汽车空气动力学仿真

Vehicle Aerodynamics Simulation

张扬军

Zhang Yang-Jun

清华大学汽车工程系应用空气动力学组

汽车安全与节能国家重点实验室

Applied Aerodynamics Group, Dept of Auto Eng., Tsinghua Univ.

State Key Lab of Automotive Safety and Energy

汽车空气动力学仿真

Vehicle Aerodynamics Simulation

1 汽车空气动力学概述

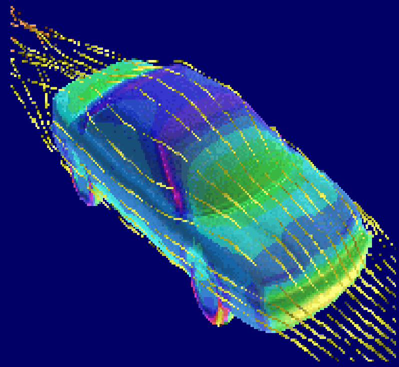
2 汽车空气动力学仿真特点

3 汽车空气动力学仿真难点

4 汽车空气动力学仿真平台

5 仿真平台(VASS)应用

6 总结与展望



1 Introduction to Road Vehicle Aerodynamics
2 Some Salient Features of Road Vehicle Flow Simulation
3 Main Difficulties of Road Vehicle Flow Simulation
4 Vehicle Aerodynamics Simulation System (VASS)
5 VASS Applications
6 Conclusions and Open Features

1 汽车空气动力学概述

Introduction to Vehicle Aerodynamics

1.1 空气动力学对汽车性能的影响

1.2 汽车空气动力学性能

1.3 汽车空气动力学特点

1.4 空气动力学研究方法

1.1 Vehicle Attributes Affected by Aerodynamics
1.2 Vehicle Aerodynamics Characteristics
1.3 Peculiarities of Road Vehicle Aerodynamics
1.4 Methods for Vehicle Aerodynamic

1.1 空气动力学对汽车性能的影响

Vehicle Attributes Affected by Aerodynamics

- 动力性
- 经济性
- 舒适性
- 安全性
- 美观性

- Maximum speed & Acceleration
- Fuel Economy
- Comfort
- Safety
- Visibility

1.2 汽车空气动力学性能

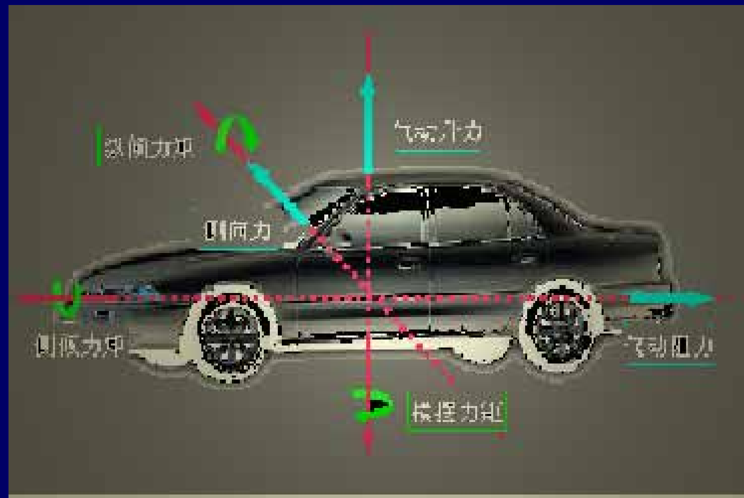
Vehicle Aerodynamics Characteristics

- 气动力气动力矩

  油耗、操纵稳定性

- 流场结构

  流动机理、气动噪声、雨水流动、尘土堆积



- aerodynamic force and moment coefficients

  fuel economy, handling stability

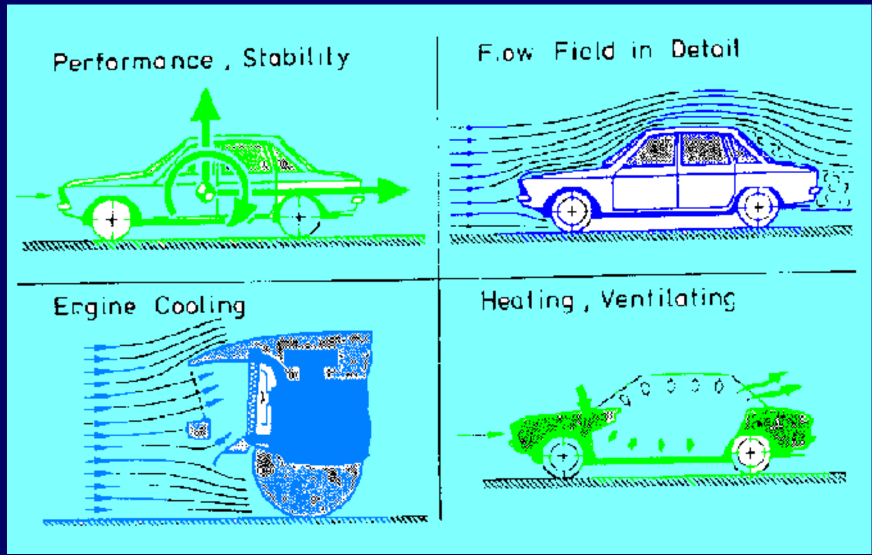
- flow structure

  flow mechanism, aerodynamic noise, rain water, dust accumulating

1.2 汽车空气动力学性能

Vehicle Aerodynamics Characteristics

- 冷却
  散热器、发动机、制动器和差速器冷却
- 通风与换气
  进出风口位置、风量、风速及风路，除结雾性能

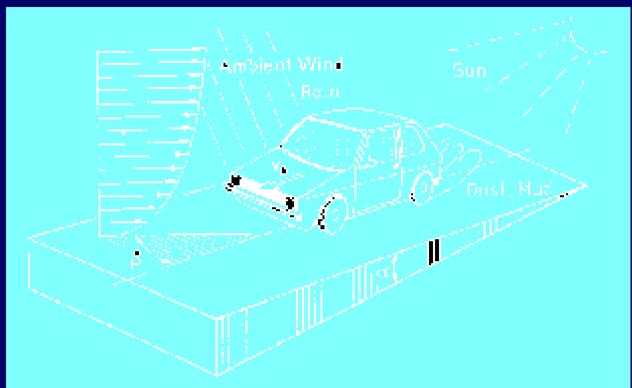


- cooling
  radiator, engine, brake, and differential cooling
- ventilation & air-exchange
  properly locate openings for air inlets and outlets, air flow rate, velocity, air flow path, defrosting, etc.

1.2 汽车空气动力学性能

Vehicle Aerodynamics Characteristics

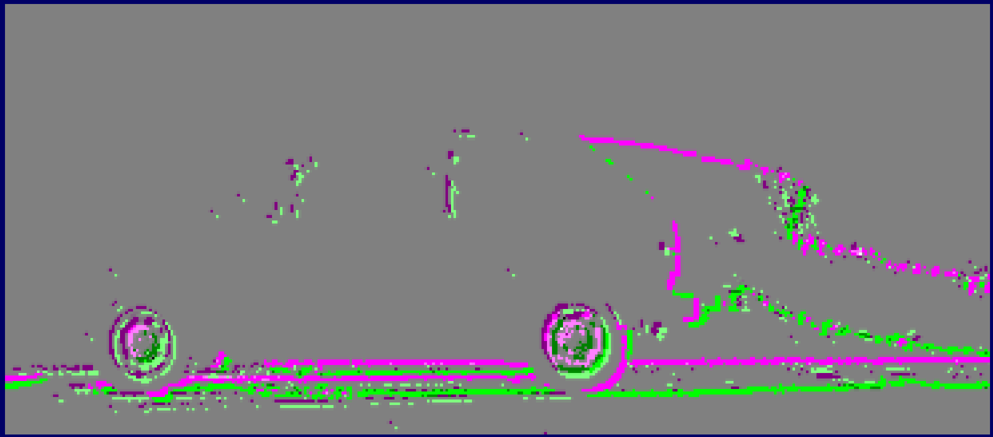
	空气动力分量	相关的动力性能
力	阻力	油耗、最高车速、加速
	升力	行驶稳定性
	侧向力	抗侧风能力(侧移)
力矩	侧倾力矩	抗侧风能力(侧移)
	横摆力矩	抗侧风稳定性
	纵倾力矩	行驶稳定性



	air force components	performance
air forces	air drag	fuel economy, top speed, acceleration
	lift	direction stability
	side force	crosswind
moment	rolling moment	crosswind
	yawing moment	crosswind
	pitching moment	direction stability

1.3 汽车空气动力学特点
Vehicle Aerodynamics peculiarities

- 研究空间有限
- 目标多元化
- 无法量化比较
- 流动复杂



- many details primarily determined by “other than aerodynamic” arguments
- the objectives of aerodynamics differing widely
- difficult to be quantified for weighing the relative importance
- complex flow

1.3 汽车空气动力学特点

Vehicle Aerodynamics peculiarities

❖研究空间有限

车身设计主要由机械工程学、人体工程学和美学等决定，外形和车内空气动力学设计优化的研究与应用均仅能在有限的机动空间中进行

❖ many details primarily determined by “other than aerodynamic” arguments

With the race car being the only exception, the shape of a road vehicle is not primarily determined by the need to generate specific aerodynamic effects--as, for instance, an airplane is designed to produce lift

1.3 汽车空气动力学特点
Vehicle Aerodynamics peculiarities

❖无法量化比较

汽车空气动力学优化设计与控制研究需考虑艺术、时尚和喜好。而艺术、时尚和喜好与气动性能的重要性是无法具体量化比较的。

❖difficult to be quantified for weighing the relative importance

While the process of weighing the relative importance of a set of needs from various disciplines is generally comparable to that in other branches of applied fluid mechanics, the situation in vehicle aerodynamics is unique in that an additional category of arguments has to be taken into account: art, fashion, and taste. In contrast to technical and economic factors, these additional arguments are subjective in nature and cannot be quantified.

1.3 汽车空气动力学特点

Vehicle Aerodynamics peculiarities

❖目标多元化

共同目标	低阻力	
不同目标	负升力	赛车、高速车
	通风与空调	轿车、公共汽车
	气动噪声	轿车、公共汽车
	侧风敏感性	轿车、厢式车

the objectives of aerodynamics differing widely

desirable for all road vehicle: low drag
different requirements:

negative lift	race cars, high speed cars
ventilation & air cond.	cars, buses
crosswind sensitivity	cars, vans
low wind noise	cars, buses

1.3 汽车空气动力学特点

Vehicle Aerodynamics peculiarities

- 流动复杂
  - 几何结构复杂
  - 内外流相互作用
  - 热交换
  - 地面效应
- 粘性
- 湍流
- 强三维性
- 非定常
- 旋涡分离

complex geometry
external flow and internal flow interaction
heat exchange
ground effect
Viscosity
turbulence
strong 3D effects
unsteady
vortex & flow separation

1.4 空气动力学研究方法

Methods for Vehicle Aerodynamics

❖风洞试验

堵塞效应

地面效应

投资大

周期长

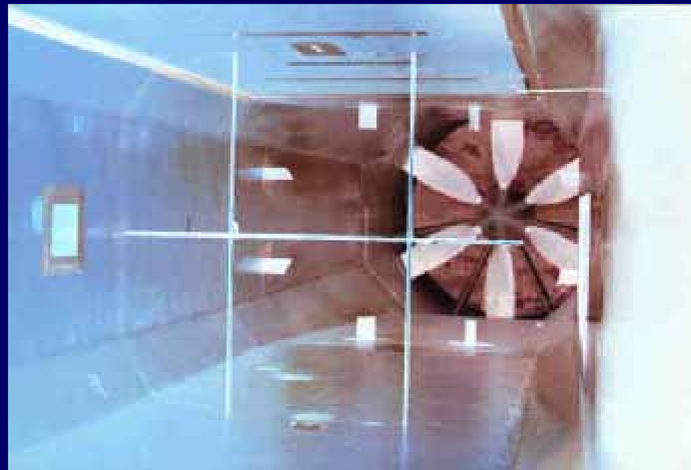
内部流动困难

❖数值仿真

发展迅速

应用广泛

新途径



wind-tunnel testing:

blockage

ground proximity effect

high cost

cost time

difficulties for internal flow

CFD, numerical simulation

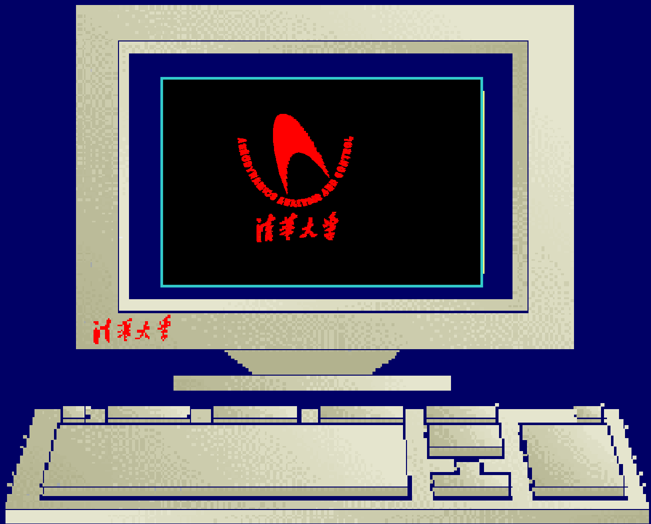
developed quickly

widely used

new design tool

2 汽车空气动力学仿真特点

Features of Vehicle Aerodynamics Simulation



- 可预先研究
- 不受条件限制
- 信息丰富
- 成本低
- 周期短

❶used in early design stage
❷nearly no limitations
❸more information
❹low cost
❺saving time

2 汽车空气动力学仿真特点

Features of Vehicle Aerodynamics Simulation

❶ 可预先研究

- 性能研究　　设计初期：空气动力学性能预测、分析
- 设计优化　　与气动设计交叉进行，得最佳气动设计
- 提供依据　　为汽车造型、空调和通风系统的设计与布置提供依据。

used in early design stage

performance research
　during the early design period, it can generate information before a testable model even exists.
design & optimize
　aerodynamic design and performance study may be studied iterativelly
wide range of design options
　CFD simulation is well suited to the analysis of a wide range of shape options, etc.

2 汽车空气动力学仿真特点

Features of Vehicle Aerodynamics Simulation

❷ 不受条件限制

- 流动参数范围广泛

  不受湍流、风速风向、气温气压以及Re数等限制，能给出流场参数定量结果。

- 无干扰效应等影响

  无洞壁干扰、风洞试验段堵塞效应，可避免风洞试验的支架干扰、模型弹性变形等技术问题以及道路试验条件和交通状况的影响

Without the limitations of turbulence,wind velocity, pressure, temperature and Reynolds numbers etc.
numerical methods are not necessarily burdened with the limitations of the wind tunnel. For example, computational space can be made large enough to eliminate blockage effects.

2 汽车空气动力学仿真特点

Features of Vehicle Aerodynamics Simulation

❸ 信息丰富

- 各种行驶状况　　各种行驶状况，获得比风洞试验更丰富的信息，包括试验难以测量和解释的信息
- 瞬态气动特性　　侧风时瞬态气动特性影响操纵稳定性的研究，数值仿真比风洞试验方法更具优势
- 非定常气动干扰　　同向及对开时非定常气动干扰等试验难于研究，仿真更有其独特优点

much more information available than from a routine experiment

sometimes a CFD simulation permits the investigation of situations that can note be realistically duplicated in a wind tunnel, such as transient crosswind sensitivity.

the aerodynamics of two vehicles in the passing or overtaking mode, for example, poses a difficult problem for wind tunnel tests.

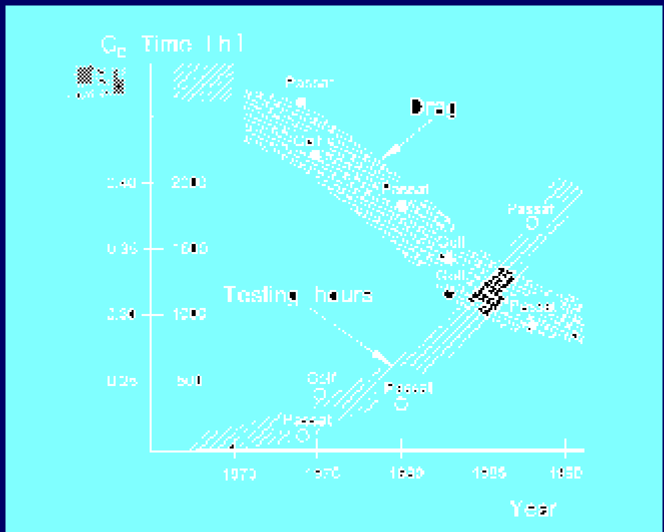
2 汽车空气动力学仿真特点

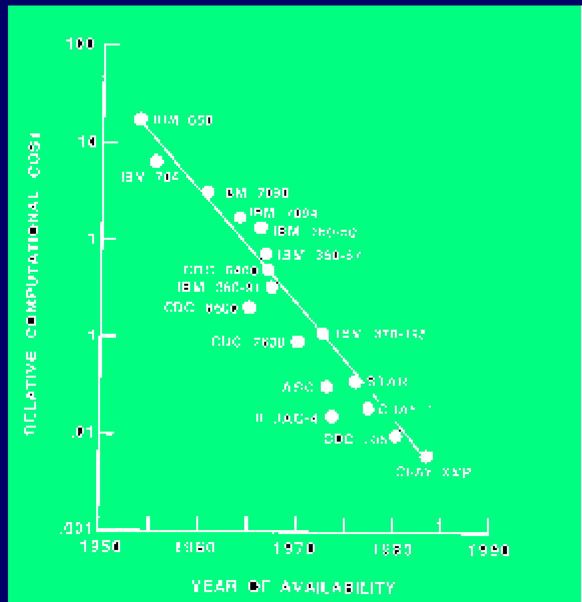
Features of Vehicle Aerodynamics Simulation

❹ 成本低

❺ 周期短

	70's		90's
◆风洞试验	100h	➡	1800h
◆数值仿真	成本降低了近100倍		





❹ low cost

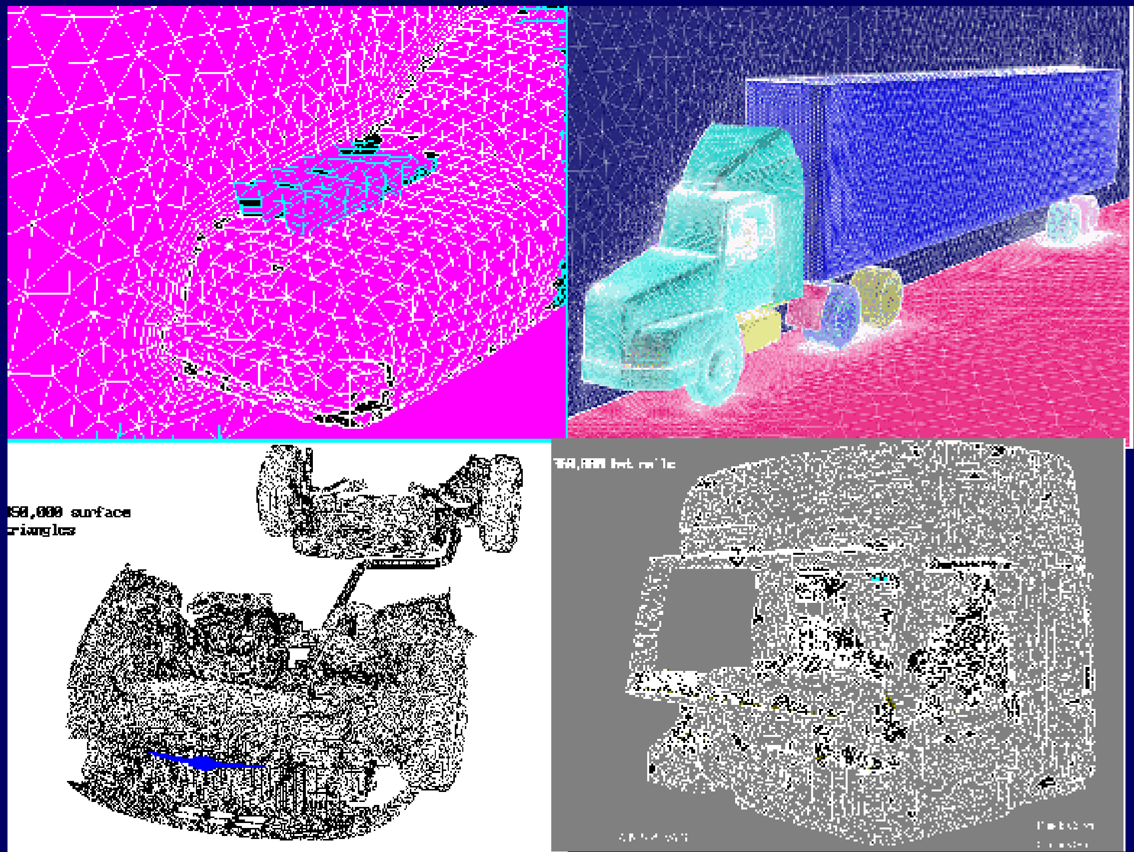
❺ saving time

CFD is becoming markedly more economical as time goes due to the revolutionary advances in computer technology

3 汽车空气动力学仿真难点

Difficulties of Vehicle Aerodynamics Simulation

1)、几何结构复杂，网格建模困难

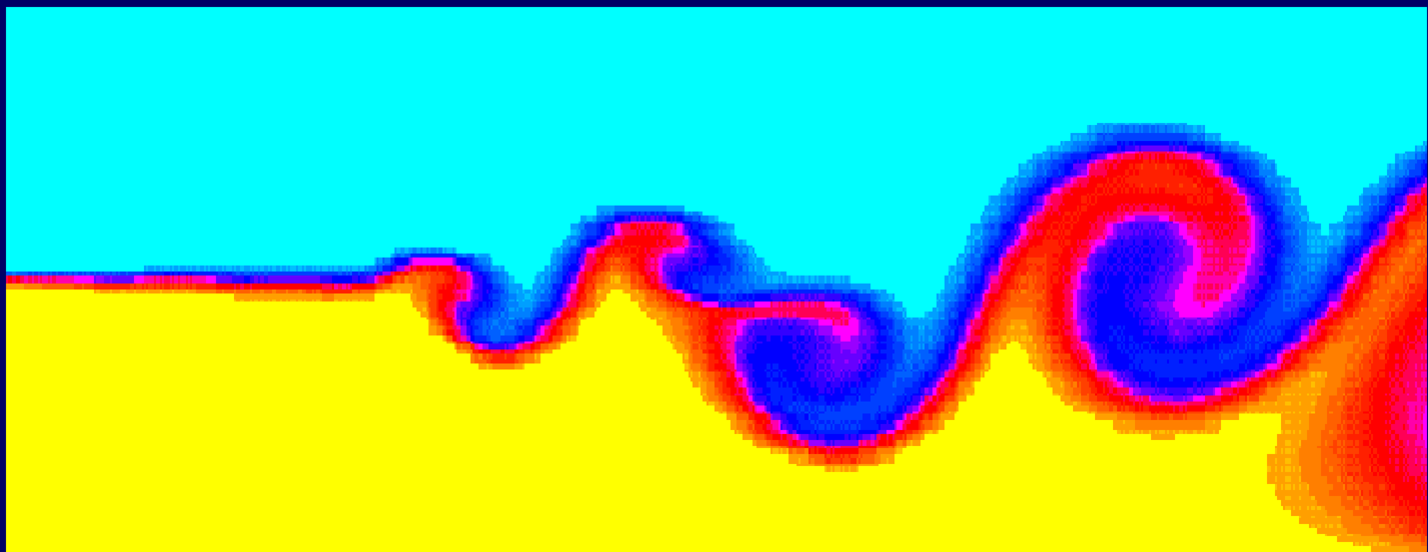


complex geometry, grid generation difficult

3 汽车空气动力学仿真难点

Difficulties of Vehicle Aerodynamics Simulation

2)、为充分发展湍流，不能按层流计算流动

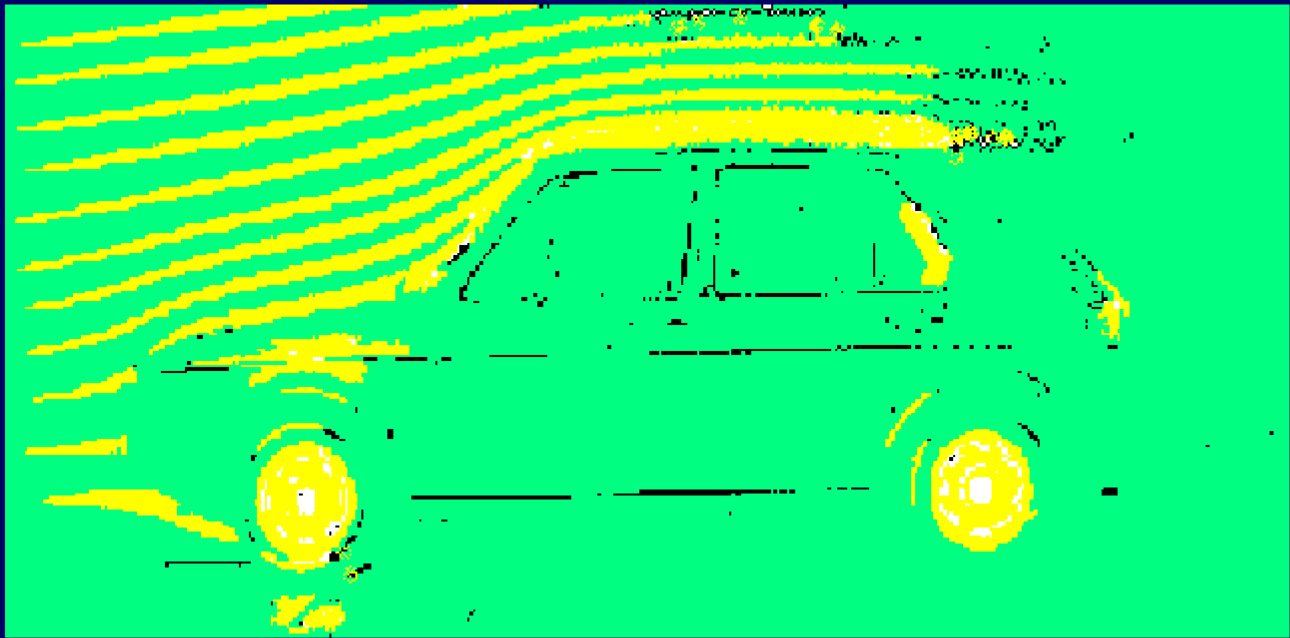


full developed turbulent flow

3 汽车空气动力学仿真难点

Difficulties of Vehicle Aerodynamics Simulation

3)、流动强三维性，难于简化分析



strong three-dimensional effects,2D solution is not suitable

3 汽车空气动力学仿真难点

Difficulties of Vehicle Aerodynamics Simulation

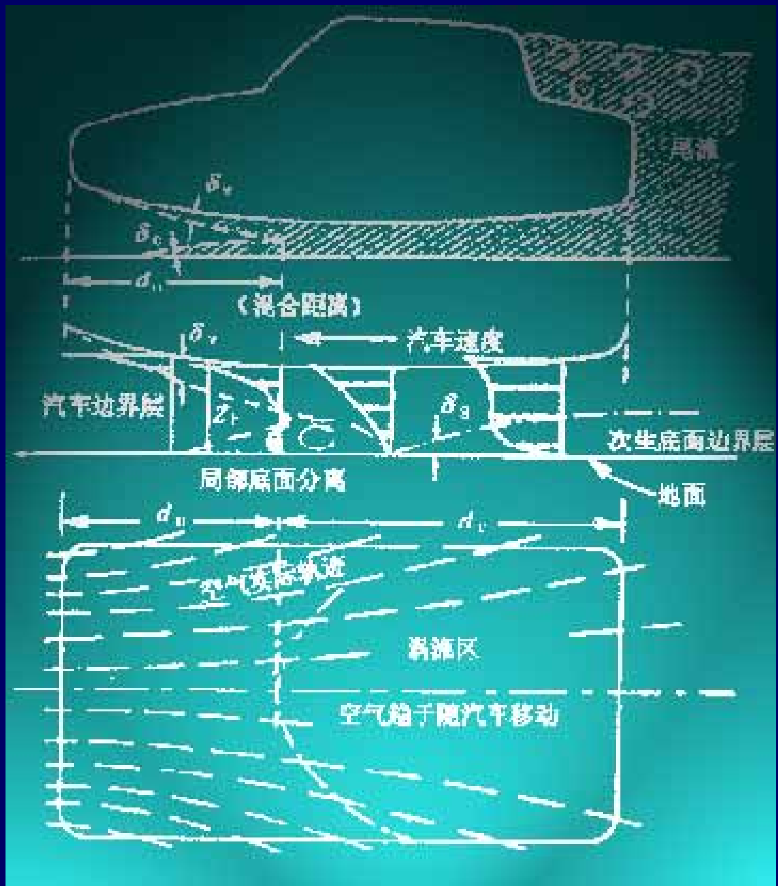
4)、各种尺度旋涡，闭式、开式分离流动；机理还不很清除，物理数学模型问题

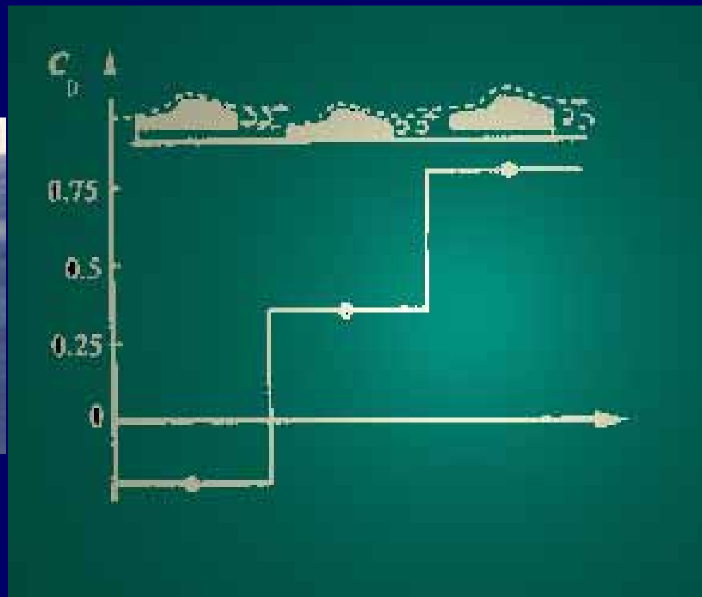
vortex flow, closed or open regions of separated flow

3 汽车空气动力学仿真难点

Difficulties of Vehicle Aerodynamics Simulation

5)、地面效应，航空成果难于直接应用



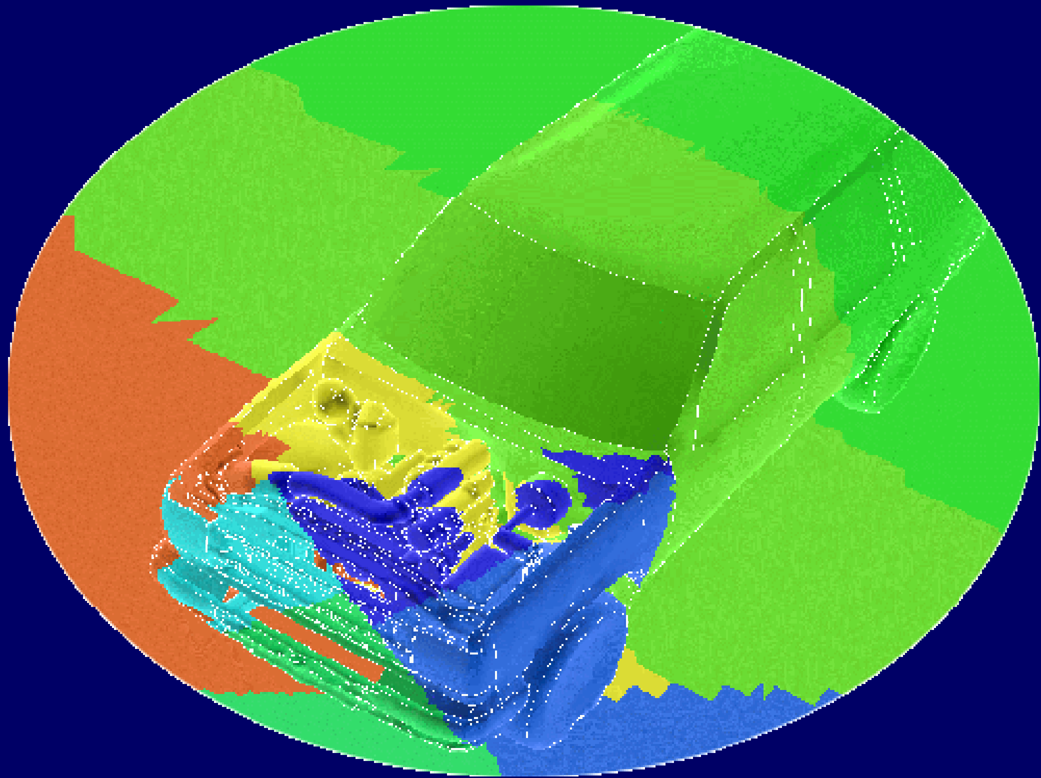


ground proximity effect, aircraft industry methods are not always directly applicable or suitable to treat problems of vehicle aerodynamics.

3 汽车空气动力学仿真难点

Difficulties of Vehicle Aerodynamics Simulation

6)、流场与温度场耦合,热源难于确定

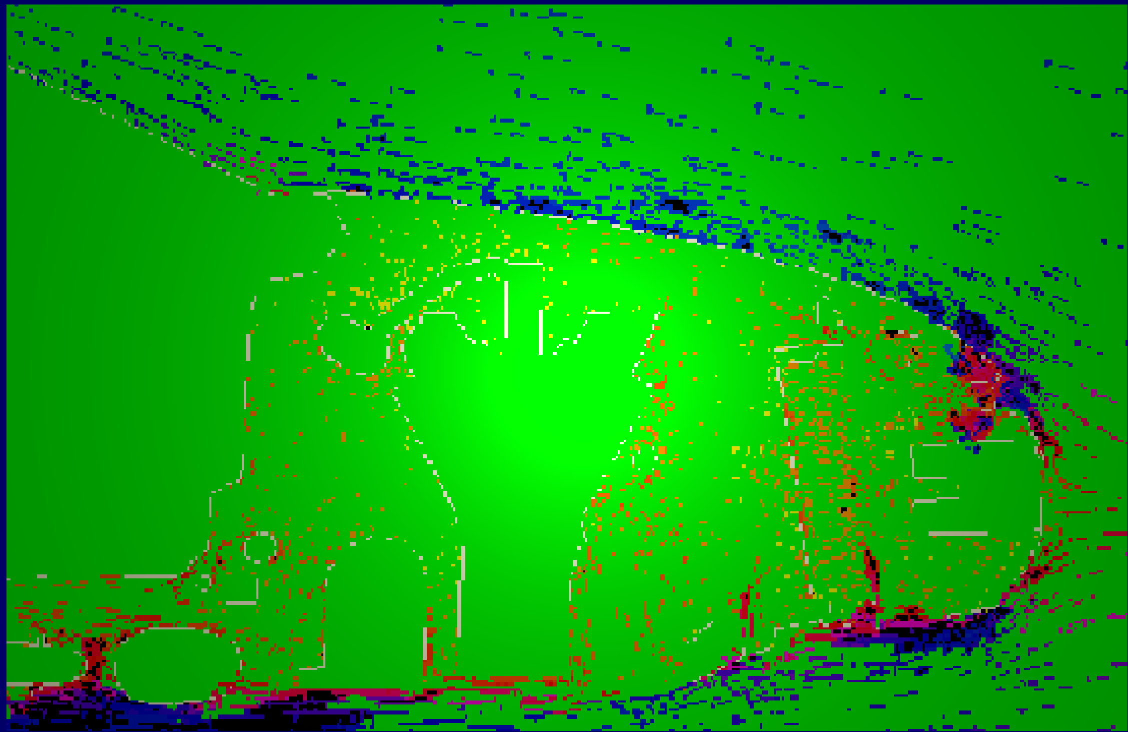


flow field coupling temperature field, engine cooling, etc.

3 汽车空气动力学仿真难点

Difficulties of Vehicle Aerodynamics Simulation

7)、流场相互作用问题，存在内外流耦合问题

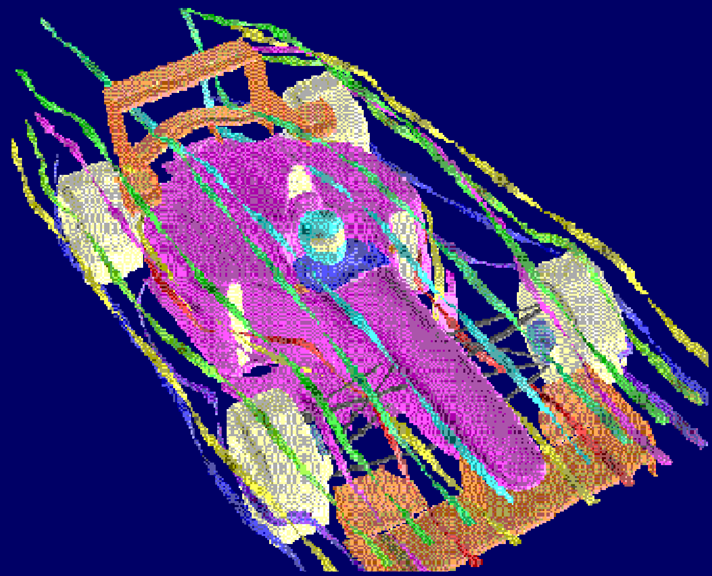


internal and external flow fields are closely related. Both flow fields must be considered together

3 汽车空气动力学仿真难点

Difficulties of Vehicle Aerodynamics Simulation

8)、流动非定常，定常体系难于考虑



flow is unsteady, however, the aerodynamic design system is a steady one.

4 汽车空气动力学数值仿真平台

Vehicle Aerodynamics Simulation System (VASS)

4.1 开发背景

4.2 技术方案

4.3 系统组成

4.4 仿真环境

4.5 技术水平

4.6 主要特色

- background
- technical blue print
- system introduction
- simulation environment
- technical level
- main features

4.1 仿真平台开发背景

Background of VASS Development

- 国 外　　国际上各有关厂、所和学校在80年代初就开始了相关研究并建立了仿真系统

- 国 内　　我国还处于起步阶段,还没有相应仿真系统

background

overseas: CFD is a development tool
domestic: no CFD system for vehicle aerodynamics simulation

4.1 仿真平台开发背景

Background of VASS Development

◆清华大学

985项目 轿车车身车型总体设计关键技术和轿车数字化工程均将汽车空气动力学仿真平台作为主要研究内容，希为汽车车型车身设计能力的提高提供技术支持

vehicle aerodynamics simulation system (VASS) development is one of the main research subjects at Tsinghua University

4.2 仿真平台技术方案

Technical Blue Print for VASS

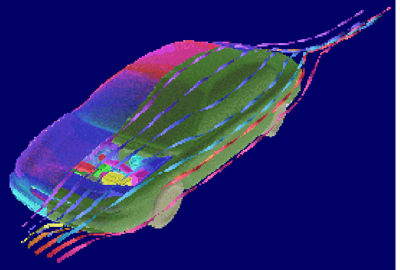
网格系统	分区生成结构/非结构杂交化网格
流场控制方程	雷诺平均N－S方程RANS
方程求解方法	有限体积法FVM
湍流模型	二方程湍流模型/低Re数修正

The capability of an unstructured hybrid grid method to compute the Navier-Stokes equations is discussed for complex flows around automobile configurations. The unstructured hybrid grid, which is composed of a structured or a semistructured grid for the near-wall viscous region, a prismatic grid for the exterior regular computational domain, and an unstructured grid for the remainder of the computational domain, is used to treat such viscous flows. The Navier-Stokes equations are solved on the hybrid grid by a cell-vertex, upwind finite volume method.

4.3 仿真平台系统组成

VASS Introduction

① 主要模块



非结构网格生成模块
结构网格生成模块
数学物理模型选择模块
初始条件设定模块
边界条件设定模块
计算控制条件设定模块
数值计算模块
气动力及气动力矩计算模块

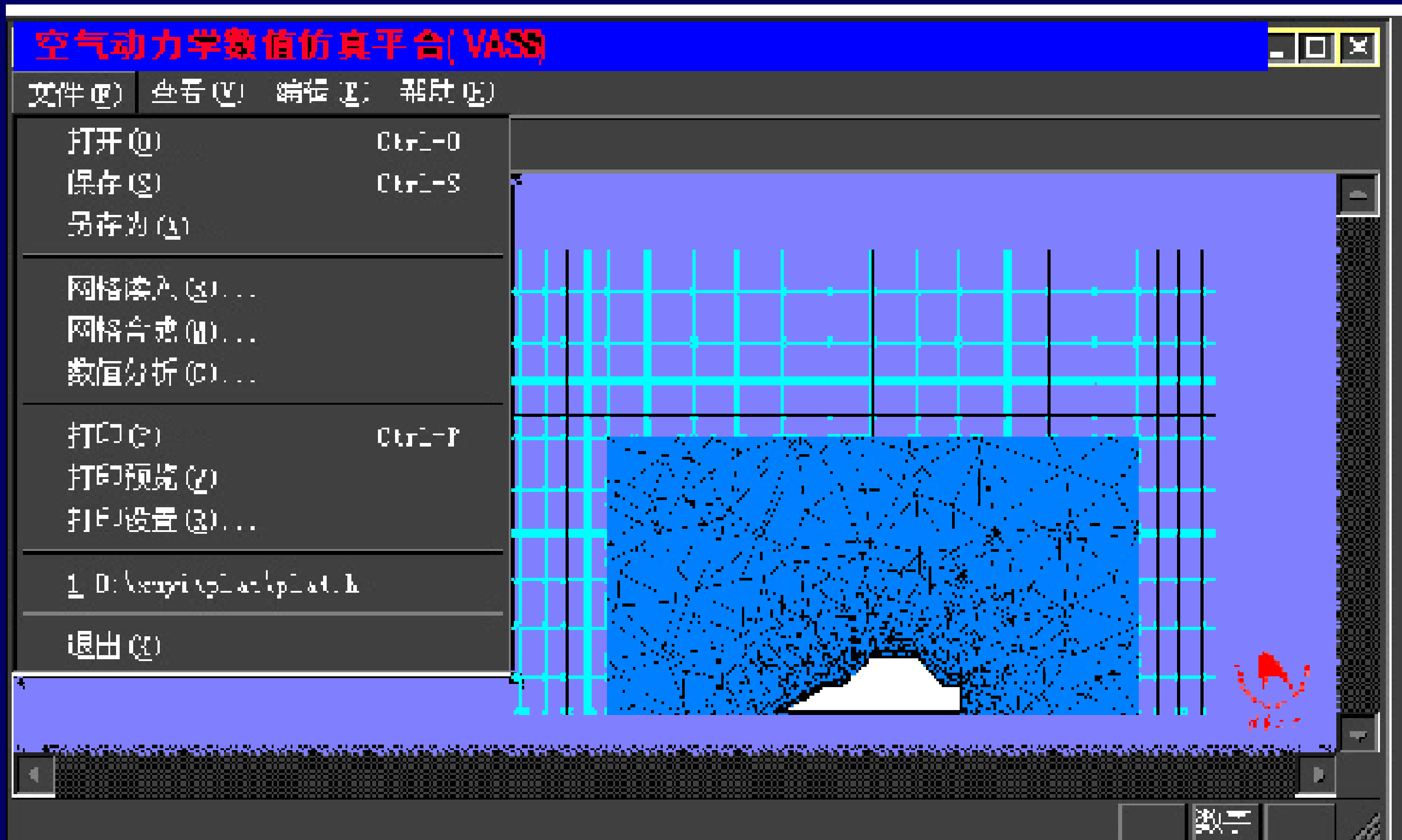
① main modules
unstructured grid generation
prismatic grid generation
theoretical model selection
initial condition determination
boundary condition determination
calculation control determination
numerical computation
aerodynamic force and moment cal.

4.3 仿真平台系统组成

VASS Introduction

② 系统窗口

②system windows

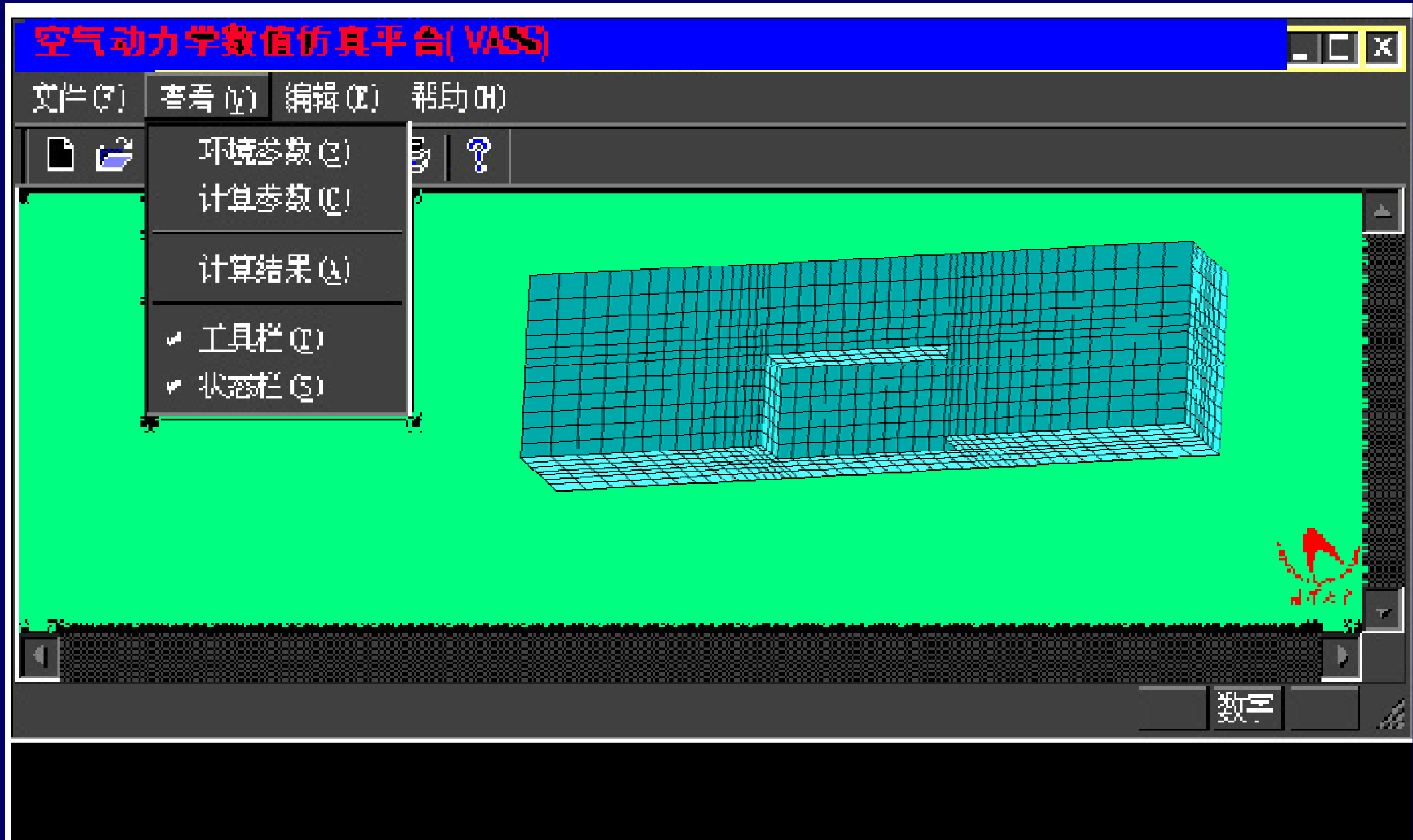


4.3 仿真平台系统组成

VASS Introduction

② 系统窗口

②system windows



4.3 仿真平台系统组成

VASS Introduction

② 系统窗口

②system windows

4.4 仿真平台仿真环境

Hardware, software & Operating System

硬件环境：　内存>256MB 硬盘空间>3G

操作系统：　Windows NT 或 UNIX

软件环境:　ICEM-CFD 4.0　STAR-CD 3.1

simulation environment

hardware　memory >256MB　hard disk >3G

operating system　Windows NT or UNIX

software　ICEM-CFD 4.0　STAR-CD 3.1

4.5 仿真平台技术水平

VASS Evaluation

- ◆专业性：能够有效模拟复杂的三维汽车流场的粘性湍流流动。
- ◆通用性：能对于不同车型生成自适应网格，并可模拟多种工况条件，参数设置方便
- ◆平台具有良好的操作性和可扩容性

specialized in vehicle aerodynamics simulation, automatic adaptive grid system generation, suitable to extensive situations,easy to operate

4.5 仿真平台技术水平

VASS Evaluation

◆核心软件可靠，仿真精度高

ICEM－CFD，STAR－CD

国际汽车界广泛实用

◆系统验证考核

国际通用考核汽车流场计算准确性的Ahmed模型，

气动阻力试验值0.184,计算值0.181,误差1.63%

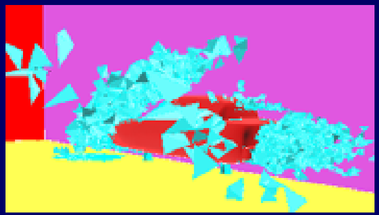
The numerical accuracy of VASS has been discussed by comparing with the experimental data for the same configurations. It demonstrates that the VASS (the main parts are ICEM-CFD and STAR-CD) is efficient and accurate enough to predict the complex flow fields around automobiles.

4.6 仿真平台主要特色

Main Advantages of VASS

❶ 求解域分区

❷ 结构/非结构网格



❸ 壁面半结构网格、网格自适应

❹ 定解条件可重复应用

❺ 地面效应模型

❻ 专业程度高

❼ 实用性好

❽ 一定程度自动化

- domain into several subareas
- hybrid grid system
- semistructured grid near-wall grid adaptive
- parameter setup reusable
- ground proximity effect model
- specialized in vehicle aerodynamics simulation
- practical
- automatic in some extent

4.6 仿真平台主要特色

Main Advantages of VASS

❶ 求解域分区

特 点　求解域大、车身几何形状复杂。不同区域对网格和流场的精度要求不同

分 区　按照车身流场结构和求解特点，将求解域分为内区和外区，包围车身的局部求解区域为内部区域，其余部分为外部区域

unstructured grid methods could not attain superiority over structured grid methods until recently, particularly for high Reynolds number viscous flows around automobiles. This was mainly because of the painful inefficiency and memory overhead for the unstructured grid methods as compared with the available structured grid methods. Because of the unstructured distributions of grid points, simple explicit schemes had been generally employed for integrating the Euler and Navier-Stokes equations to steady state.

4.6 仿真平台主要特色

Main Advantages of VASS

❷ 结构/非结构网格

结构网格 外部的巨大区域采用结构化网格技术，可以大大降低网格生成的计算量，并可以根据流场特点有效控制疏密，提高流场仿真计算的精度，减小流场仿真计算所需的网格量。

非结构网格 包围车身的内部区域采用非结构网格生成技术，可以适应车身的复杂几何形状。

In addition, many three-dimensional unstructured schemes require the excessive memory overheads, which severely limit the size of the meshes, and hence, the class of problems to which they are applied. These problems become much worse for flow computations of high Reynolds number viscous flows around automobiles. More precisely, accurate resolution of the thin boundary layers developed along the wall surface requires very fine grid that causes a stiffness problem of the flow solver. Moreover, the generation of such a fine and stretched grid near wall is another challenging issue of the unstructured grid methods. Several important advancements were made in the tetrahedral grid method to overcome these problems for solving viscous flows.

4.6 仿真平台主要特色

Main Advantages of VASS

❸ 壁面半结构网格

航空流场仿真经验。半结构化网格技术生成的表面网格可满足车身壁面湍流边界层计算对壁面网格的特殊要求

❹ 定解条件可重复应用

该系统所生成的外部区域网格和基于网格上的定解条件可重复应用，提高了工作效率

In Tsinghua, in contrast to the fully tetrahedral grid method, a hybrid grid that is composed of a structured or a semistructured grid for the near-wall viscous region, a prismatic grid for the exterior regular computational domain, and an unstructured grid for the remainder of the computational domain.

4.6 仿真平台主要特色

Main Advantages of VASS

❺ 地面效应模型

地面效应　　地面效应是汽车空气动力学的主要难点，亦是该学科区别于航空空气动力学的主要特点

模 型　　通过分析汽车和地面之间相对运动关系，提出地面效应模型。模型能反映随离地间隙增大，气动阻力系数非单调性变化这一正确趋势，而一般风洞试验结果均未能体现出这一特征

When compared to the conditions without the ground, the flowfield and resulting aerodynamic forces can change as the ground plane is approached. This is important for vehicles moving in close proximity to the ground. Unfortunately, the ground effect is very to simulate in a windtunnel. VASS system has advantages to simulate the ground effect

4.6 仿真平台主要特色

Main Advantages of VASS

❻ 专业程度高

物理数学模型应用了汽车空气动力学的最新研究成果，能较有效模拟汽车复杂的三维汽车的粘性分离、旋涡、湍流流场，专业程度高。

❼ 实用性好

整个仿真系统具有良好的适应性和操作性，能对各种类型、大小和工况下的汽车流场进行分析。所有操作步骤完全程序化和模块化，大大简化工作量。

specialized in vehicle aerodynamics simulation, automatic adaptive grid system generation, suitable to extensive situations,easy to operate

4.6 仿真平台主要特色

Main Advantages of VASS

❽ 一定程度的自动化

对于不同车型生成自适应网格，在读入相关车身几何数据后，能自动根据缺省设置的初始条件和边界条件，对典型工况的汽车气动受力情况作出计算分析，并得出主要数据结果。所有操作步骤完全程序化和模块化，简化了工作量

automatic in some extent

5 仿真平台应用举例

VASS Applications

5.1　结构/非结构耦合网格系统

5.2　AHMED模型，后部倾角对车身流场的影响

5.3　离地间隙对车身流场的影响

5.4　转静相互作用

5.1 hybrid grid system
5.2 AHMED model
5.3 ground effects
5.4 rotating/non-rotating interaction

5.1 结构/非结构网格系统

Hybrid Grid System

采用TETRA生成车身外部流场计算网格。揭示网格系统处理汽车旗杆等细微结构的能力.

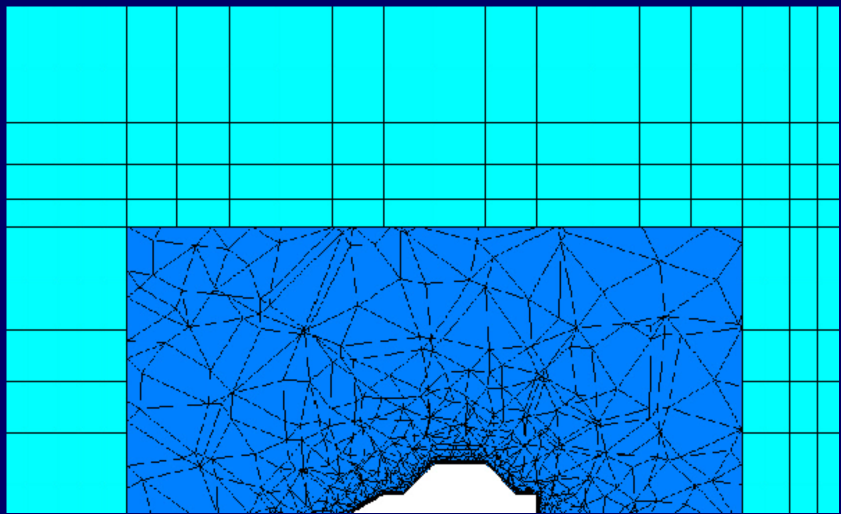
车身表面采用三棱柱半结构化网格，形成半结构化网格和非结构化网格的耦合，能够满足车身壁面湍流粘性边界层计算的需要。

A new efficient, robust and automatic method for hybrid grid generation for complex vehicle geometries.

5.1 结构/非结构网格系统

Hybrid Grid System

❖ 分区　　将整个求解域分为内、外两区,解决车身几何形状复杂、网格生成计算量大等难点。内区非结构化网格以适应复杂的车身几何形状；外区结构化网格，大大减小网格生成的计算量，并可根据流场特点有效控制疏密，提高流场仿真计算的精度，减小流场仿真计算所需的网格量。



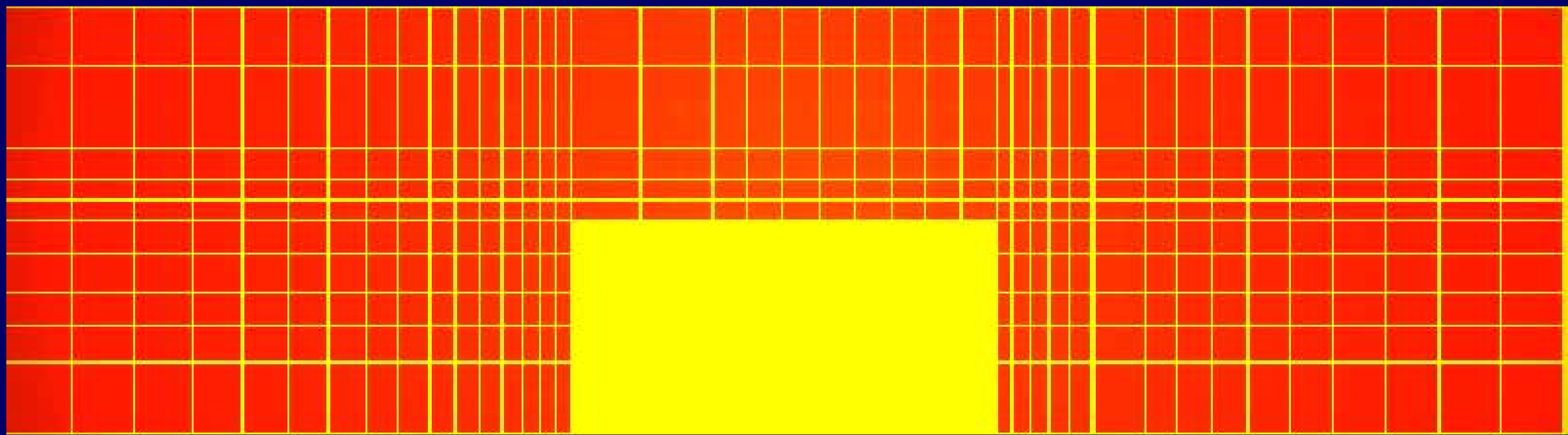
Inner domain: unstructured grids
outside domain: structure grids

5.1 结构/非结构网格系统

Hybrid Grid System

- 外区网格和定解条件可重复使用

在外部区域的网格生成之后，重新进行流场计算时，可以重复应用，不需再生成外部区域的网格。对于车身修型时的大量流场仿真计算，基于外部区域网格上的定解条件可以直接应用，大大提高工作效率。



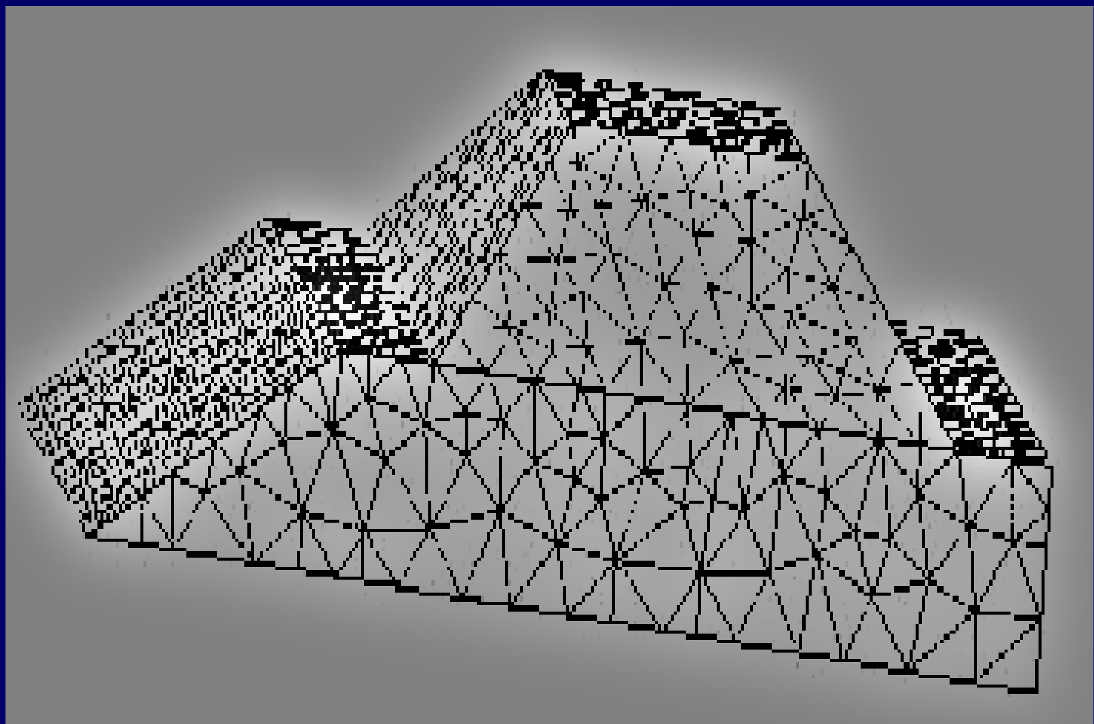
outside domain: structure grids and boundary conditions reusable

5.1 结构/非结构网格系统

Hybrid Grid System

❖ 表面半结构化网格

车身表面采用三棱柱半结构化网格，形成半结构化网格和非结构化网格的耦合，能够满足车身壁面湍流粘性边界层计算的需要。

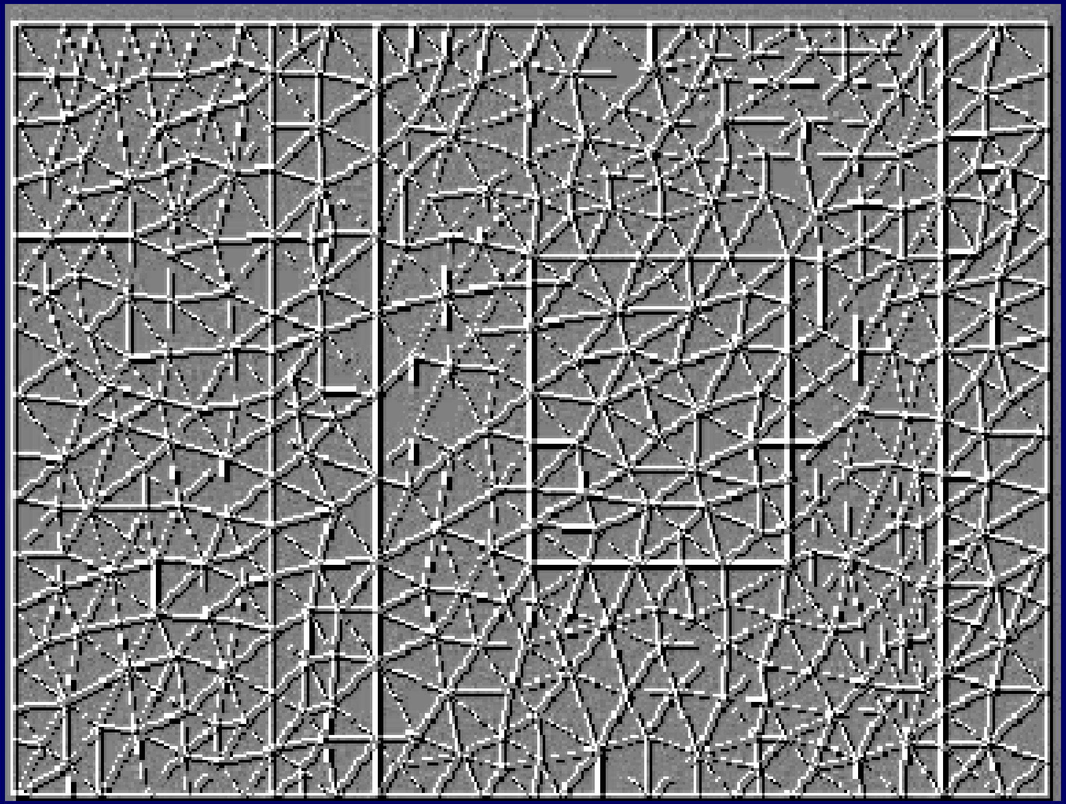


semistructured grid for the near-wall viscous region

5.1 结构/非结构网格系统

Hybrid Grid System

- 表面半结构化网格

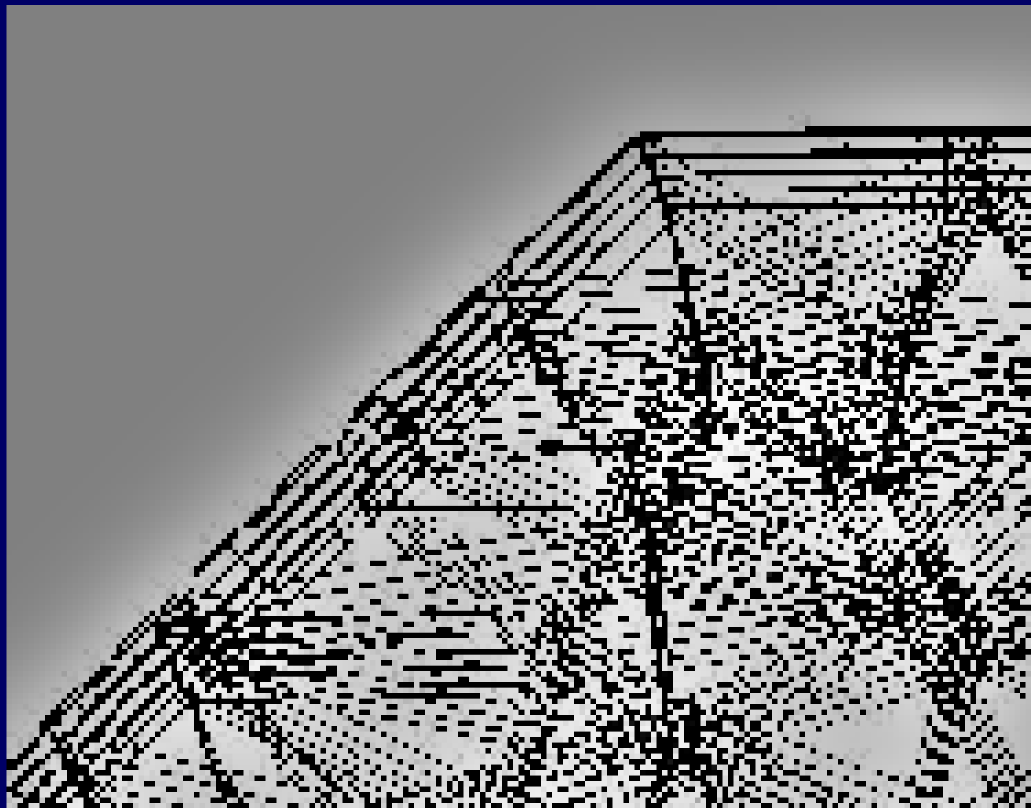


semistructured grid for the near-wall viscous region

5.1 结构/非结构网格系统

Hybrid Grid System

- 表面半结构化网格

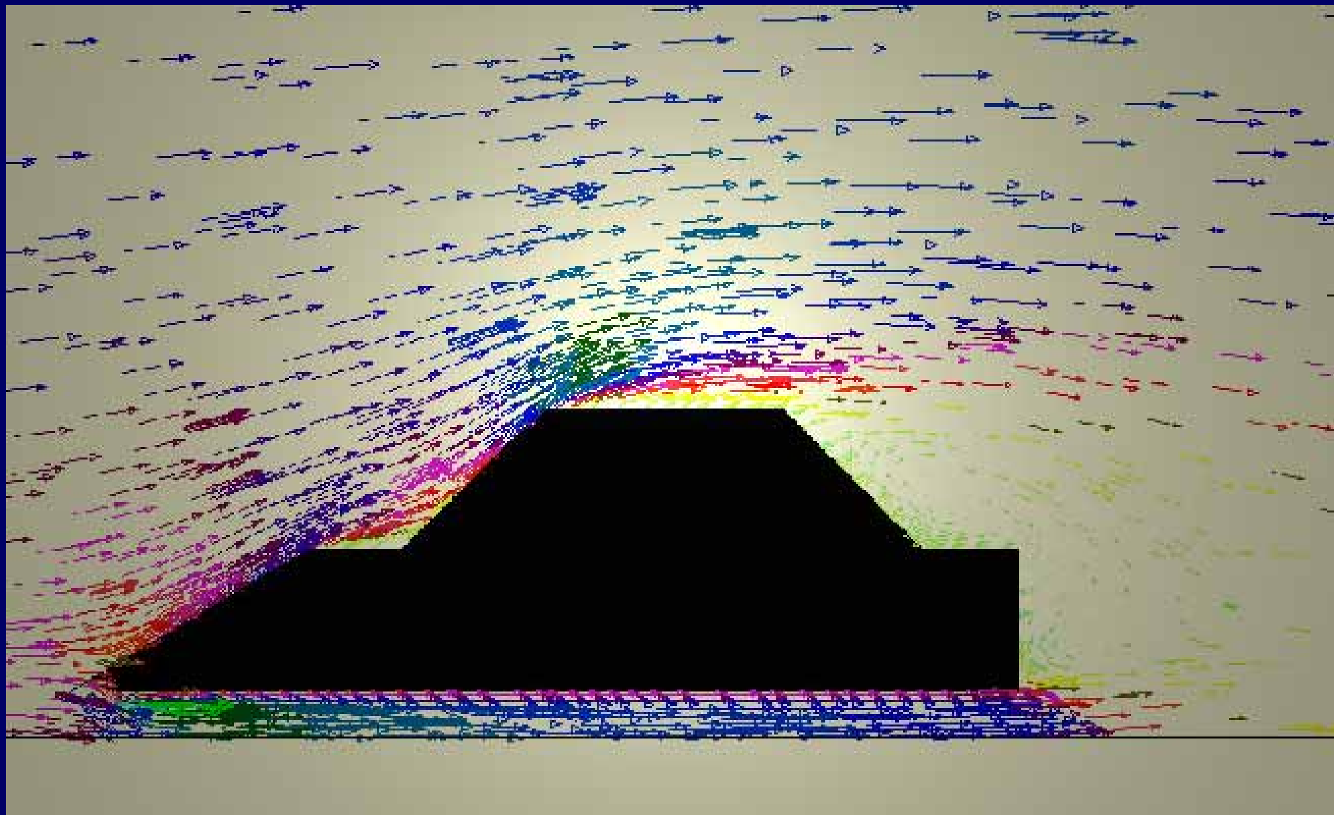


semistructured grid for the near-wall viscous region

5.1 结构/非结构网格系统

Hybrid Grid System

- 速度矢量图

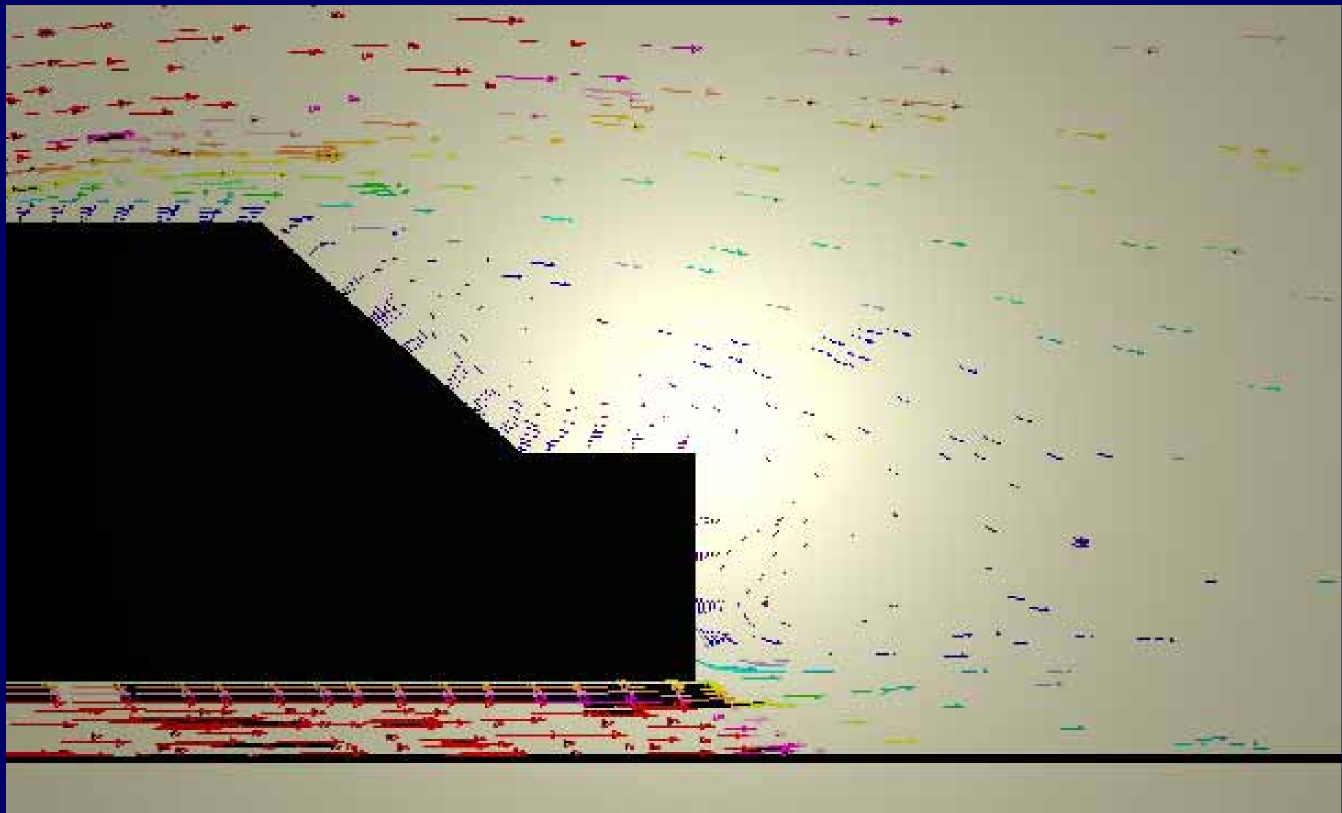


velocity vectors

5.1 结构/非结构网格系统

Hybrid Grid System

- 模型后部的分离旋涡结构

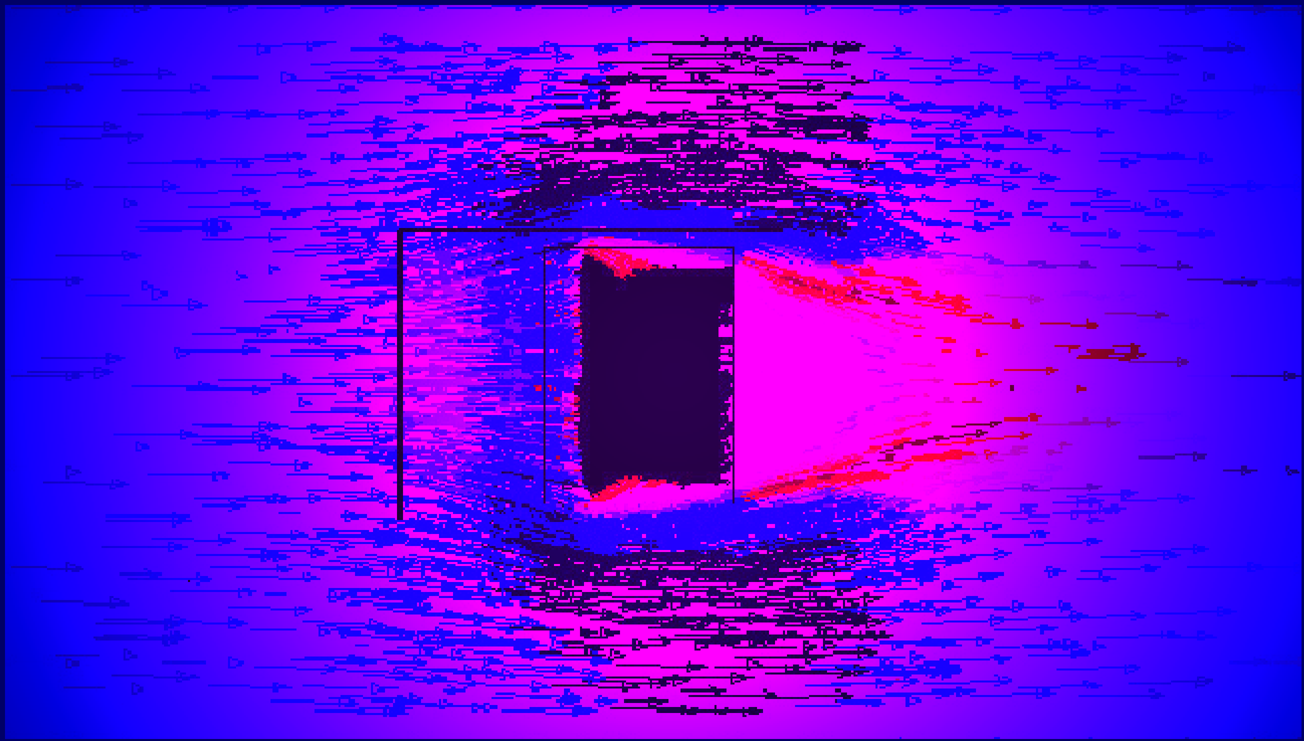


Vortex structure

5.1 结构/非结构网格系统

Hybrid Grid System

- 模型两侧及后部的复杂旋涡流场



Complex flow structure

5.1 结构/非结构网格系统

Hybrid Grid System

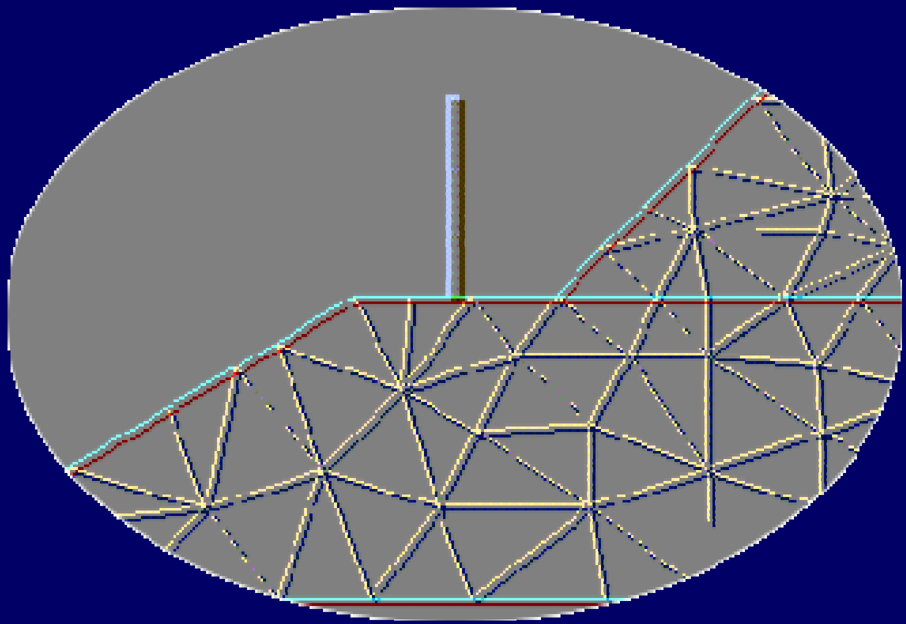
- 压力分布图

Pressure distribution

5.1 结构/非结构网格系统

Hybrid Grid System

- 系统网格处理复杂结构的能力－旗杆

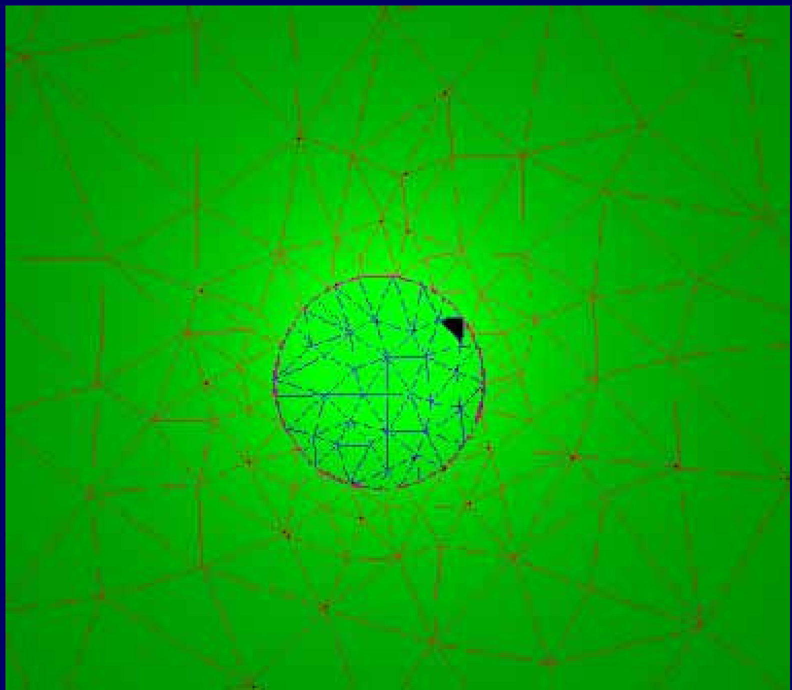


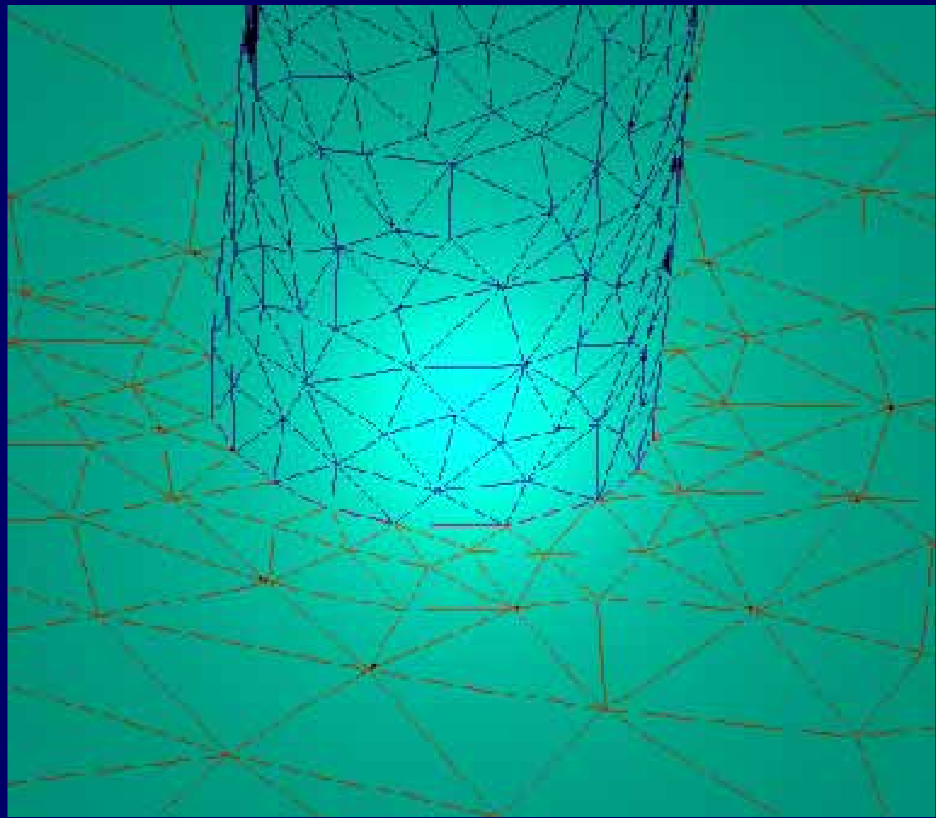
Complex geometry

5.1 结构/非结构网格系统

Hybrid Grid System

- 旗杆附近的网格结构





Grid structure

5.2 后部倾角对车身流场的影响

Flowfield of AHMED Wake

对AHMED 汽车模型后部详细流场进行了研究,探讨了车身后部倾角对流场及性能的影响. 计算结果和实验结果基本一致.

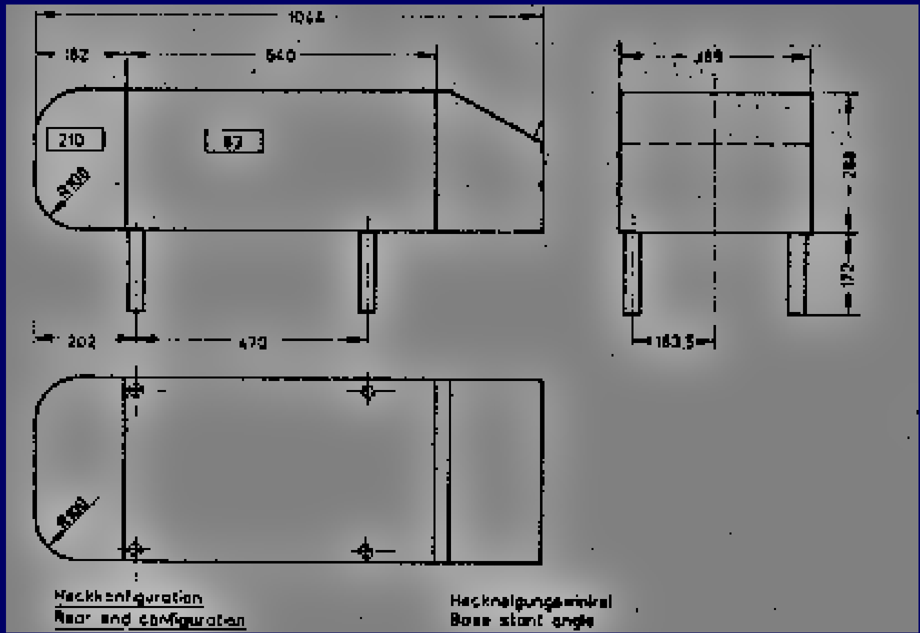
随着倾角的增加,三维效应影响越来越大,在后部倾角为25-33度出现气动阻力增加现象可能是由于三维效应和后部倾角耦合产生的,而非仅仅由后部倾角变化引起.

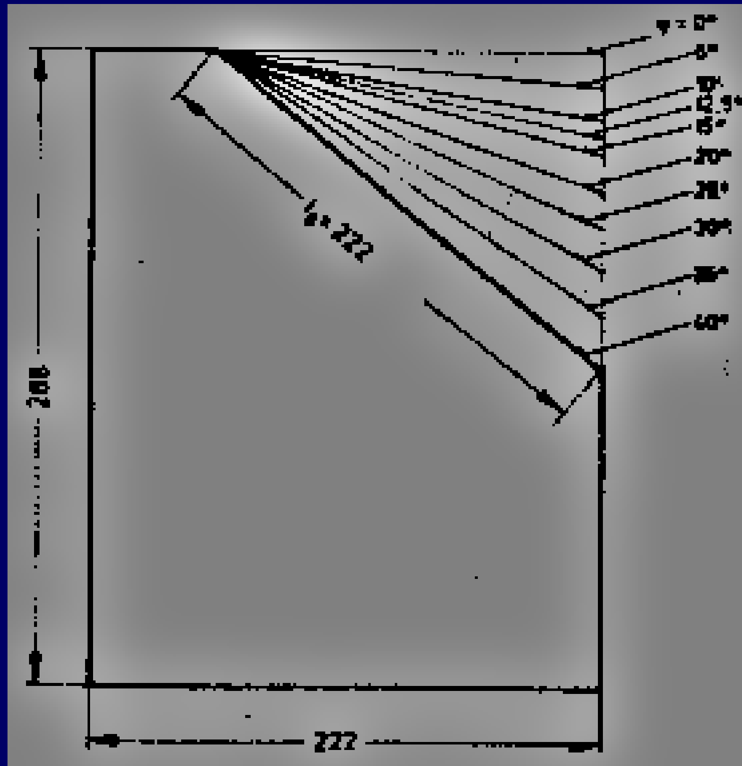
AHMED, a basic ground vehicle type of bluff body, the wake structure is analyzed. The results agree with the experimental.

5.2 后部倾角对车身流场的影响

Flowfield of AHMED Wake

❖ AHMED模型



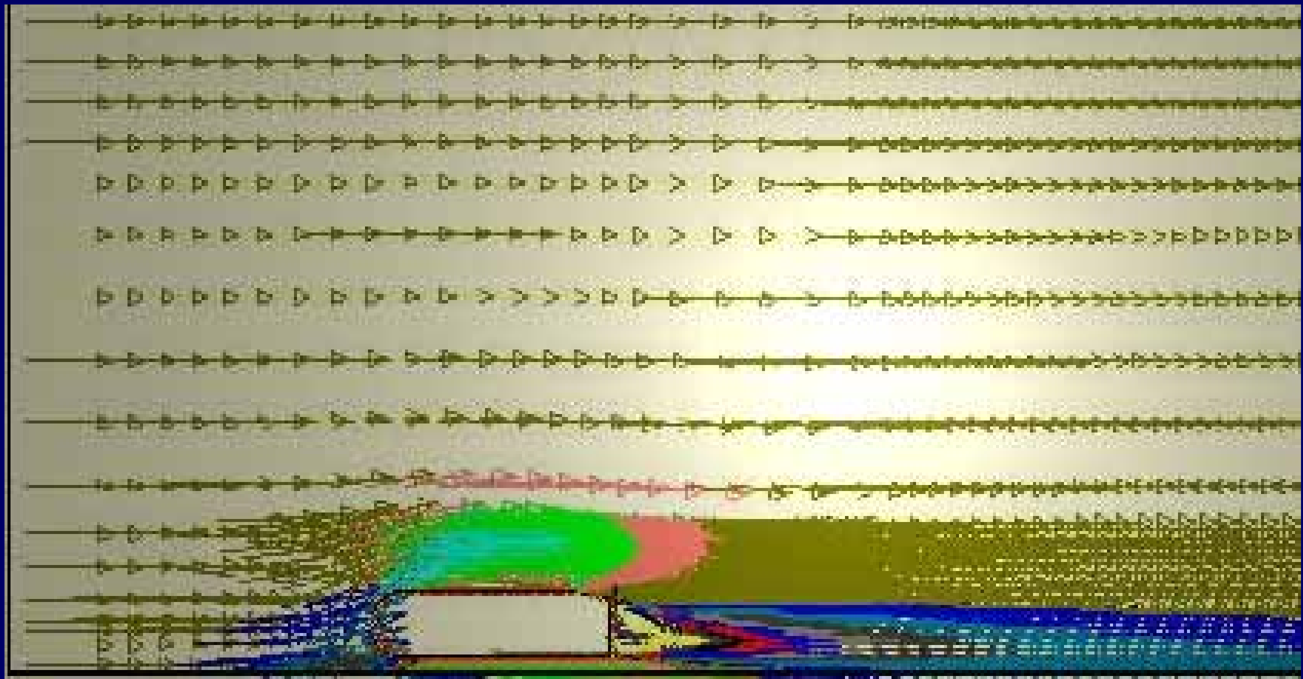


AHMED MODEL

5.2 后部倾角对车身流场的影响

Flowfield of AHMED Wake

- 12.5度全场速度矢量图

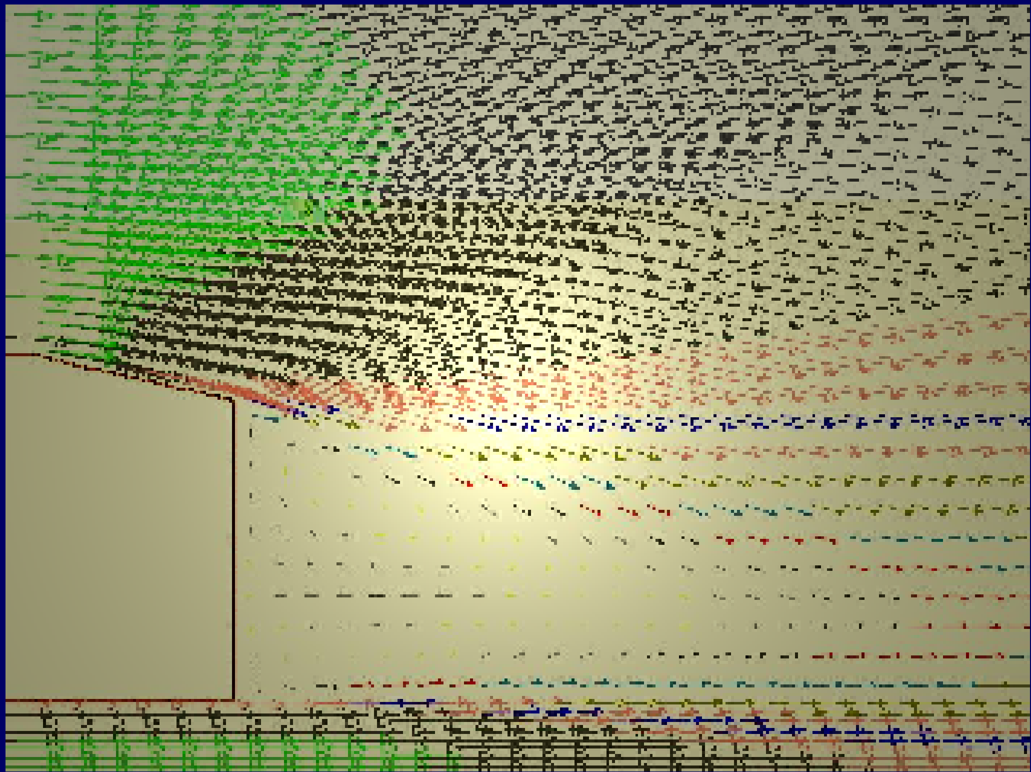


Base slant angle =12.5, velocity vectors

5.2 后部倾角对车身流场的影响

Flowfield of AHMED Wake

- 12.5度模型后部速度矢量图



Base slant angle =12.5, velocity vectors of model rear

5.2 后部倾角对车身流场的影响

Flowfield of AHMED Wake

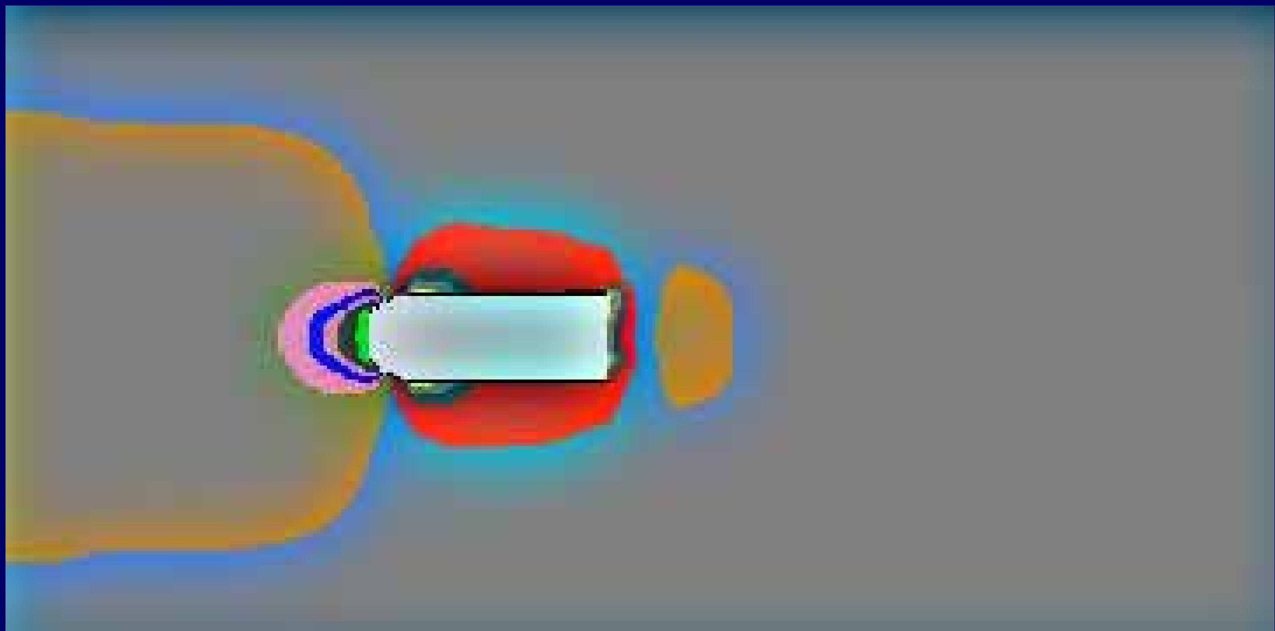
- 12.5度压力分布图

Base slant angle =12.5, pressure distribution

5.2 后部倾角对车身流场的影响

Flowfield of AHMED Wake

- 12.5度压力分布图

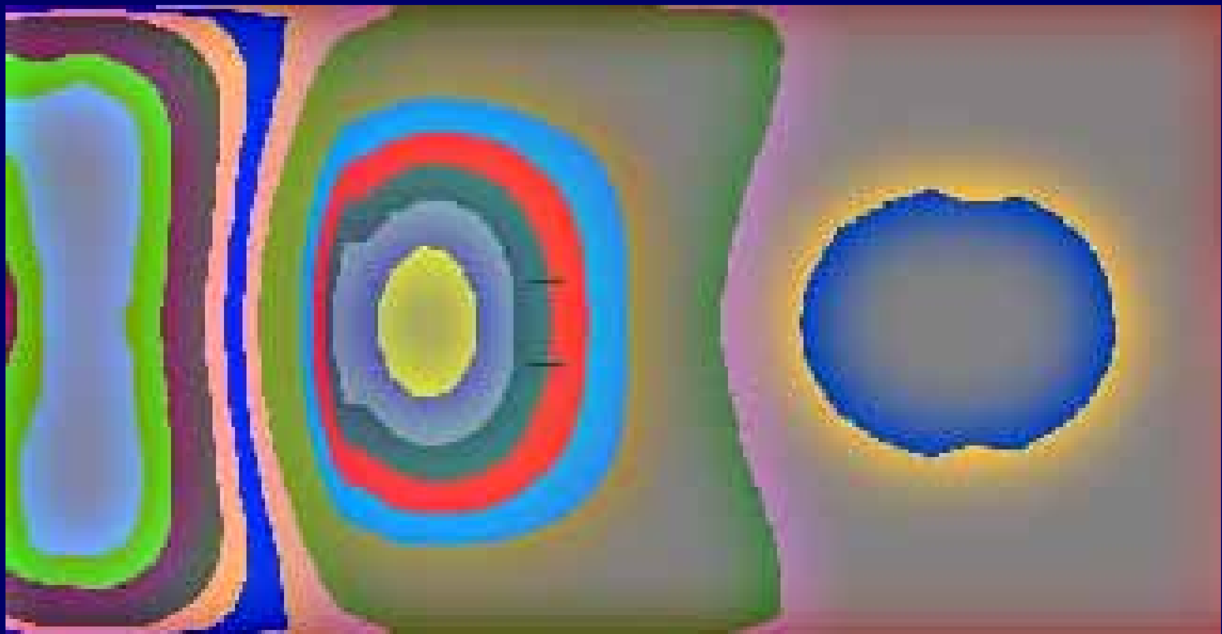


Base slant angle =12.5, pressure distribution

5.2 后部倾角对车身流场的影响

Flowfield of AHMED Wake

- 12.5度压力分布图



Base slant angle =12.5, pressure distribution

5.2 后部倾角对车身流场的影响

Flowfield of AHMED Wake

- 阻力系数仿真与试验值

后部倾角	试验值	仿真值	误 差
5.00	0.184	0.181	1.63%
12.5	0.175	0.173	1.14%

Aerodynamic drag values: experimental results, calculation results and the errors

5.3 离地间隙对车身流场的影响

Ground Effects

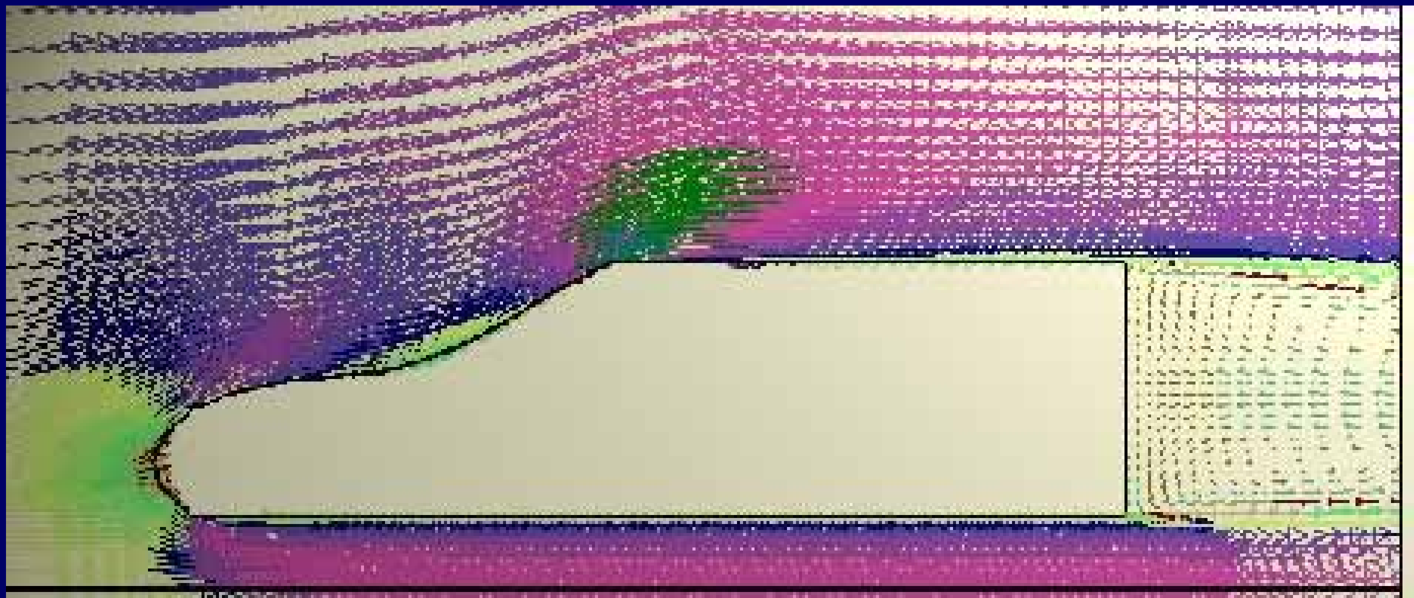
研究离地间隙对车身流场和气动性能的影响

计算结果正确反映了随离地间隙增大,汽车阻力系数曾非单调变化的趋势．而一般风洞实验数据均没能体现这一特征．德国鲁尔大学Papenfuss and Kronast对该实际生产用车（沙漠车）的流场采用移动模型技术进行了详细试验。

Using a two dimensional model car, the effect of the ground on the body pressure distribution and aerodynamic drag has been investigated.

5.3 离地间隙对车身流场的影响
Ground Effects

- 速度矢量图

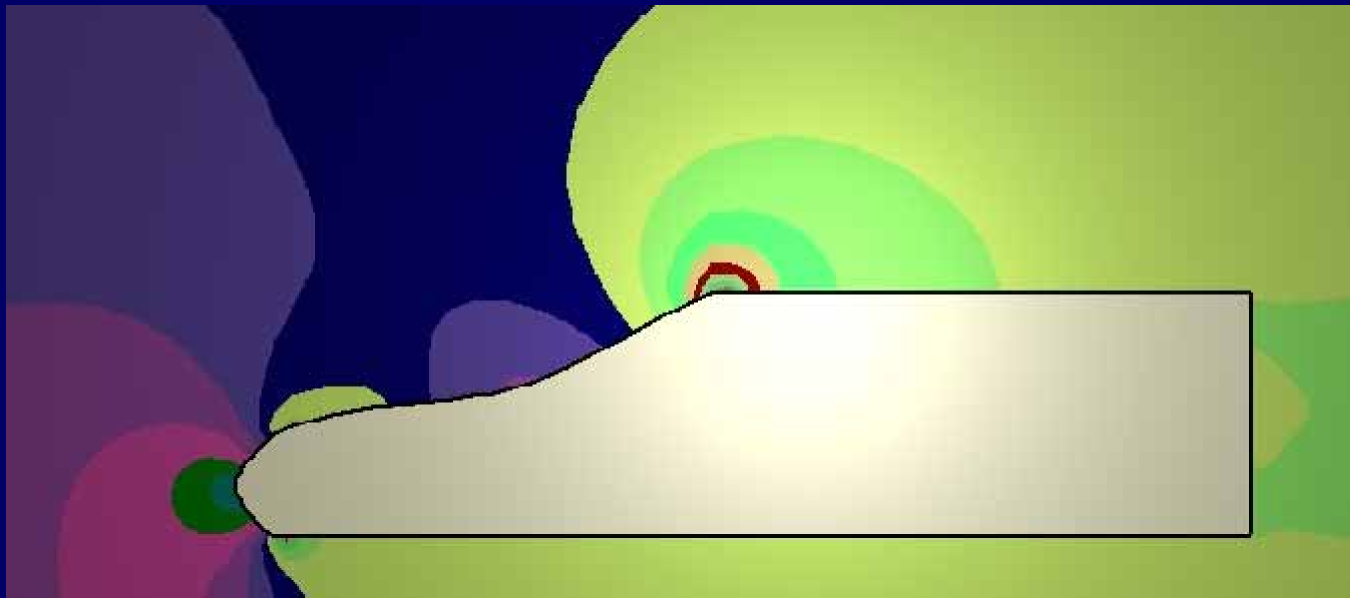


Velocity vectors

5.3 离地间隙对车身流场的影响

Ground Effects

- 压力分布图

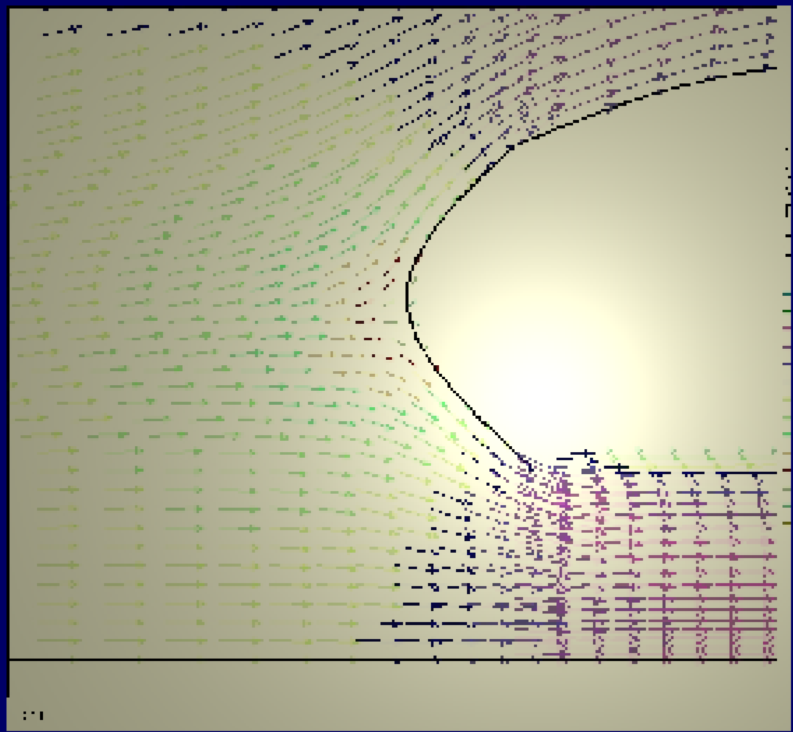


Pressure distribution

5.3 离地间隙对车身流场的影响

Ground Effects

- 前缘速度矢量图

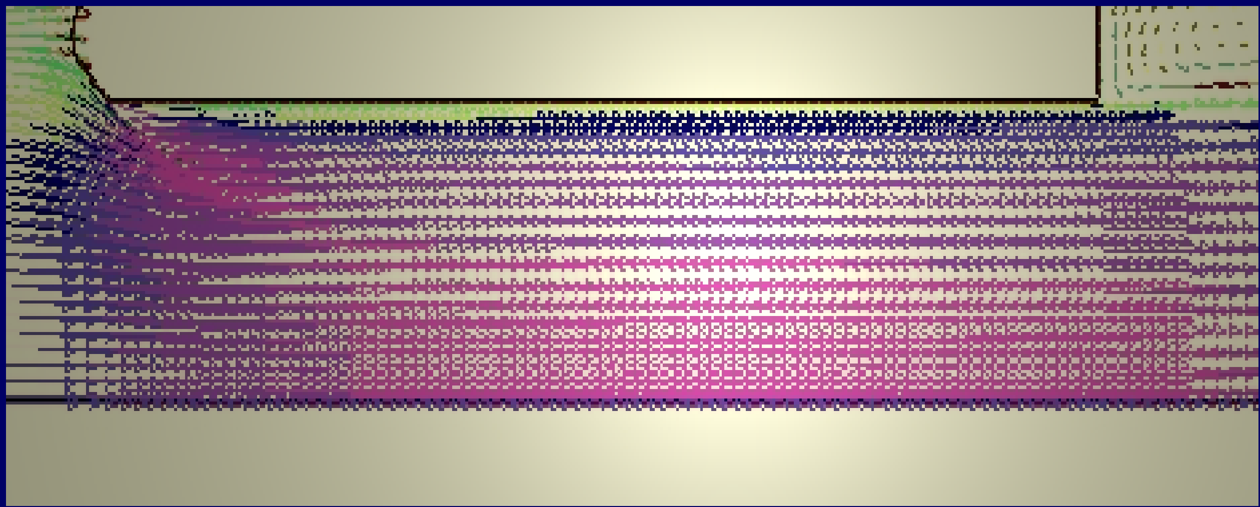


velocity vectors at model forepart

5.3 离地间隙对车身流场的影响

Ground Effects

- 底部速度矢量图

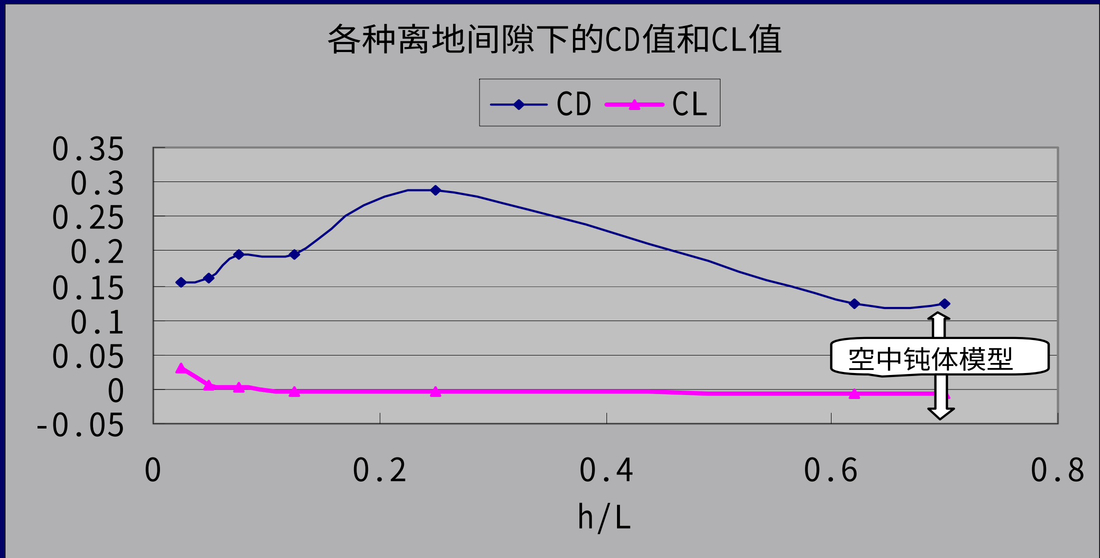


velocity vectors at model bottom

5.3 离地间隙对车身流场的影响

Ground Effects

❖ 气动阻力随间隙的变化



Aerodynamic force vs clearance

5.3 离地间隙对车身流场的影响

Ground Effects

❖ 离地间隙影响的试验研究

- 移动模型弹射系统

  消除了传统风洞技术对地面效应研究的限制

- 风阻随离地间隙的增加而增加

  在一定范围内，风阻存在随离地间隙的增加而增加的趋势。而现有传统的风洞技术（包括移动带技术）的试验结果为风阻随间隙增大而减小，均未能反映这一趋势。

Ground effects: Experimental results
a pneumatic high speed launch system for the study of automobile aerodynamics was built at the Ruhr-University Bochum. The drag is increased when the clearance is larger.

5.3 离地间隙对车身流场的影响

Ground Effects

仿真系统地面效应模型

- 阻力单调变化

在正常间隙内，风阻随离地间隙而增加。在较大间隙时，随间隙增大而减小

- 模型正确有效

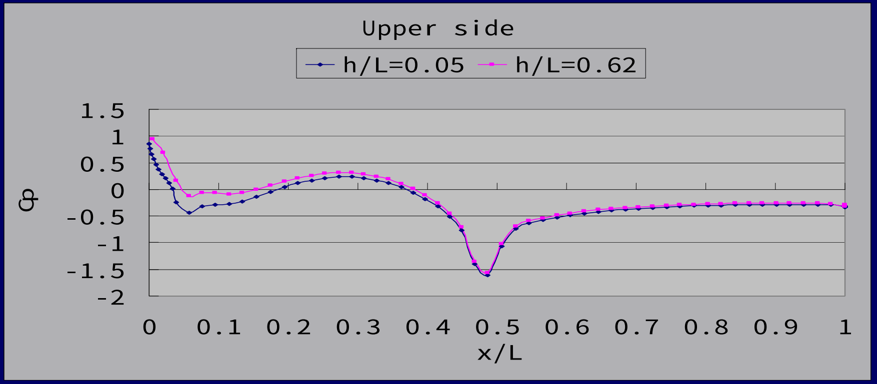
实际沙漠车的仿真结果和试验结果结论一致，说明模型能很好地模拟实际地面效应

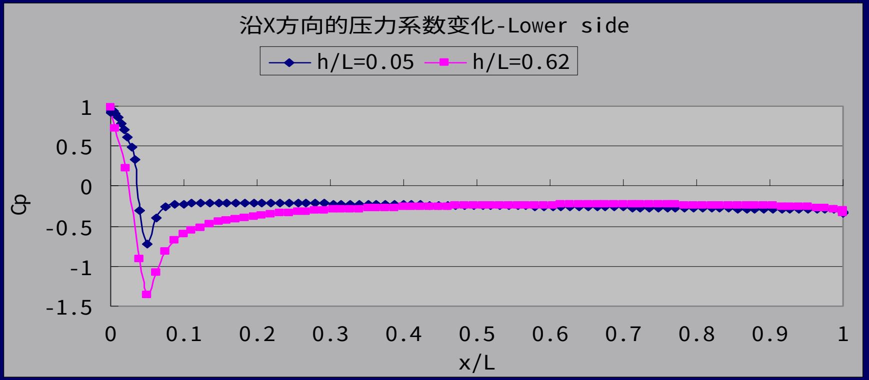
Ground effect model simulates the effect effectively

5.3 离地间隙对车身流场的影响

Ground Effects

❖ 不同离地间隙车身上下部压力系数仿真值





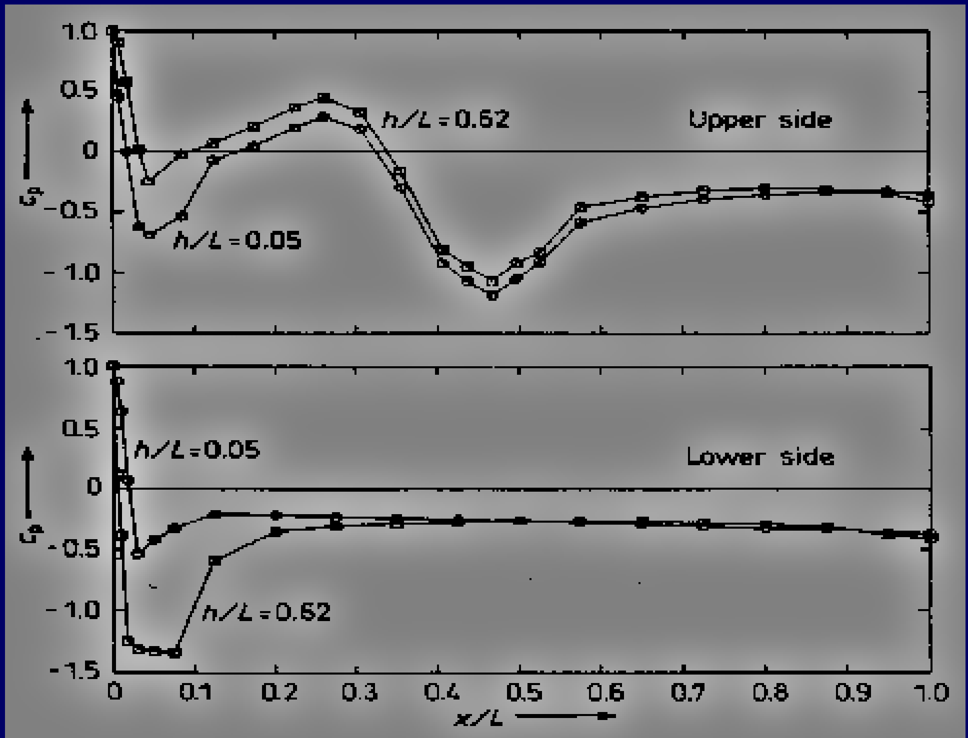
h/L=0.05代表汽车正常离地间隙，h/L=0.62已经接近忽略地面效应的离地间隙

Pressure distribution on the surface of the model simulation results

5.3 离地间隙对车身流场的影响

Ground Effects

❖ 不同离地间隙车身上下部压力系数试验值



计算结果与实验数据基本一致。汽车上表面，离地间隙大压力大，下表面则相反，离地间隙大压力变小

Pressure distribution on the surface of the model experimental results

5.4 转静相互作用

rotating/non-rotating interaction

转静相互作用

冷却风扇、油泵、扭矩转换器、刹车盘冷却等旋转机械内部复杂流动

系统应用

旋转效应，不同坐标系
网格自适应

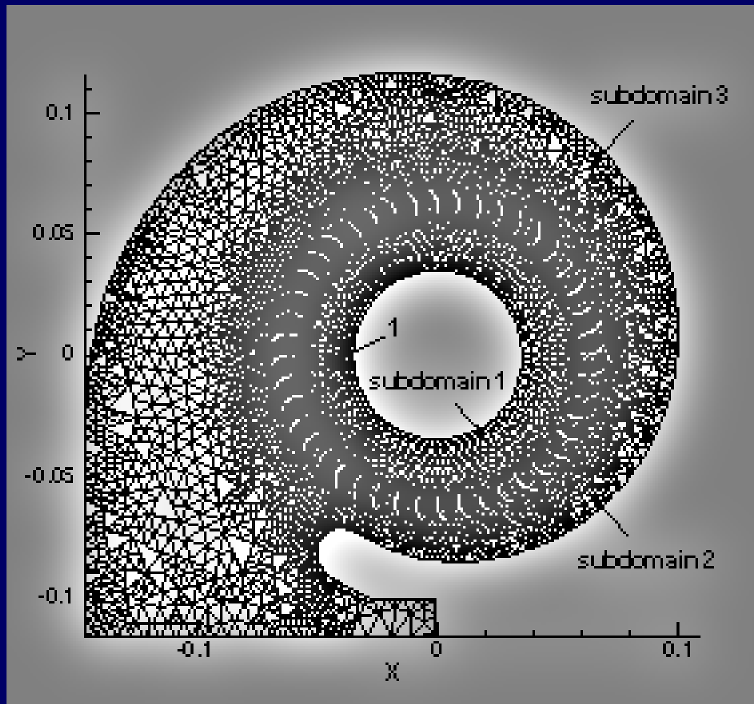
Cooling fans, blowers, torque converters, oil pumps, disk brake cooling are some of the areas involving rotating machinery.

Rotating effects, grid-adaptive

5.4 转静相互作用

rotating/non-rotating interaction

❖ 风机流场计算域



分区技术

不同坐标系

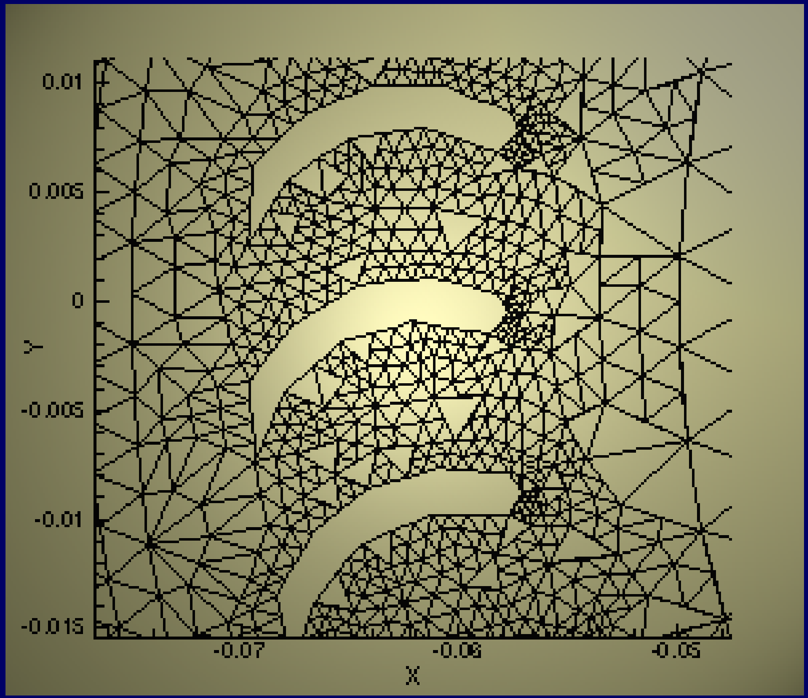
旋转效应

Calculation domain of centrifugal blower

5.4 转静相互作用

rotating/non-rotating interaction

❖ 网格自适应

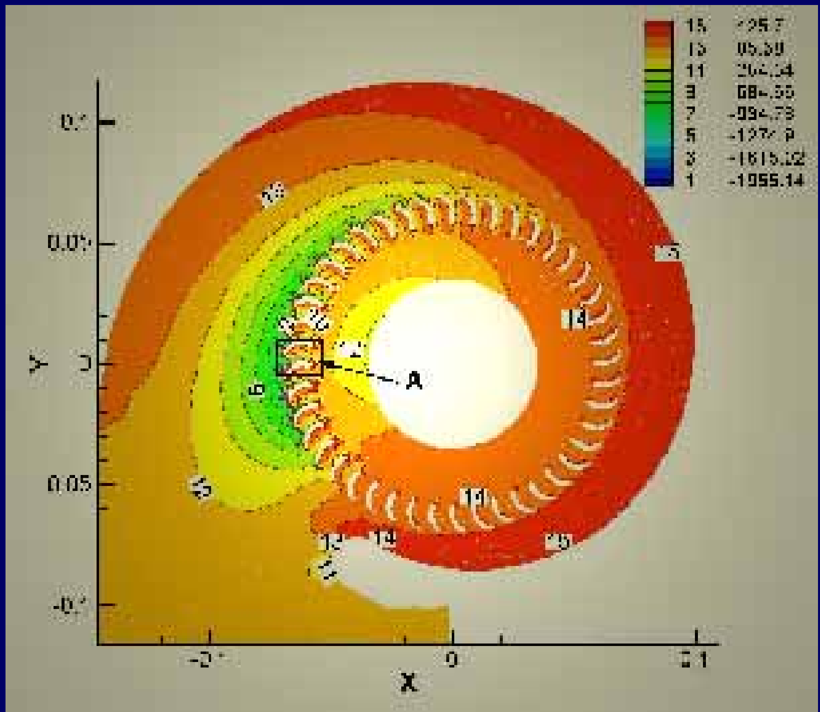


Solution-Adaptive Mesh

5.4 转静相互作用

rotating/non-rotating interaction

❖ 静压分布(Pin=200pa, n=261r/m.in)



蜗壳对压升的显著影响

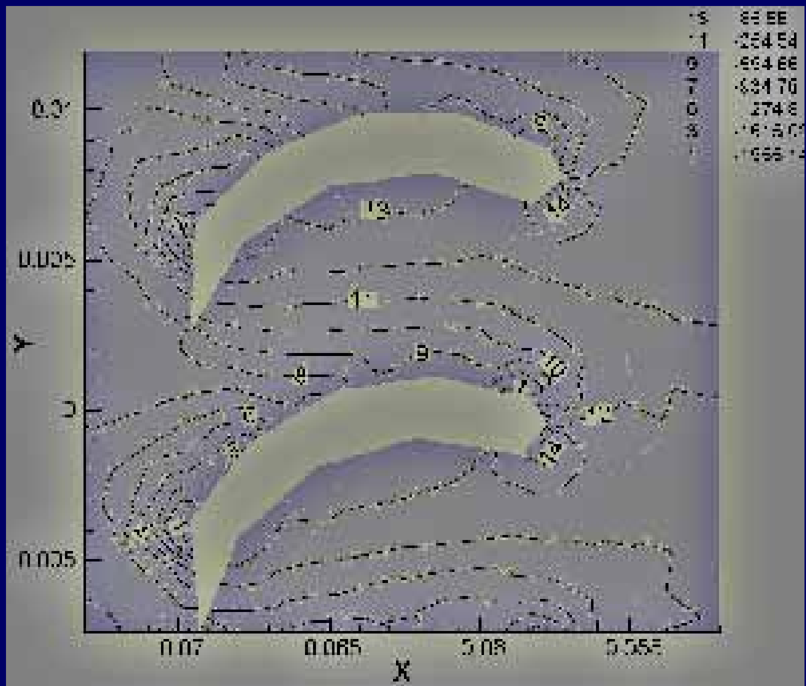
Static pressure distribution of whole in blade-to-diffuser domain(Pin=200pa, n=261r/m.in)

Most of the pressure rise occurs within the part far from the outlet duct, clearly revealing the effect of volute wall in achieving the pressure rise in centrifugal blowers.

5.4 转静相互作用

rotating/non-rotating interaction

❖ 区域A的放大



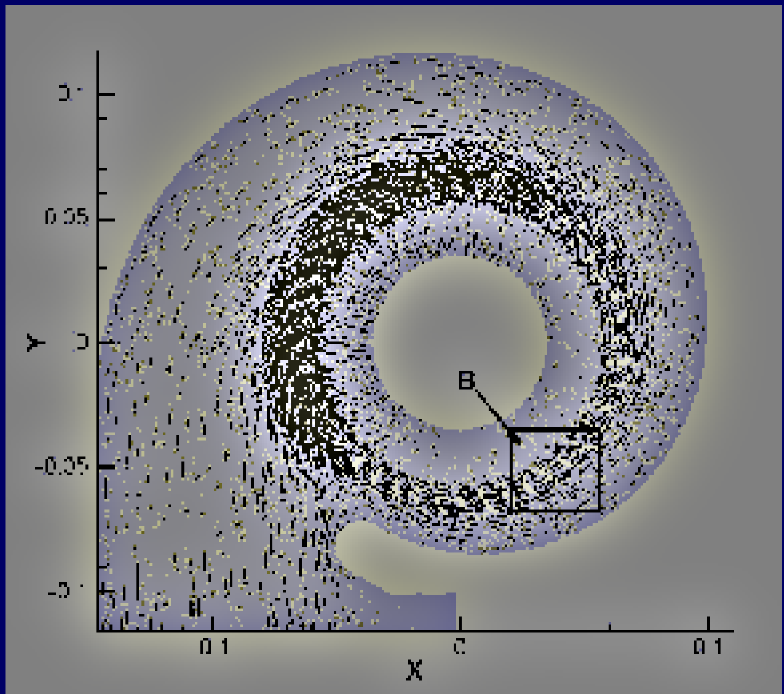
Part magnification of area A

The minimum section is usually designed to follow the streamline at that location. The static pressure increases along the blade from the leading edge to trailing edge. But the static pressure is decreasing along the blade passage in the maximum cross-sectional area because of strong secondary flow effects. However, the stagnation pressure is still increasing.

5.4 转静相互作用

rotating/non-rotating interaction

❖ 叶片至扩压器之间的速度矢量图



Because of the anuulus-wall boundary layer blockage, the relative velocity in the blade passages of the small casing tip clearance area is low, and a high-velocity jet comes out radially at the exit of the rotor, and it is then diffused in the vaneless diffuser. The result is achieved at considerable loss in stagnation pressure.

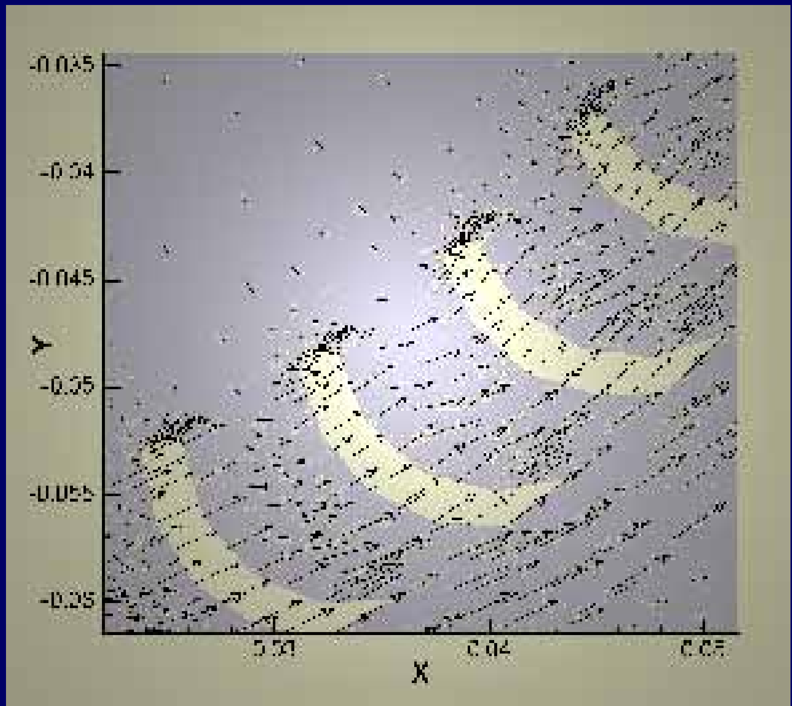
The flow in the volute is complex, and viscous effects dominate the flow .The major effect in a diffuser is the adverse pressure gradient that controls the boundary layer growth and separation.

Velocity distribution of whole in blade-to-diffuser domain

5.4 转静相互作用

rotating/non-rotating interaction

❖ B区域的速度矢量分布



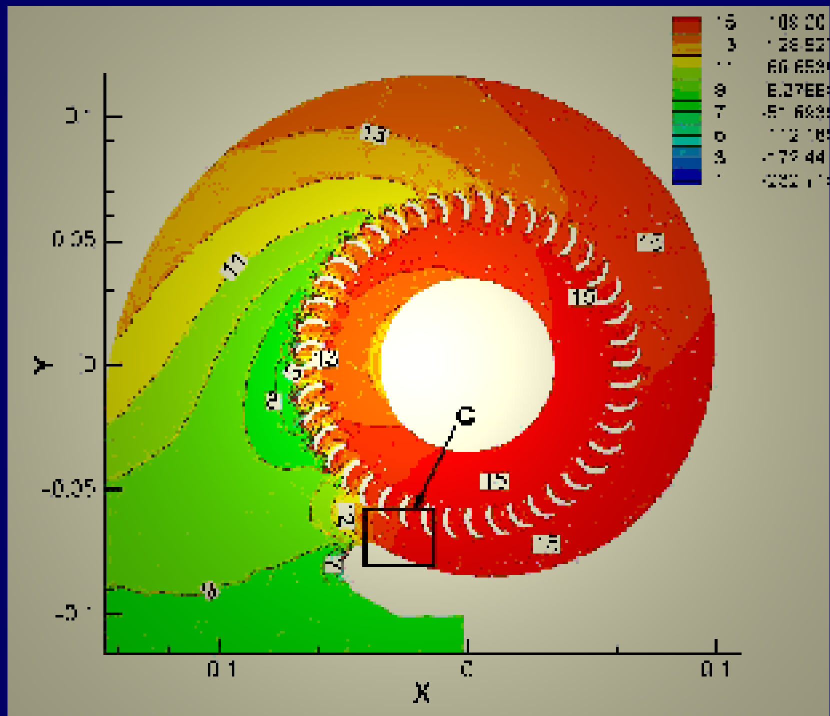
Velocity distribution of area B

There exists a vortex structure flow in the leading edge. And a wake type of profile (flow separation) existing in this region near the trailing edge results in the observed complex velocity profiles

Velocity distribution of area B

5.4 转静相互作用
rotating/non-rotating interaction

❖ 静压分布(Pin200pa, n=0r/min)



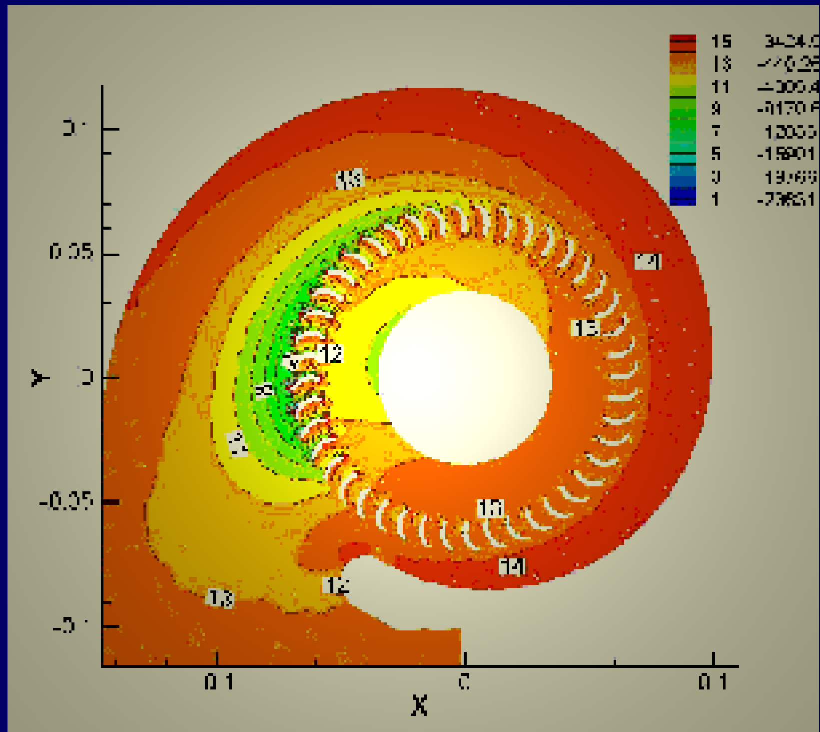
Static pressure distribution (Pin200pa, n=0r/min)

Without the blade rotation effect, it reveals it clearly that the non-symmetrical configuration leads to the non-symmetrical pressure distribution.

5.4 转静相互作用

rotating/non-rotating interaction

❖ 静压分布(Pin200pa, n=1000r/min)

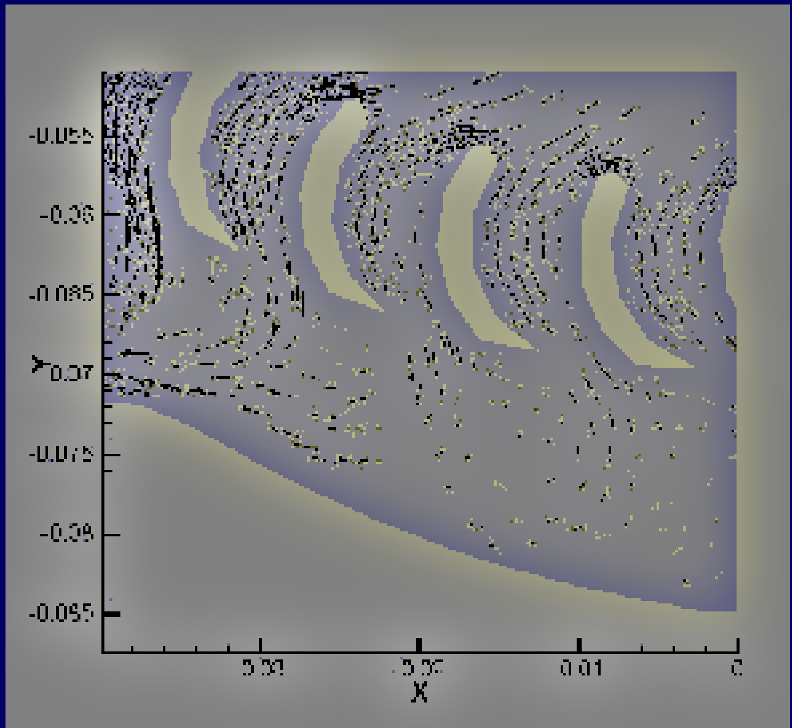


Static pressure distribution (Pin200pa, n=1000r/min)
the blade rotation will cause more serious unsymmetry flow structure and large pressure gradient

5.4 转静相互作用

rotating/non-rotating interaction

❖ 喉部区域速度矢量分布



In the region, this complex, perturbed, and nonuniform flow field is influenced by viscous and turbulence mixing effects. The boundary layer is no longer of the conventional type, and the terminnology “annulus-wall boundary layer” is misleading.

And when the rotating speed increase, the vortex structure will disappear because of the periodic perturbations and the scraping caused by the rotor blade. The rotation effects make the trailing surface boundary stable.

Velocity distribution in throat area

6 总结与展望

Conclusions and Open features

初步建立平台

在清华大学为设成世界一流大学的985学科规划重点项目“数字化轿车”和“轿车关键技术研究”,以及国家自然科学基金项目“工程分离流动结构及旋涡特性”的支持下，以STAR－CD和ICEM－CFD为核心软件，已初步建立了汽车空气动力学仿真分析平台。

VASS system

6 总结与展望

Conclusions and Open features

◆专业性：能够有效模拟复杂的三维汽车流场的粘性湍流流动。应用已取得较好成果。

◆通用性：能对于不同车型生成自适应网格，并可模拟多种工况条件，参数设置方便

◆平台具有良好的操作性和可扩容性

specialized in vehicle aerodynamics simulation, automatic adaptive grid system generation, suitable to extensive situations,easy to operate

VASS system

6 总结与展望

Conclusions and Open features

平台主要特色

1. 求解域分区
2. 结构/非结构网格
3. 壁面半结构网格、自适应网格
4. 定解条件可重复应用
5. 地面效应模型
6. 专业程度高
7. 实用性好
8. 一定程度自动化

VASS system

6 总结与展望

Conclusions and Open features

将来研究

工程应用
预先处理(几何建模和网格生成)
旋涡分离流动模拟
非定常流动问题
数值仿真程序验证
气动噪声

Applications in industry
pre-processing
vortex flow and separated flow
unsteady flow
CFD code validation
aeroacoustics