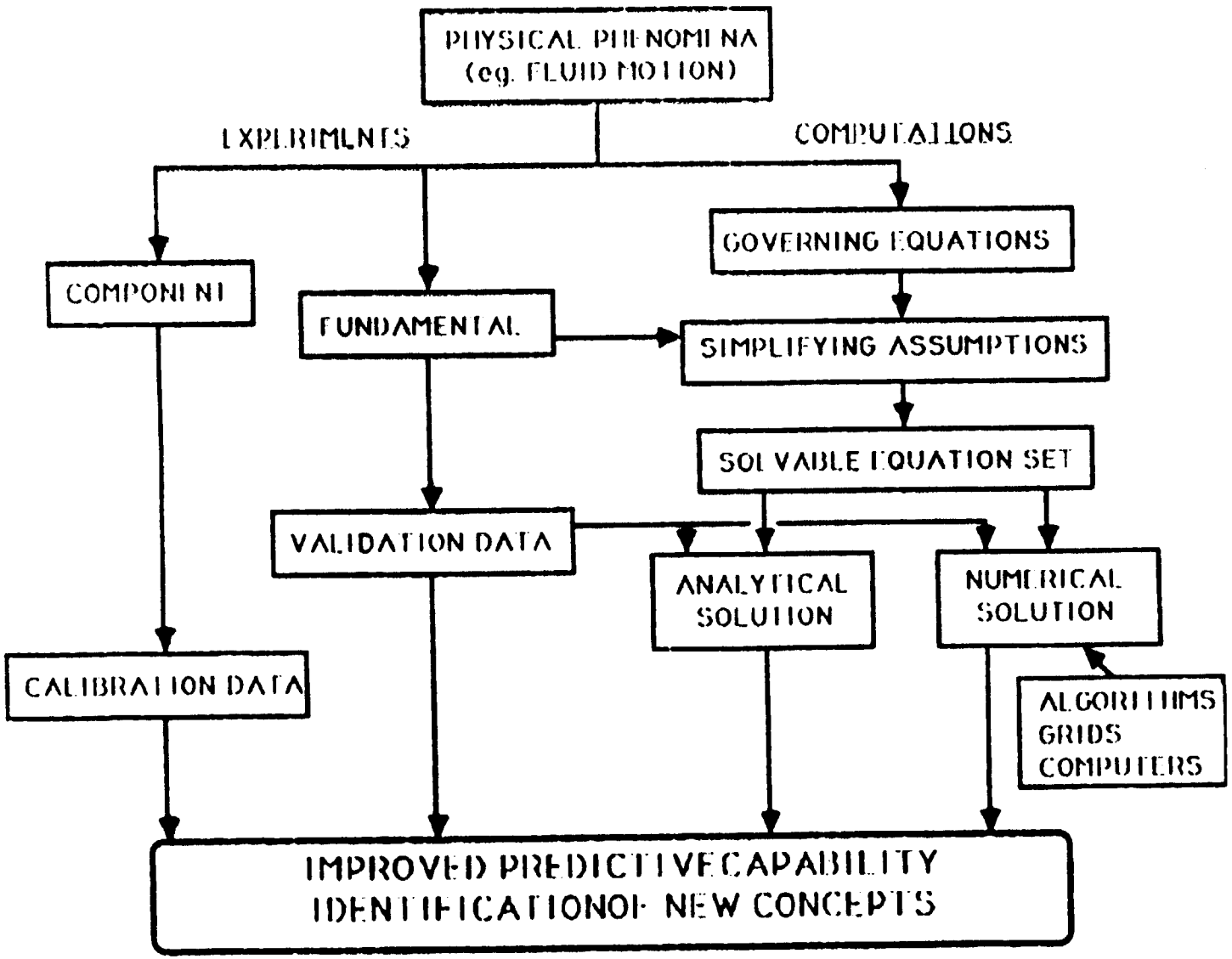
ABSTRACT

Significant advancements have been made in the last five years in the ability to model turbomachinery flows of engineering interest. This advancement can be directly attributed to the second generation of supercomputers like the Cray XMP and Cray II and advanced instrumentation techniques. Early on, the National Aeronautics and Space Administration Lewis Research Center recognized the potential gains in turbomachinery performance and life that could be achieved by taking advantage of this technology and instituted a comprehensive research program in turbomachinery flow modeling. This activity combined the areas of fluid flow analysis, computational fluid dynamics, and experimental fluid mechanics. As a result of this activity, Lewis has become an internationally recognized leader in turbomachinery flow modeling. Many of the research activities conducted under this progam have been utilized by industry. The presentation will give an overview of this program and provide sample illustration of simulation performed to date.

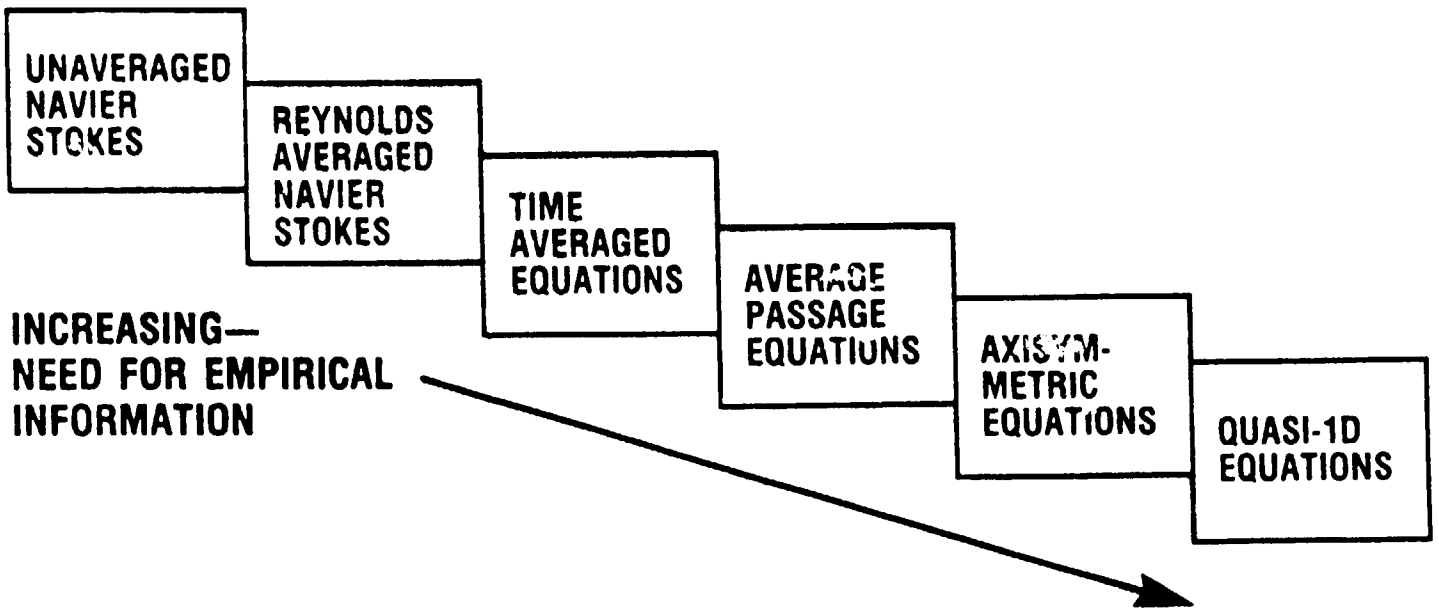




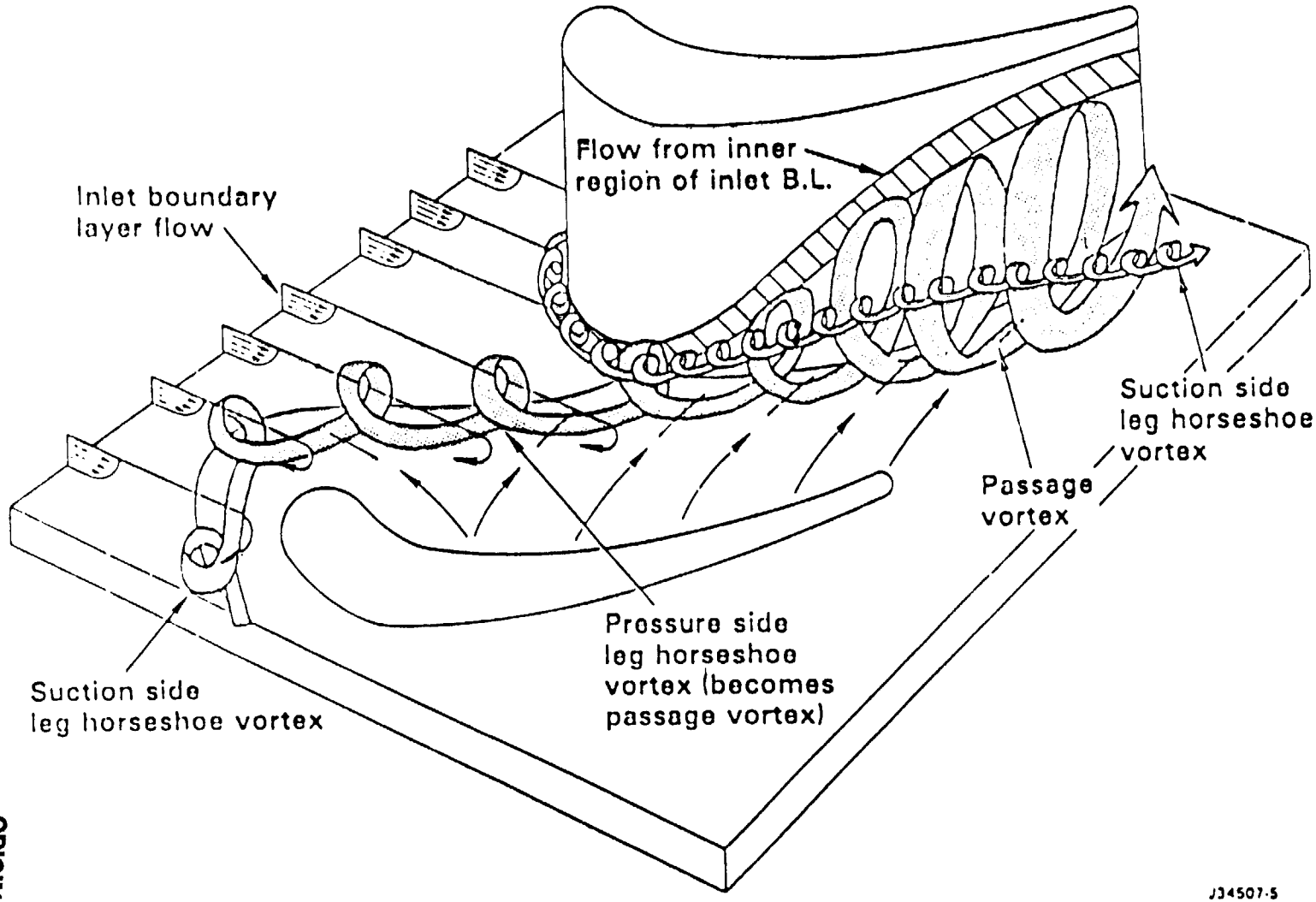
CFD IN THE PROBLEM SOLVING PROCESS



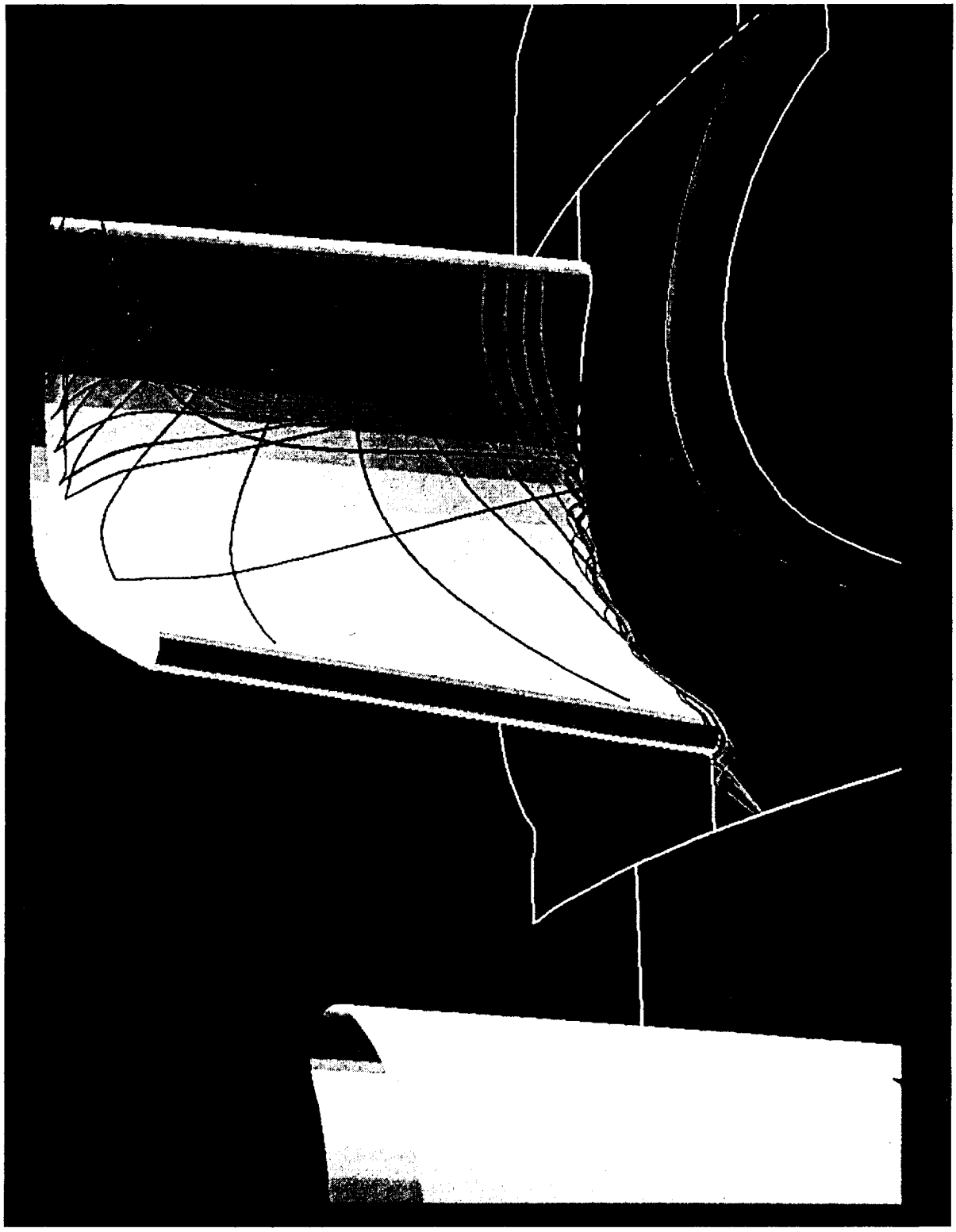
EQUATIONS HIERARCHY

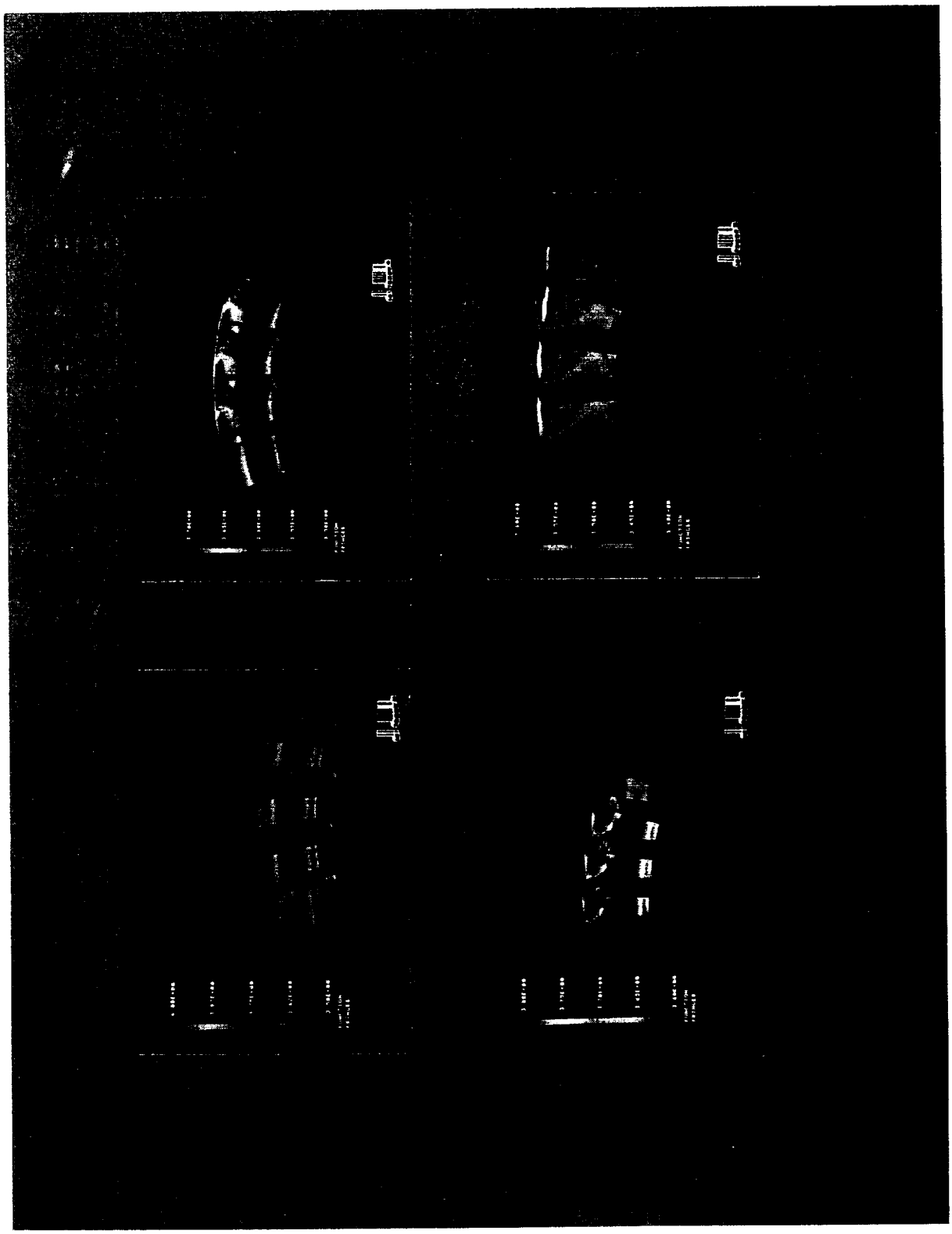


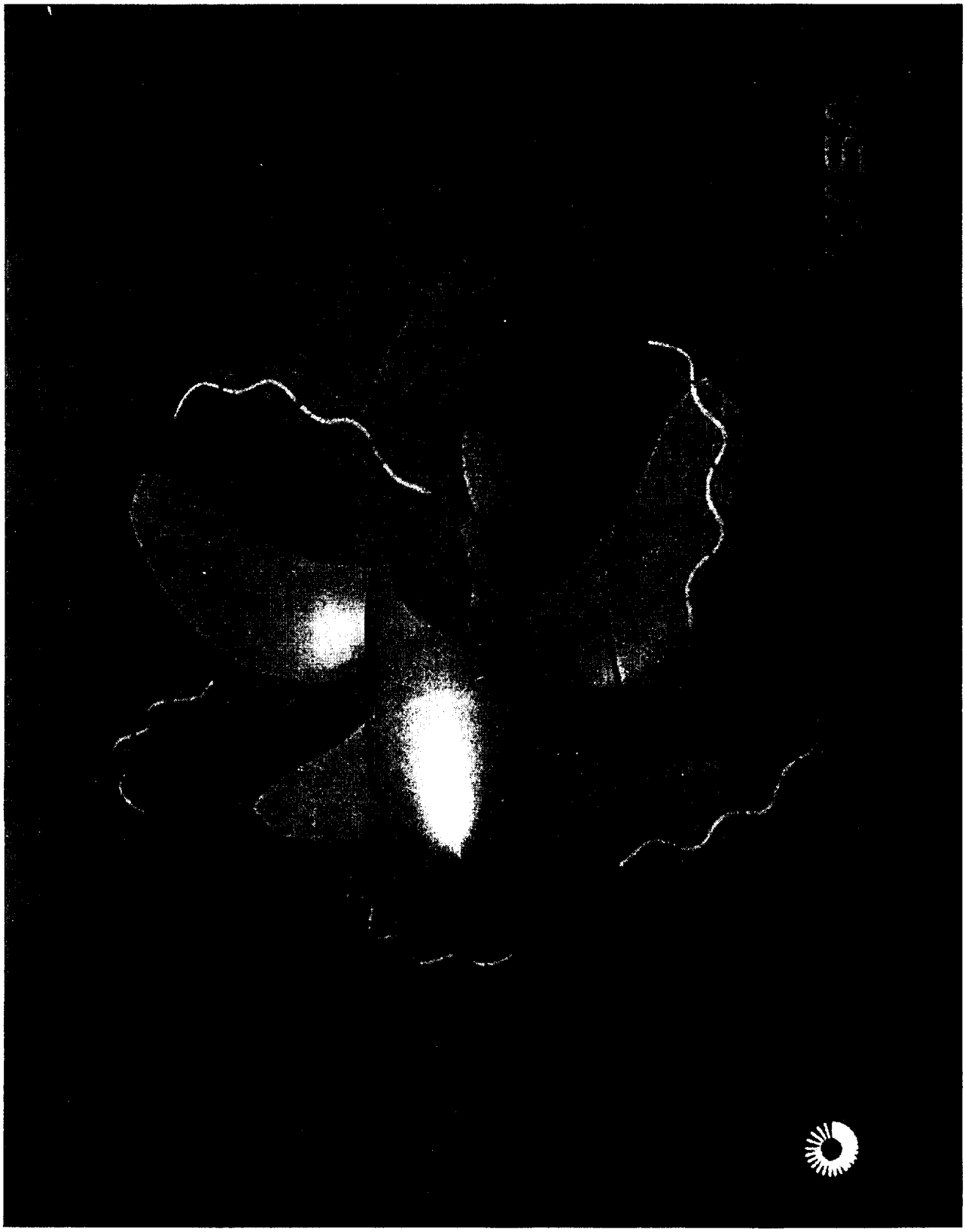
CASCADE ENDWALL FLOW STRUCTURE



J34507-5
860606







ADVANCED CONTRA-ROTATING PROPELLER NOISE PREDICTION

2' GE F7/A7, LeRC 8x6 WIND TUNNEL, DESIGN CONDITION (M=0.72)