ANSYS

IDAJ CAE Solution Conference

Your True Partner for CAE&CFD

ICSC 2015

比想象中更强大!

IDAJ-China联合ANSYS-China

共同开创中国CAE&CFD新未来

IDAJ ANSYS

ANSYS产品咨询:

Email: ansys@idaj.cn

Tel: 010-65881497; 021-50588290

ANSYS CFD汽车行业应用

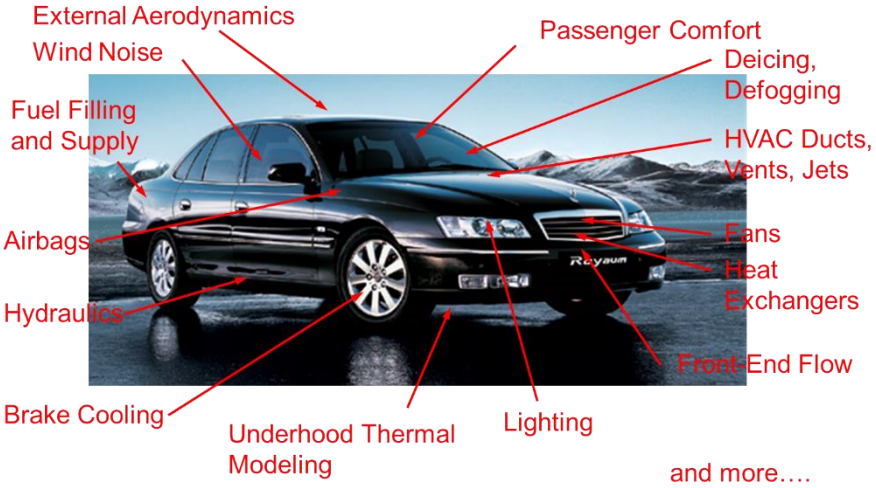
ANSYS CHINA

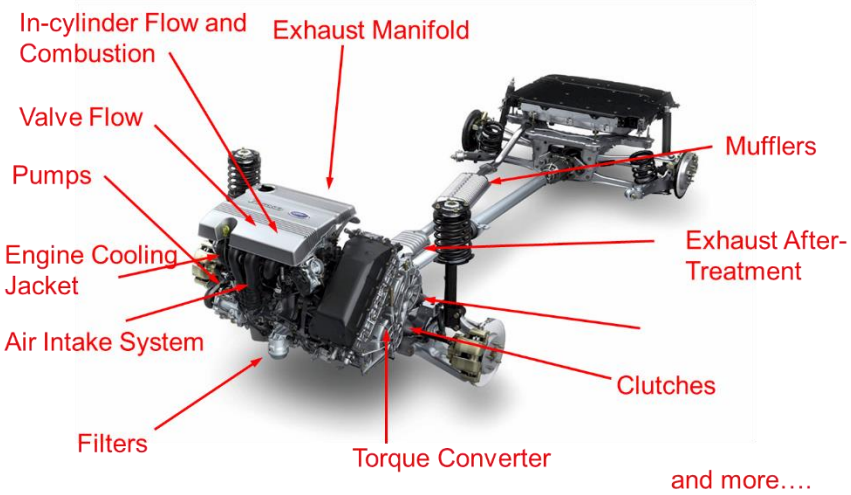
朱祥德

汽车行业中的CFD技术应用

■汽车行业是最早应用CFD技术指导工程设计的行业之一

■汽车行业的CFD遍及整车、子系统、零部件；动力系统，非动力系统





ANSYS CFD 汽车行业应用新发展

■强化功能

- ➢空气动力学
- ➢气动外形优化
- ➢气动噪声分析
- ➢整车热管理分析
- ➢尘/雨/雪/水/油多相流分析
- ➢流/固/热/电磁多场耦合分析

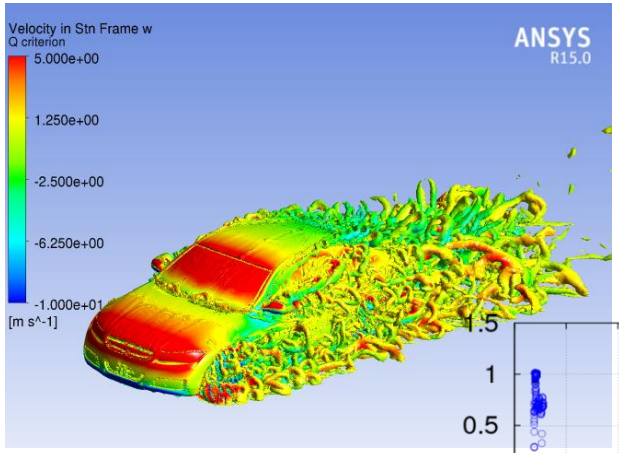
■缩短周期

- ➢网格生成更容易
- ➢求解器效率更高
- ➢降阶模型
- ➢GPU加速求解
- ➢HPC并行加速求解
- ➢自动化分析流程

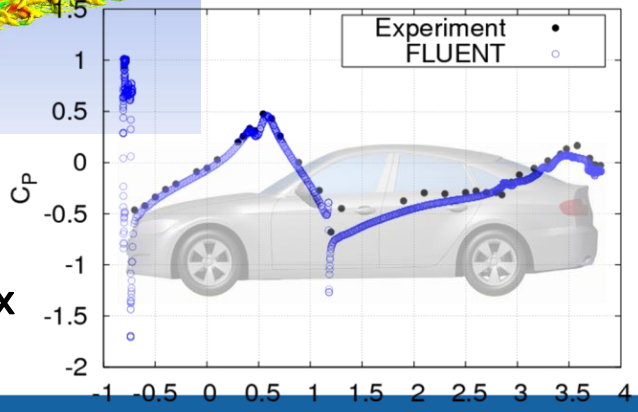
丰富的湍流模型保证气动求解精度

丰富的湍流模型

- SAS (second generation URANS)
- DDES / IDDES
- ELES / WMLES S-Omega

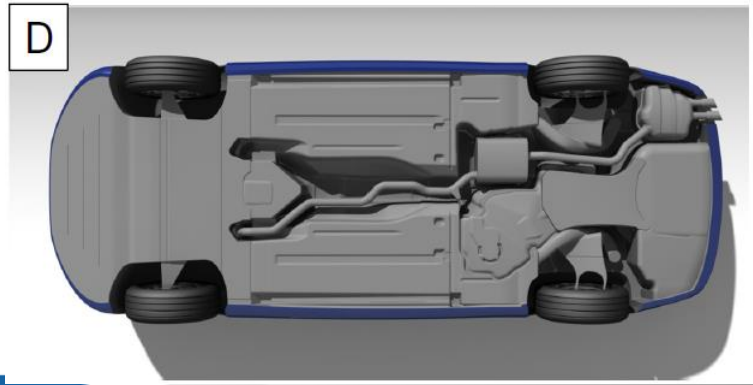


ELES box



DrivAer project

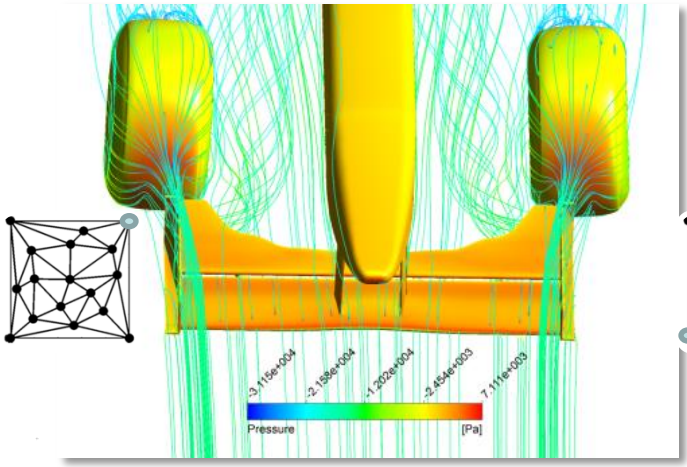
Courtesy of TU Munchen

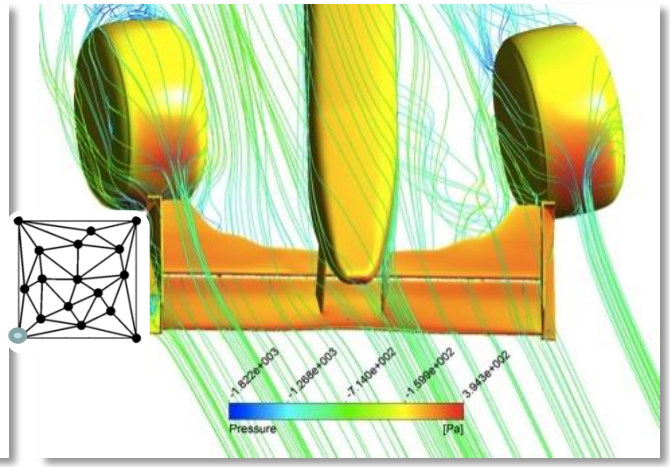


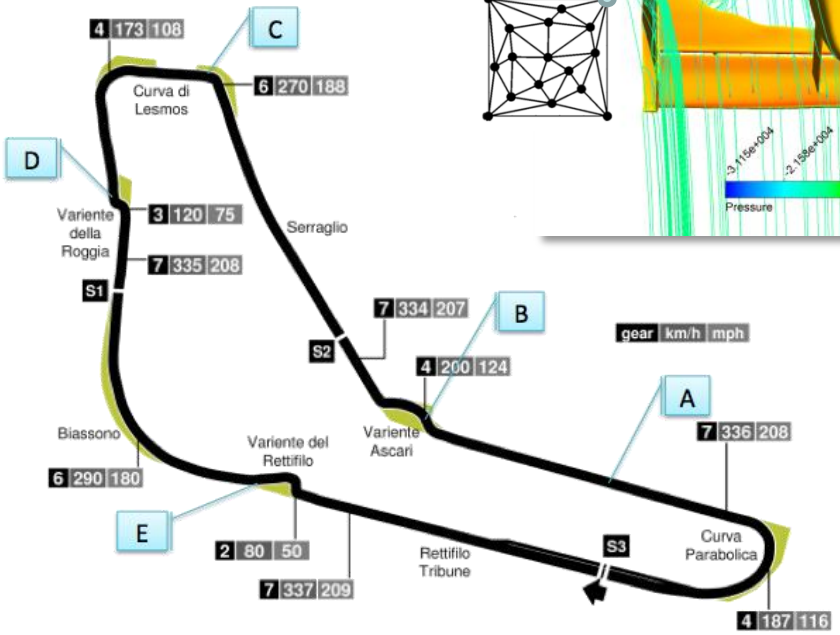
基于网格变形的气动外形优化功能

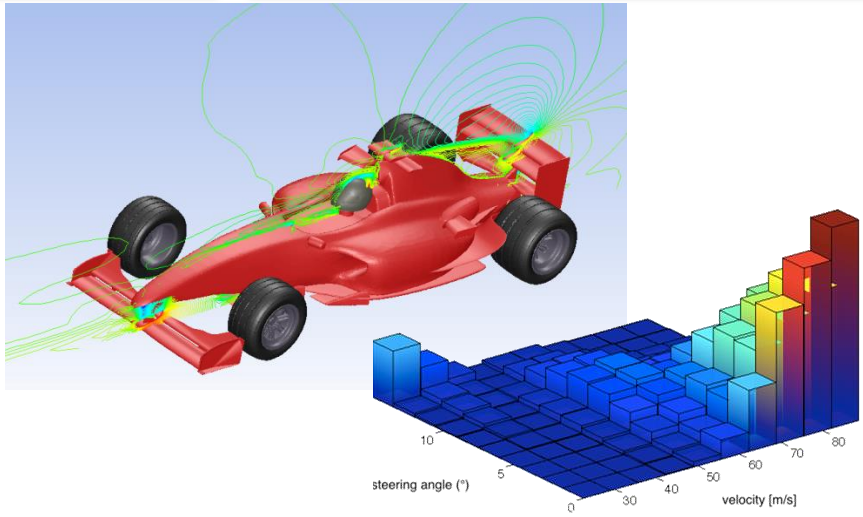
提高气动外形优化功能的鲁棒性

- Adjoint
- Mesh Morphing









Robust design optimization

丰富的气动声学仿真模型

丰富的气动声学仿真模型以满足不同需求

气动声学

稳态

宽频
1. Proudman's Formula
2. Boundary Layer Noise
3. Jet Noise

SNGR
1. LEE
2. Lilley

Fan Noise
1. Lowson
2. Gutin

瞬态

计算气动声学(CAA)
1. LES
2. DES
3. Transient RANS

Ffowcs-Williams & Hawkings Model (FWH)

Surface FFT (wall dB Map)

耦合

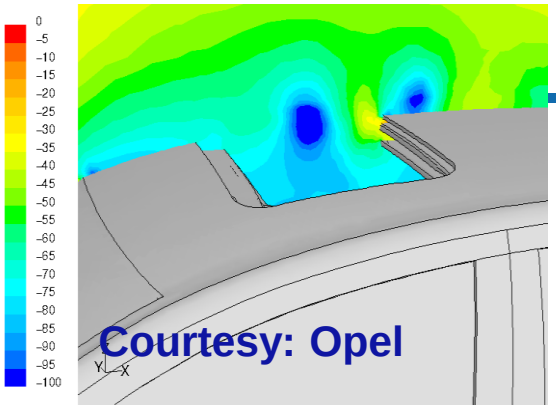
耦合第三方声学传播分析工具
1. LMS Virtual.Lab Acoustics
2. MSC Actran
3. CFS ++

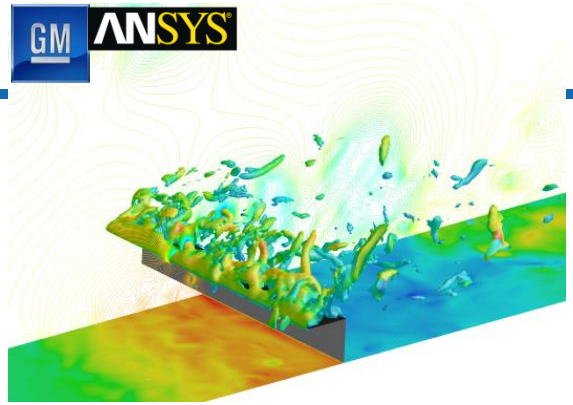
耦合ANSYS Mechanical Acoustics (ANSYS Fluent + MAPDL Acoustics Coupling)

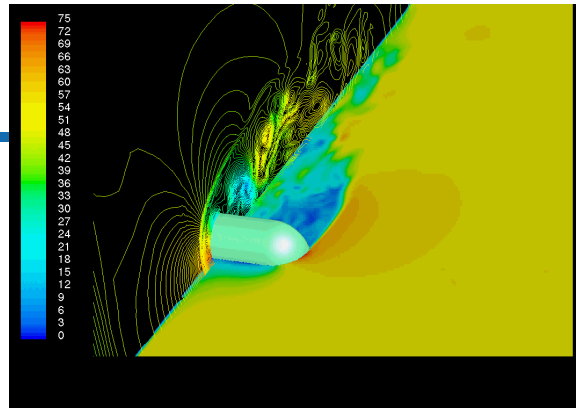
振动声学

分析类型
1. Modal Analysis
2. Harmonic Response
3. Transient Analysis
4. Response Spectrum

振动声学耦合
1. 2-way (strong) Structures Acoustics
2. 1-way (weak) Structures Acoustics



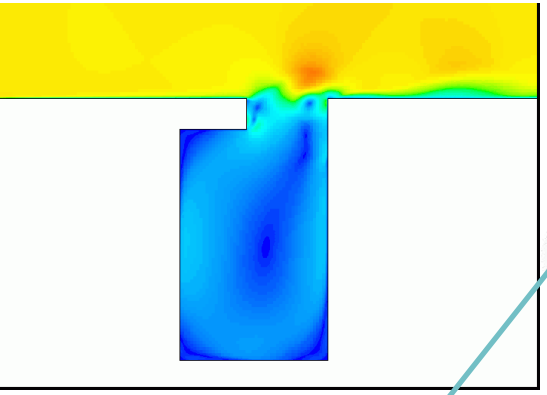


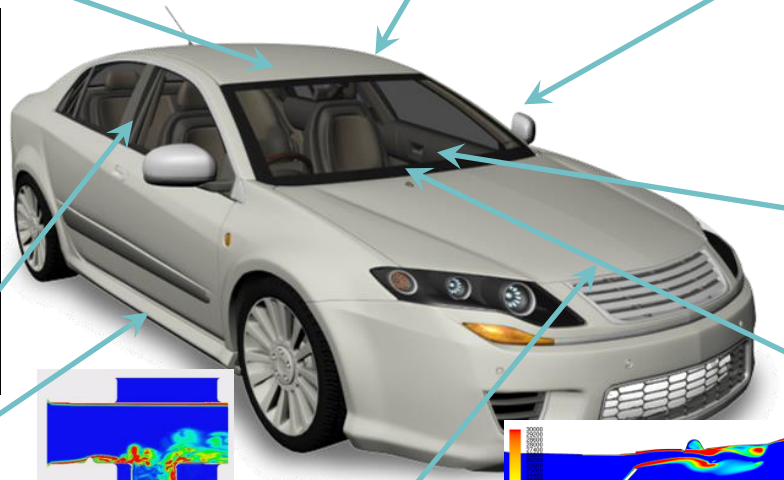


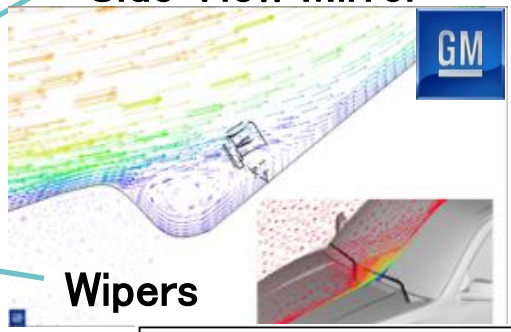
Sunroof

Rain Gutter

Side View Mirror

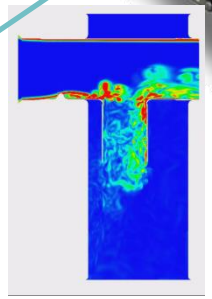






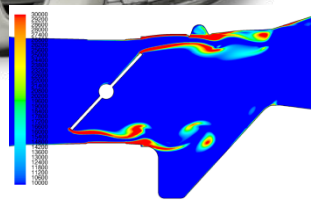
Wipers

Door Gaps

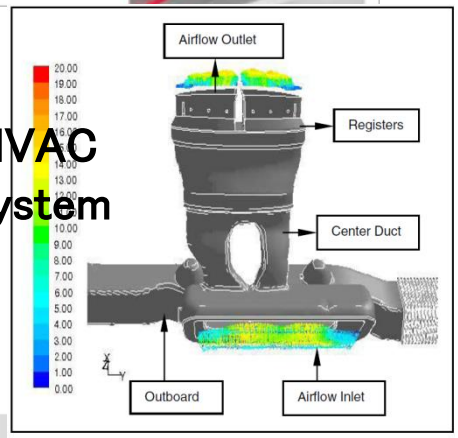


Exhaust System

Intake System



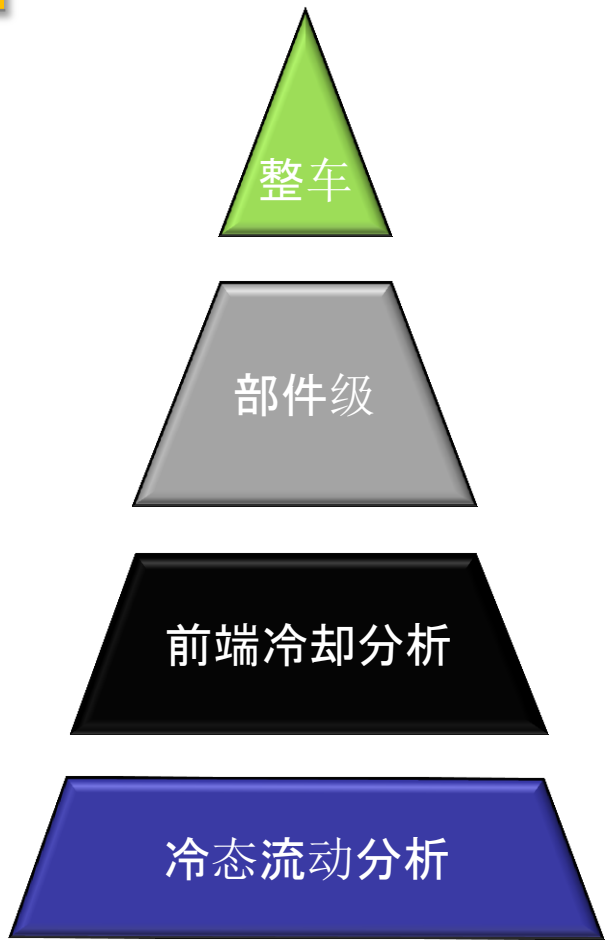
HVAC System



整车热分析功能

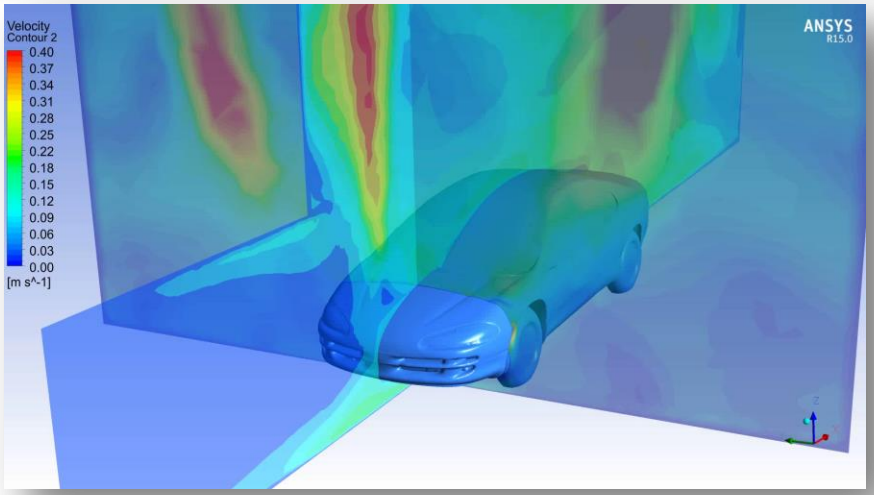
多种模型提高整车热分析功能

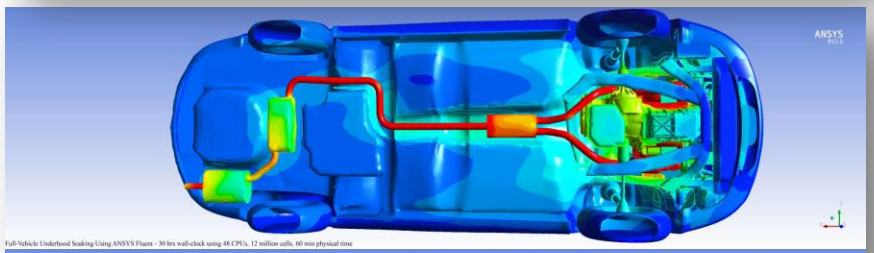
- 多孔介质 (处理换热器和格栅)
- Dual Cell和Macro based的换热器模型
- 风扇模型 (MRF, plane fan, sliding mesh)
- 单层和多层的壳导热模型
- 排气系统壁面设定修正(边界处理)
- 流固耦合换热 (包含固体导热)
- 映射网格交界面技术
- 快速的基于角系数的辐射模型
- 统一模型

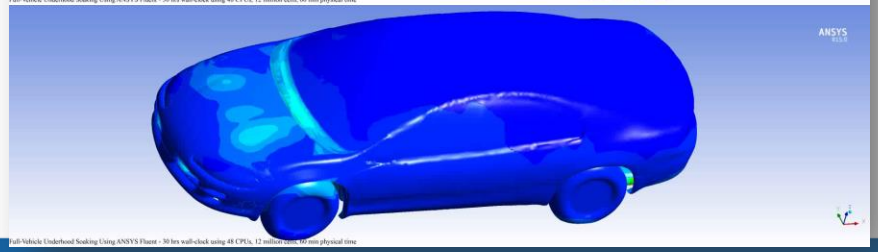


整车热分析功能

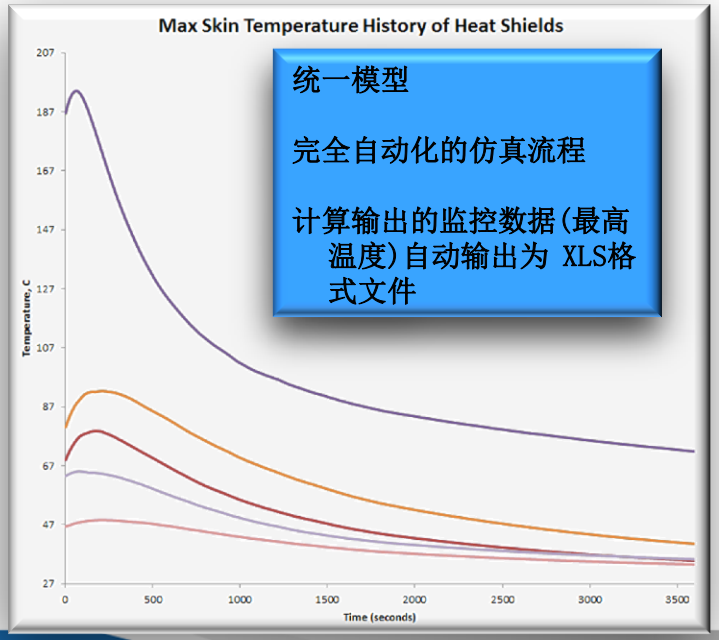
多种模型提高整车热分析功能







- 30 Hrs wall-clock,
- 48 CPUs



丰富的多相流模型

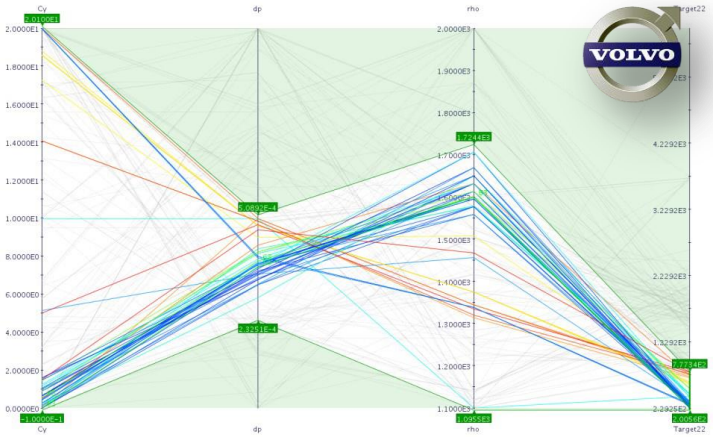
丰富的多相流应用

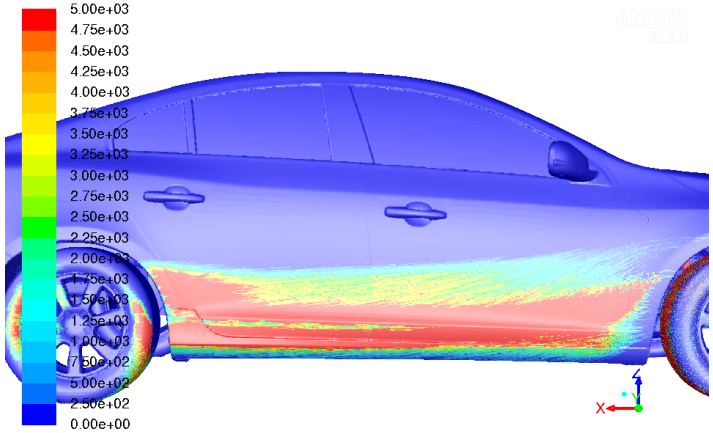
- ■ 动力系统
  - ➢ Piston Cooling
  - ➢ Boiling in Cooling Jacket
  - ➢ Cavitation in Water Pump
  - ➢ Fuel Injector
  - ➢ Lubrication HVAC System
  - ➢ Two-Phase Heat Exchangers
  - ➢ Oil Separation
- ■ 乘员舱流动
  - ➢ Window Deicing
  - ➢ Window Defogging
- ■ 燃油系统
  - ➢ Tank Filling
  - ➢ Fuel Sloshing
  - ➢ Fuel Vapor Emissions Transmission System
  - ➢ Clutch Performance
- ■ 外流
  - ➢ Tire Splashing / Hydroplaning
  - ➢ Rain Water Management
  - ➢ Windshield Wiper Performance
- ■ 制造过程
  - ➢ Spray Painting

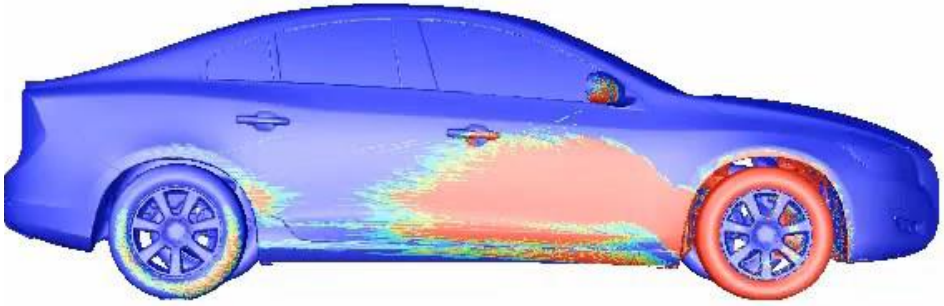
丰富的多相流模型

丰富的多相流应用

DOE优化污染面积



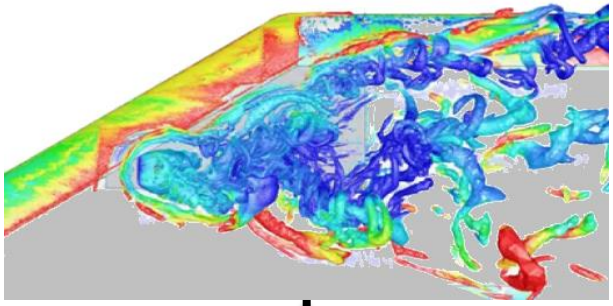


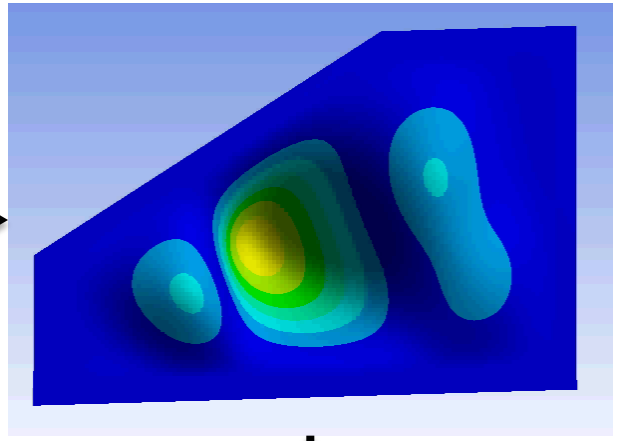


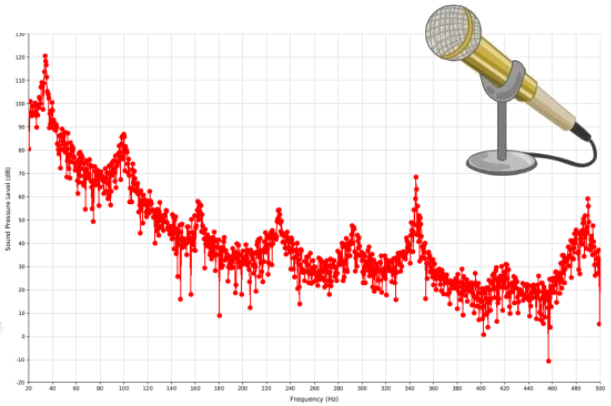
Workbench多场耦合分析功能

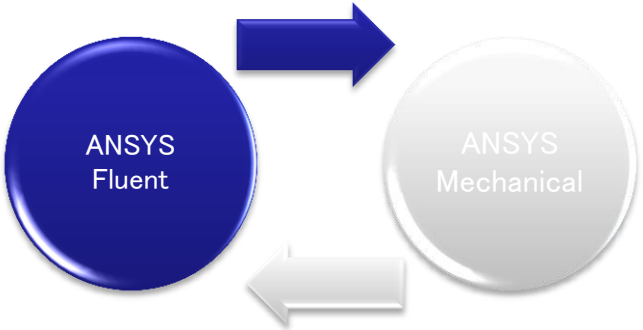
气动-振动-噪声耦合分析

空气动力学分析 → 结构振动分析 – 窗，门 … → 舱内噪声分析



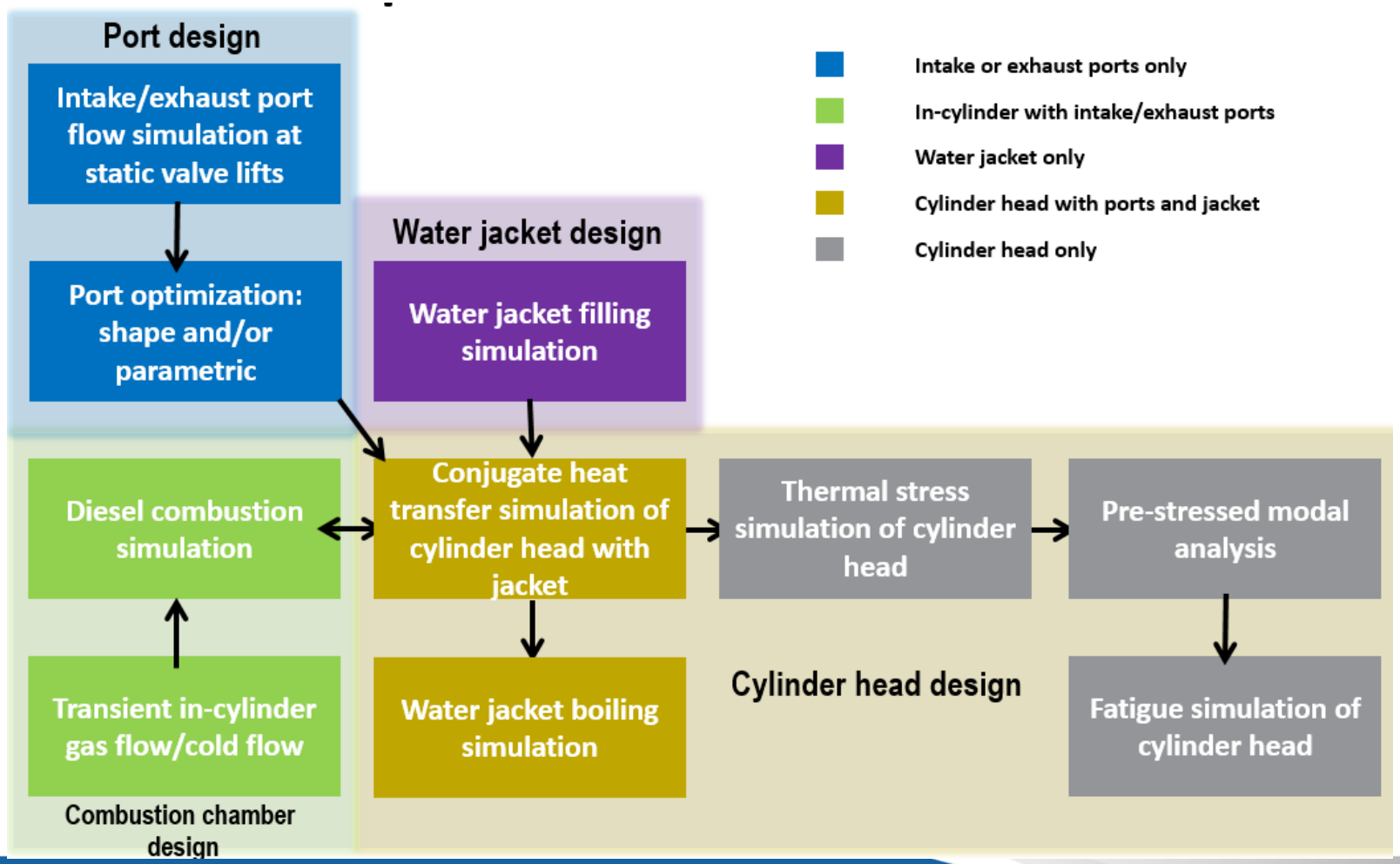






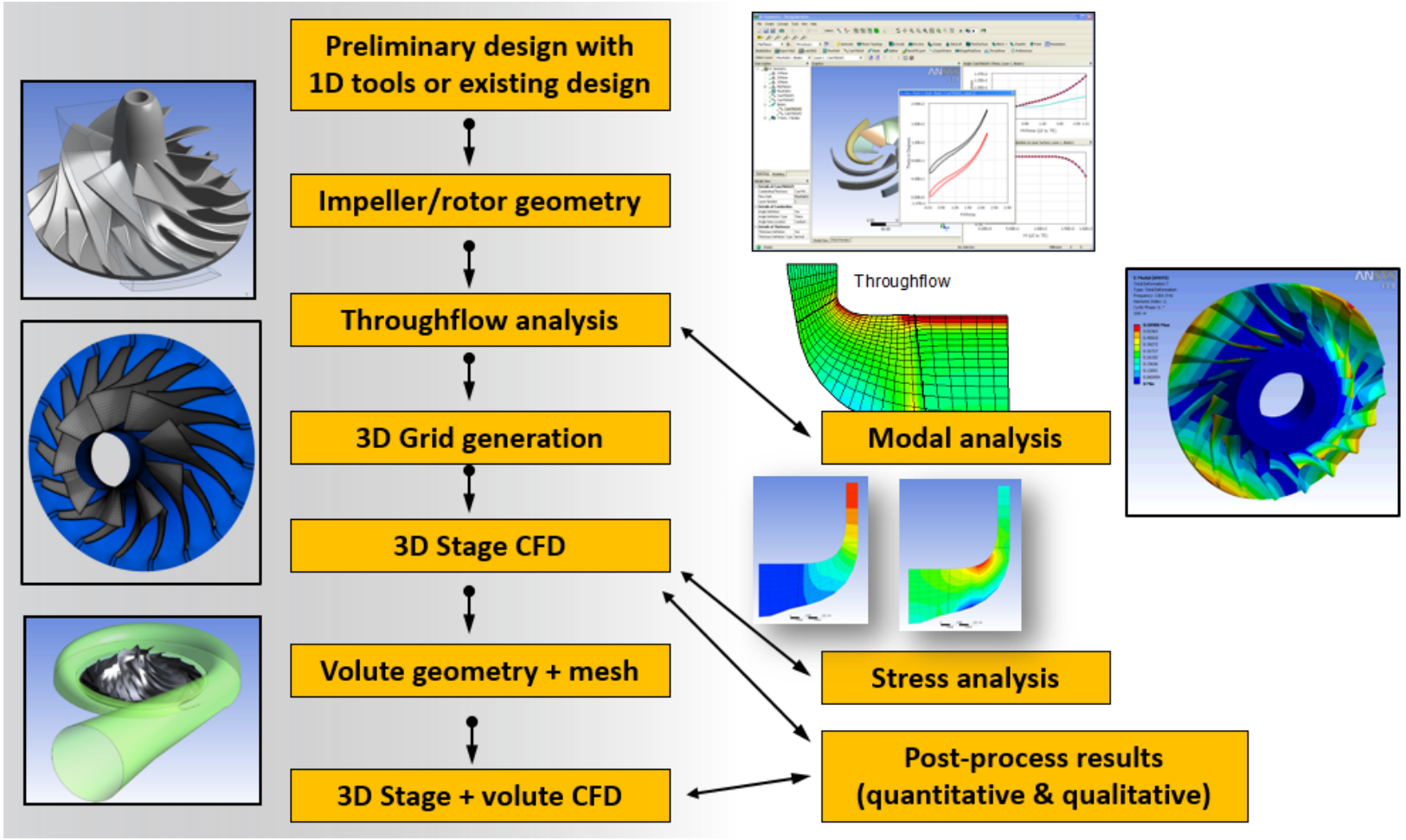
Workbench多场耦合分析功能

发动机流动-燃烧-结构耦合分析



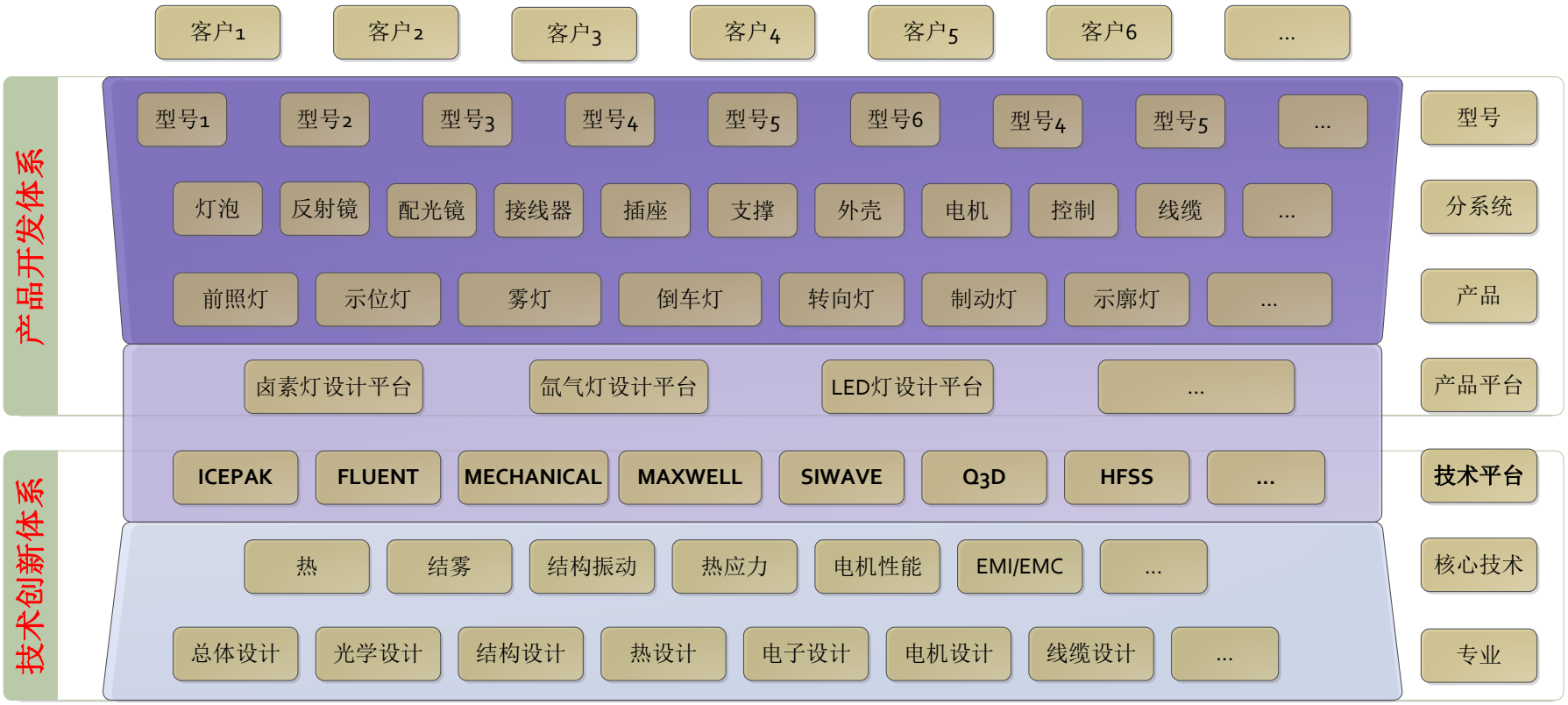
Workbench多场耦合分析功能

涡轮增压器设计及流固耦合分析



Workbench多场耦合分析功能

车灯流体-热-电磁-结构耦合分析



ANSYS CFD 汽车行业应用新发展

■强化功能

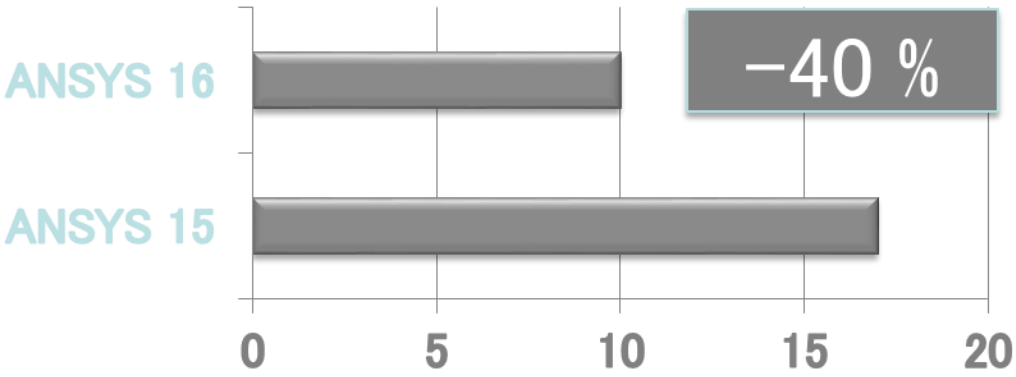
- ➢空气动力学
- ➢气动外形优化
- ➢气动噪声分析
- ➢整车热管理分析
- ➢尘/雨/雪/水/油多相流分析
- ➢流/固/热/电磁多场耦合分析

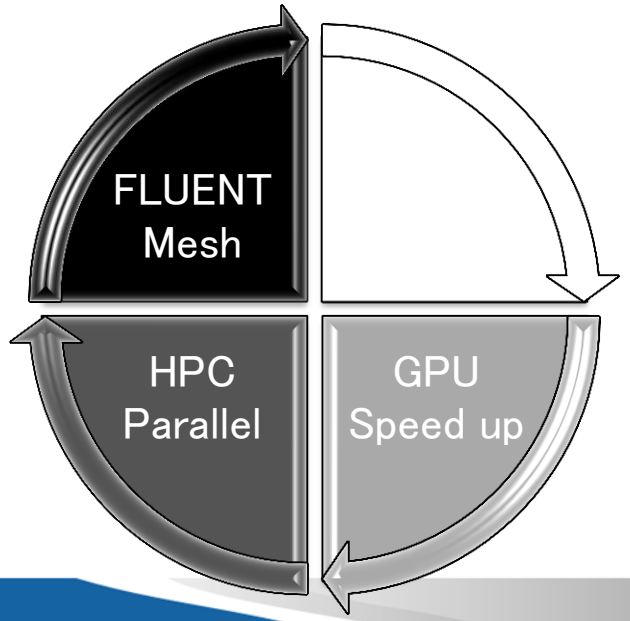
■缩短周期

- ➢网格生成更容易
- ➢求解器效率更高
- ➢降阶模型
- ➢GPU加速求解
- ➢HPC并行加速求解
- ➢自动化分析流程

从各个阶段提高仿真效率

Version	V15	V16
CAD & PLM	4 d	2 d
Mesh	5 d	3 d
Pre-process	3 d	1 d
Solve	2 d	2 d
Analyse	3 d	1d





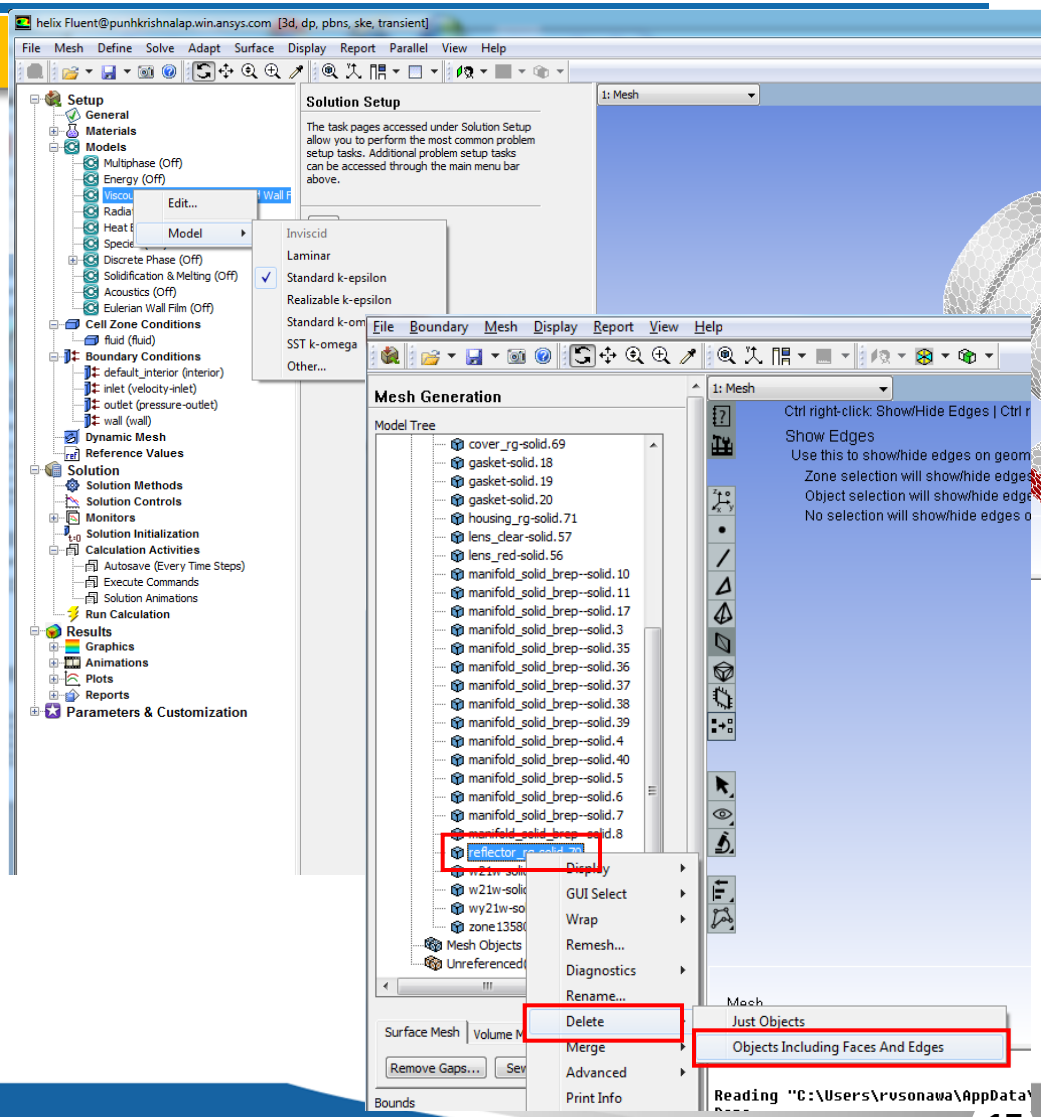
ANSYS Fluent前处理解决方案

灵活的前处理

- 全新的前处理用户环境

R16 采用新的基于树形结构的用户交互界面

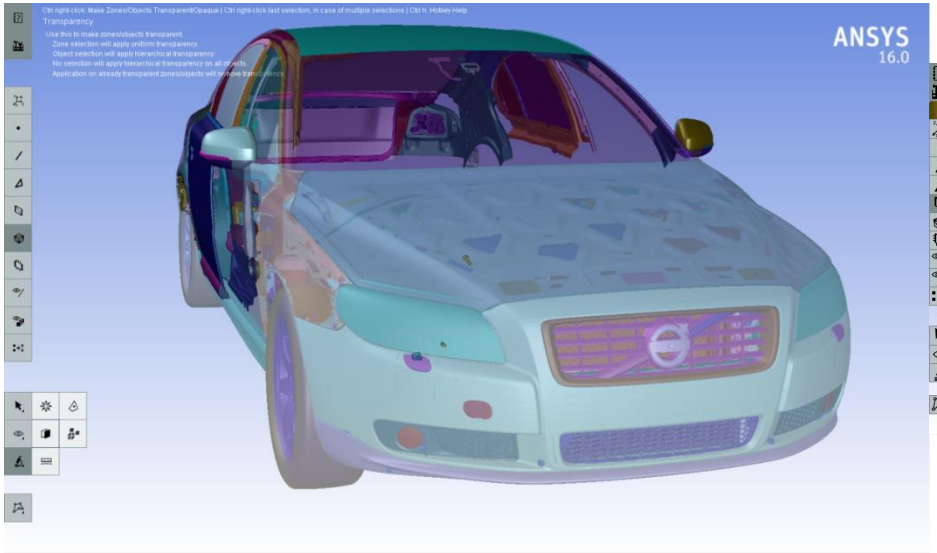
- 直接读入CAD，对应树形图管理
- 包含了案例设定的相关方面
- “右键点击”可以开启相应的最常用功能
- 后处理可以存储云图和矢量图的相应设置
- 提供清晰的设定流程指导，用户可以按照树形结构图导航完成所有设定

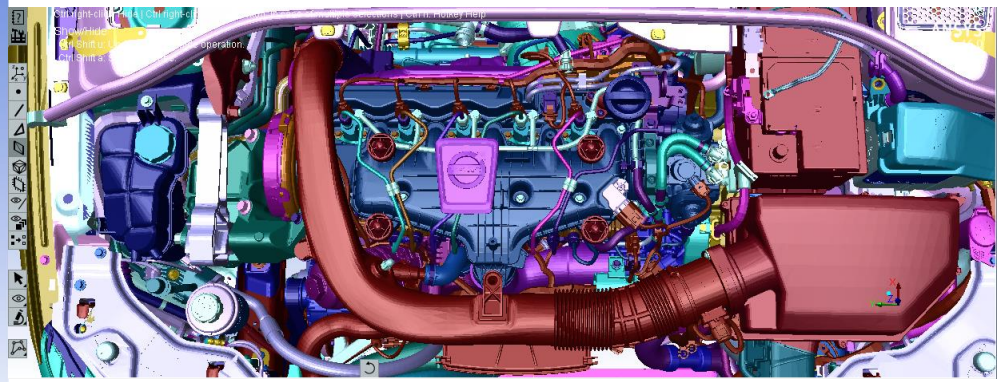


ANSYS Fluent前处理解决方案

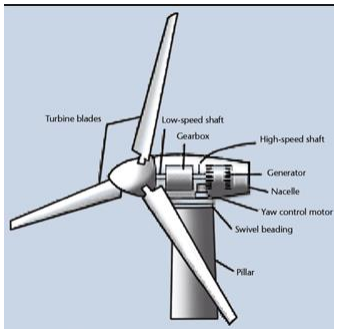
灵活的前处理

- 全新的前处理用户环境
- 复杂CAD到网格解决方案
  - Fluent Meshing方案
  - ANSA + Fluent Meshing 平台级方案
  - ANSYS SCDM + Fluent Meshing平台级方案



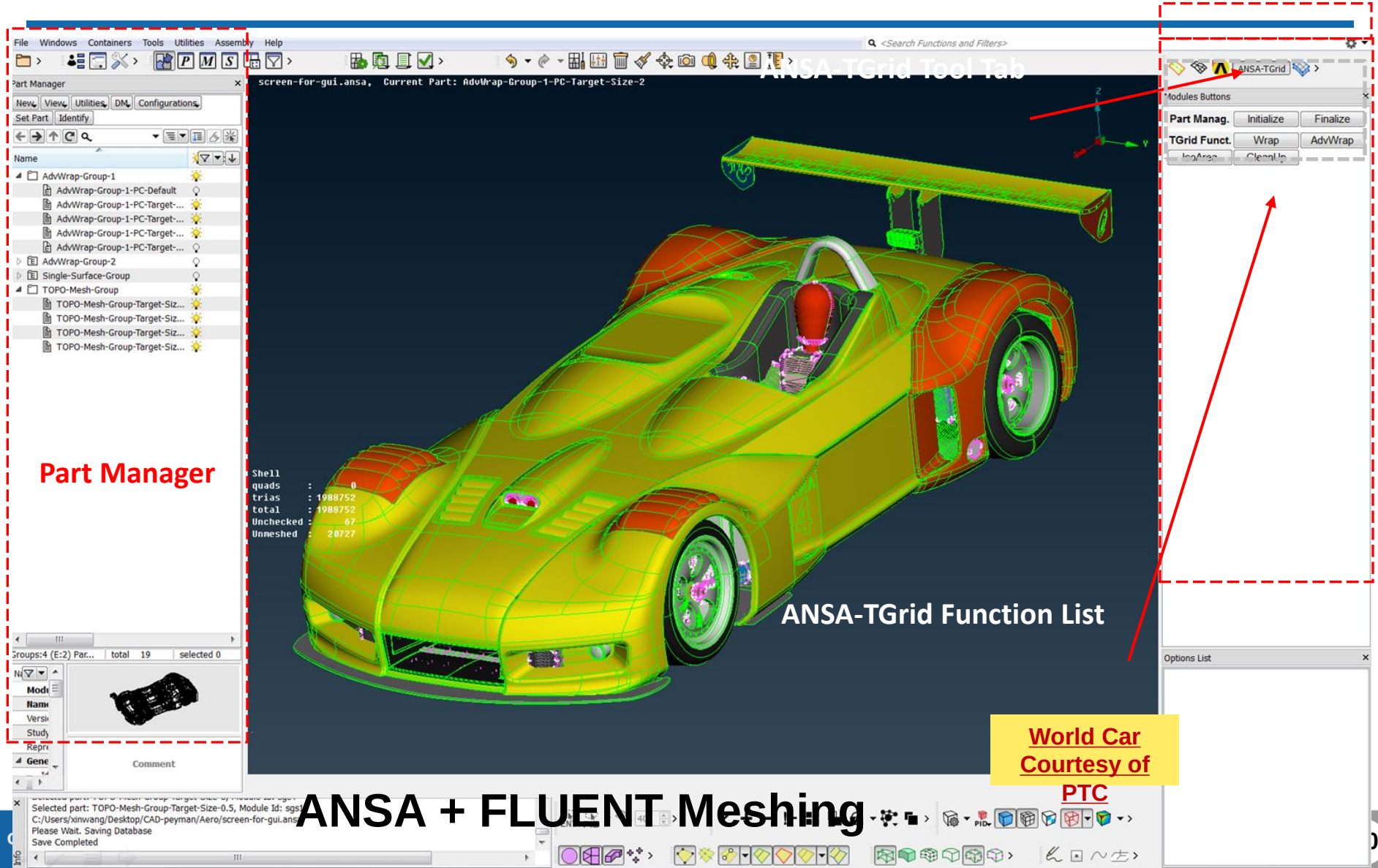


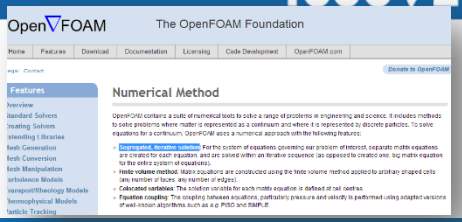
自动化网格生成效率



Application	Objective	Cell Count	Peak RAM (GB)	Effort
Loader Machine	FEAF	28 million tet (2 layers)		1 day
Small Truck	FEAF	46 million tet (2 layers)		1 day
Oven	Flow Rate	55 million tet (3 layers)		1.5 days
1/8th Scale Truck and Trailer	External Aero + FEA	40 million hexcore	24	1 day
Full-Scale Truck and Trailer	Aero	95 million hexcore (3 layers)	??	1 day
Wind-Turbine	Aero	46 million (10 layers)	21	0.5 day
Combustor (1 sector)	Aero	54 million (6 layers)	24	0.5 day
Nacelle	Aero	45 million (4 layers)	20	1 day
Oil Refinery	Aero	40 million (10 prism layers)	19.8	0.5 day
Passenger Car	Aero	80 million (2 layers) → 170 million (21 max layers – all adapted)		2-3 days
Volvo S80	Aero	95 million (2 layers)		2-3 days
Motorcycle	UTM	10 million (2 layers)		< 0.5 day

ANSYS Fluent前处理解决方案

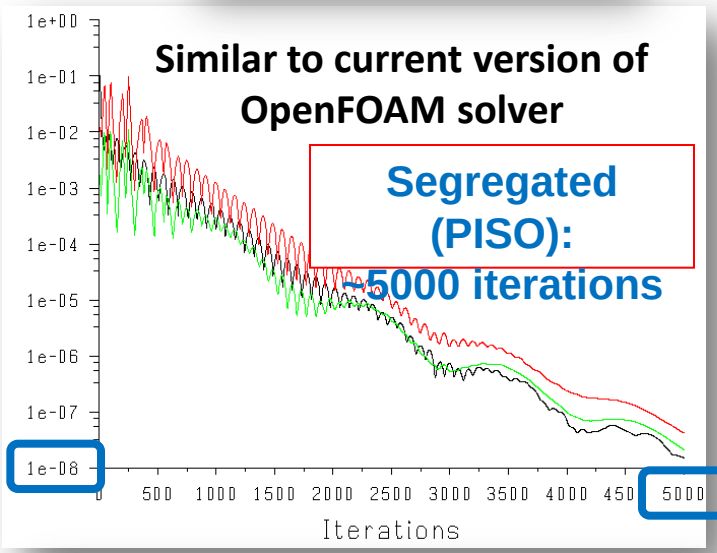




ANSYS FLUENT耦合求解器

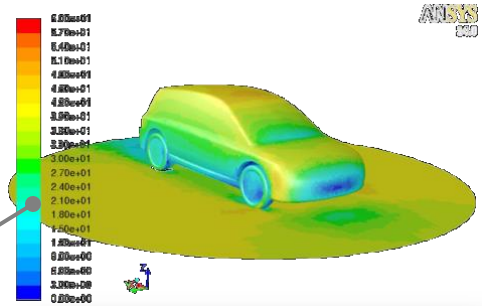
耦合方法求解器，提高气动求解收敛速度

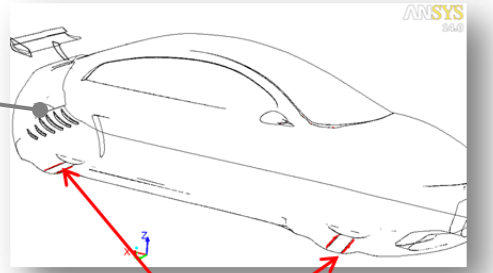
- 同类最佳的耦合求解器，特别适合进行气动分析



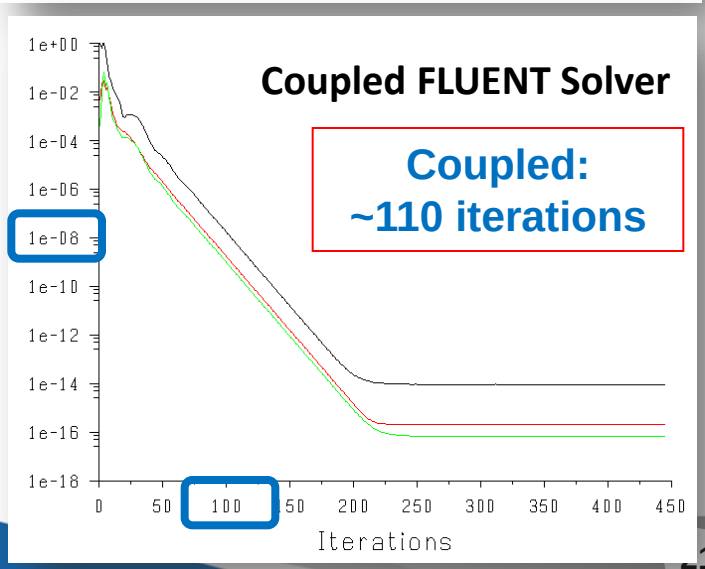
特殊选项：

- 更准确的流场初始化
- 差质量网格方法
- 伪瞬态算法
- 高阶松弛因子





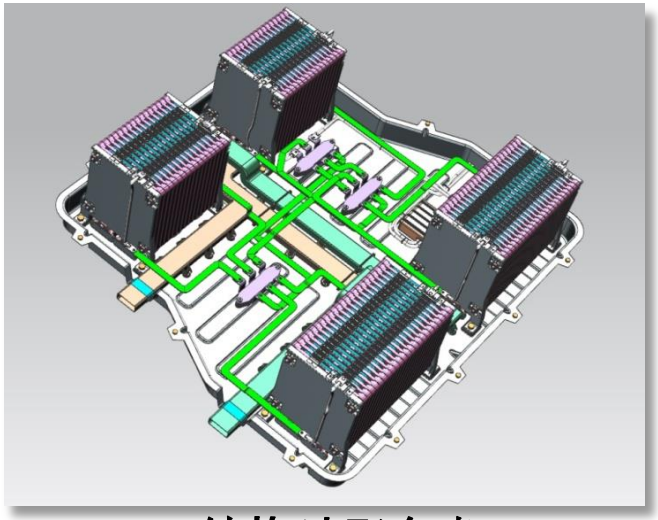
Marked poor elements at wheel ground sharp angle



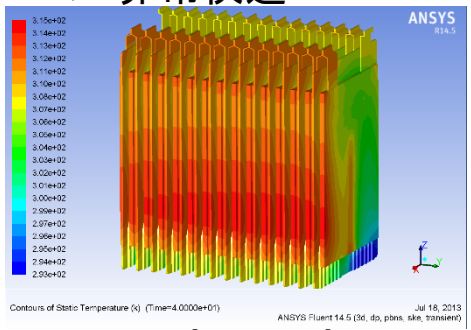
降阶模型

ROM模型提高瞬态热分析效率

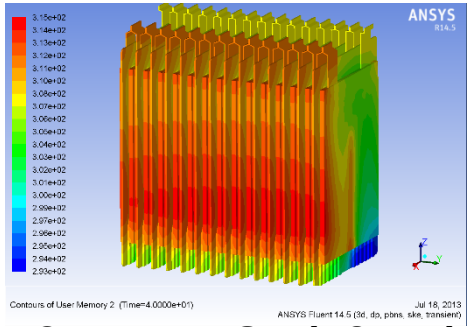
- 热网络方法
  - ➢ 基于大量的试验和计算
  - ➢ 精度有限
- LTI/SVD降阶方法
  - ➢ 基于CFD计算结果
  - ➢ 精度与CFD方法一致
  - ➢ 非常快速



结构外形复杂

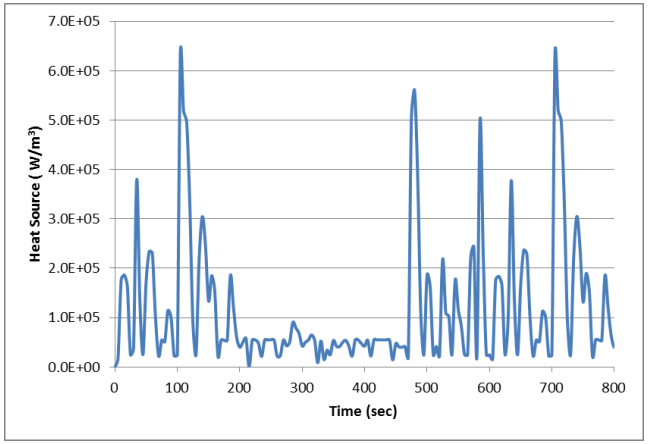


CFD (40 sec)



SVD+LTI ROM(40 sec)

CFD mesh size	12,209,486
CFD validation simulation time on six CPUs	20 hours
SVD ROM simulation time on the same single CPU	a few seconds



瞬态周期长

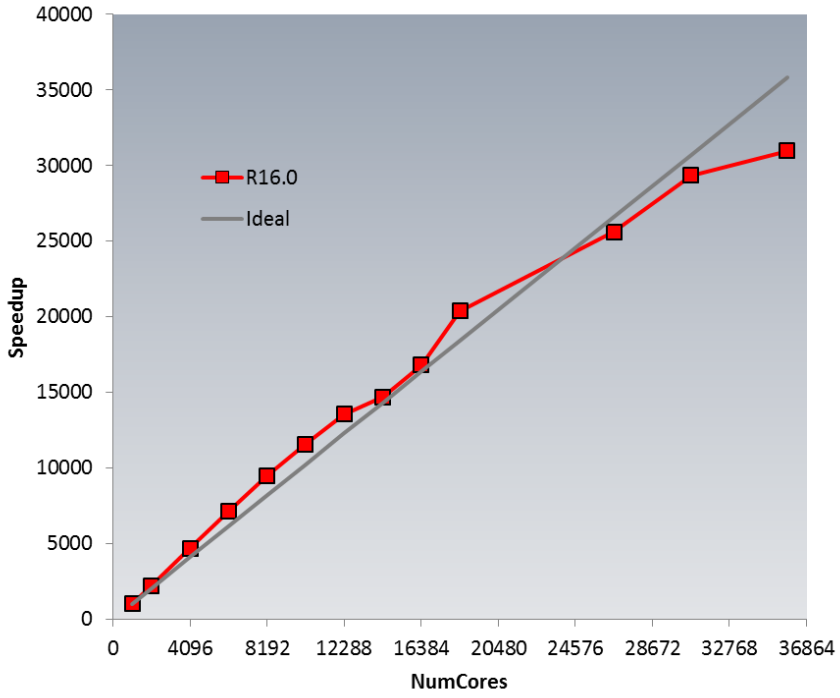
ANSYS FLUENT HPC 并行

HPC高性能并行计算加速比接近线性

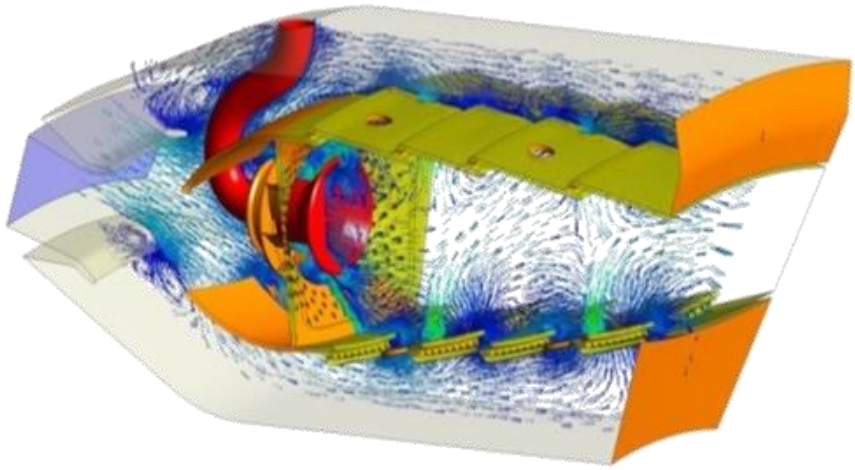
BLUE WATERS
SUSTAINED PETASCALE COMPUTING

R16 scales better than any other commercial CFD code

- 830 million cells
- 86% efficiency at 36k cores, 1024 cores as base



36,000 cores → only @ ANSYS!



ANSYS FLUENT GPU加速

VOITH HYDRO
POWER GENERATION

University of Stuttgart
Germany

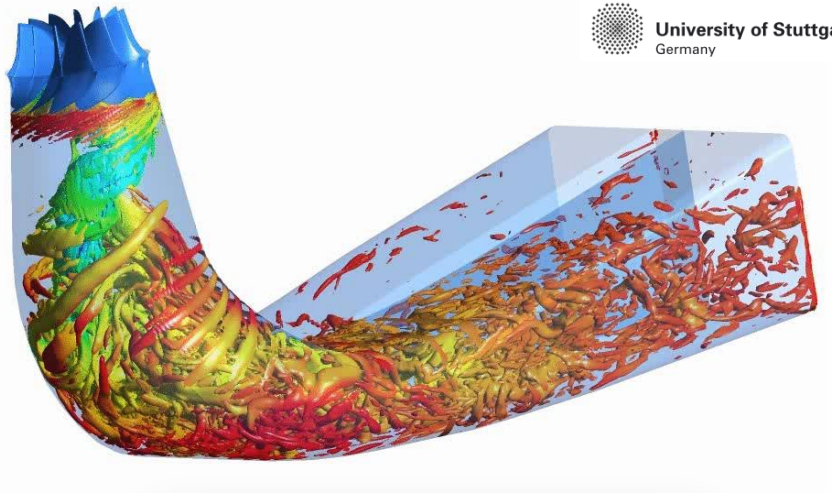
GPU加速功能提高数值求解效率

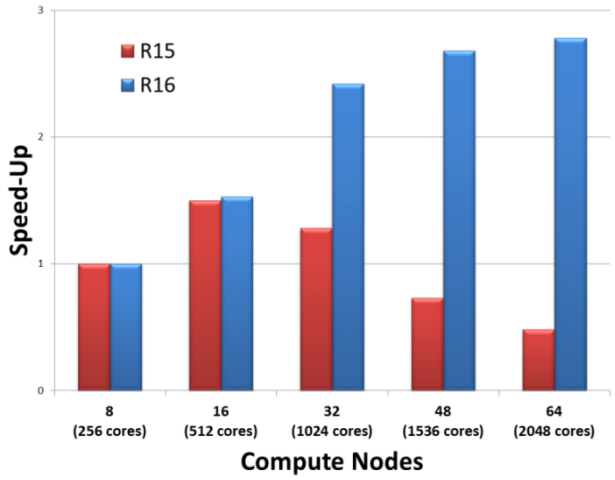
ANSYS Fluent

- GPU / CPU ≈ 6
- Parallel file I/O 10 × faster

ANSYS CFX

- 5 × faster

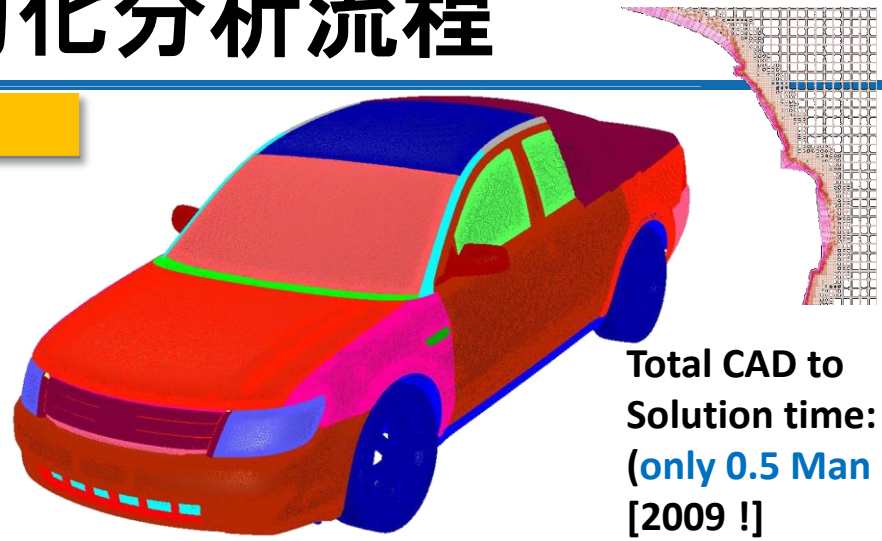




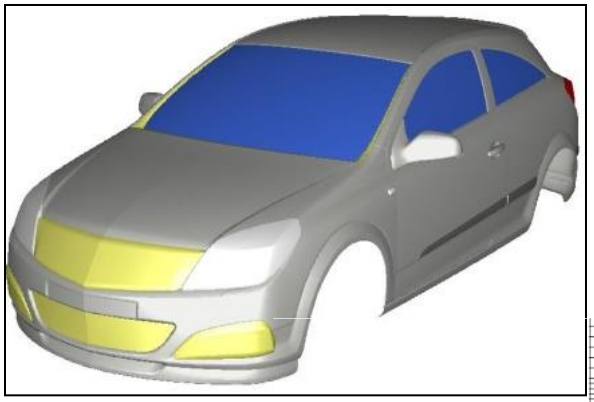
ANSYS FLUENT自动化分析流程

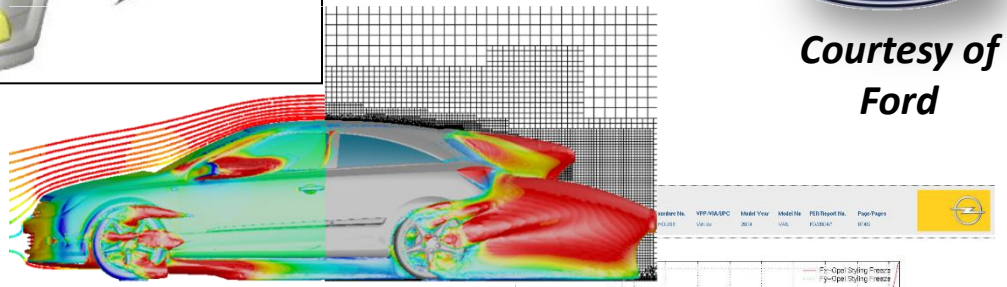
自动化的分析流程

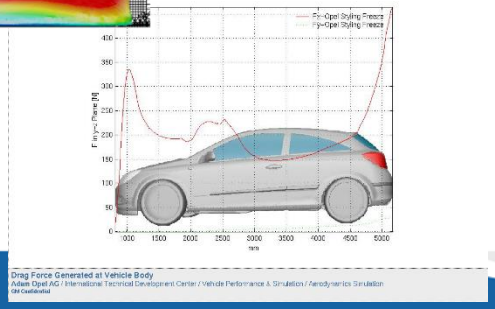
- 网格自动生成脚本

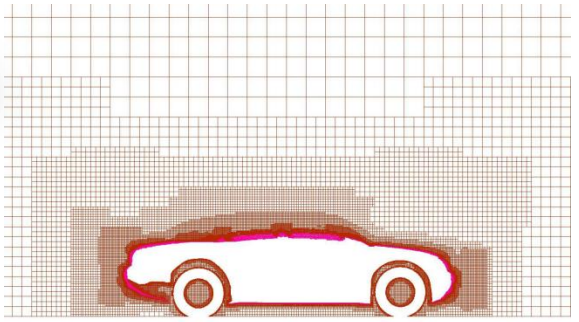


Total CAD to Solution time: 7h (only 0.5 Man hour) [2009 !]









Courtesy of Ford

Courtesy of GM/Opel

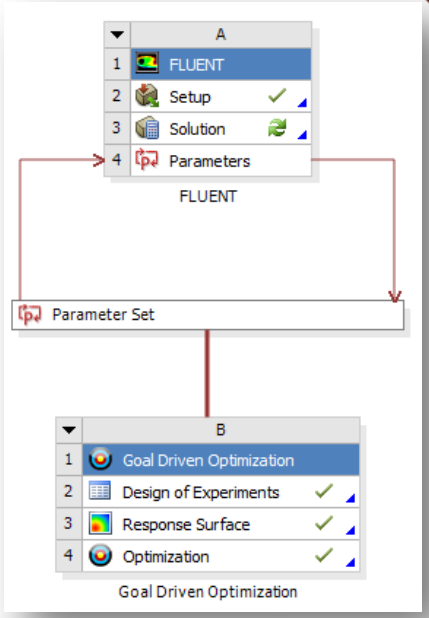
ANSYS FLUENT自动化分析流程

自动化的分析流程

- 网格自动生成脚本
- 支持参数化分析ANSYS Workbench

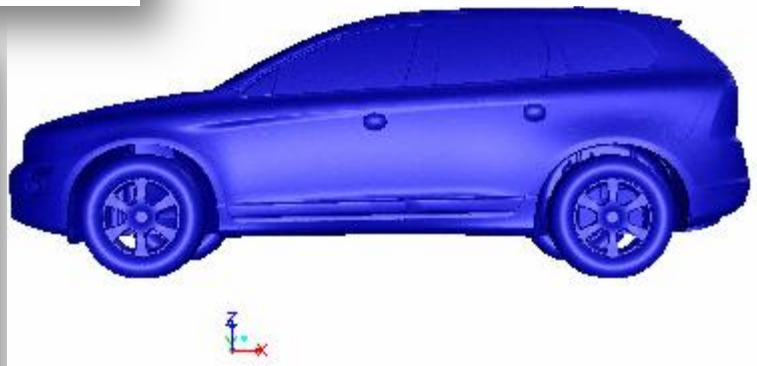
50:50:50 Project

- Volvo XC60 汽车模型
- 四个外形控制参数
- RBF Morph 定义外形参数
- 优化阻力系数



Courtesy of Volvo

50	50 design points in the design space	EXTENT
50	50 million cells used in CFD simulation of each design point	ACCURACY
50	50 hours total elapsed time to simulate all the design points	SPEED



ANSYS FLUENT自动化分析流程

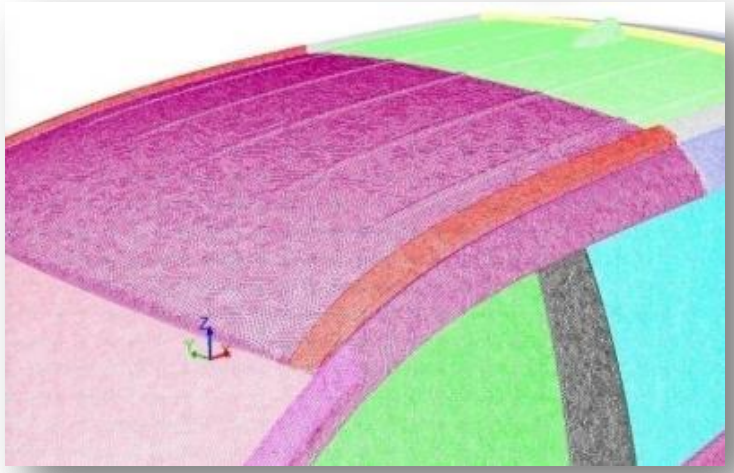
自动化的分析流程

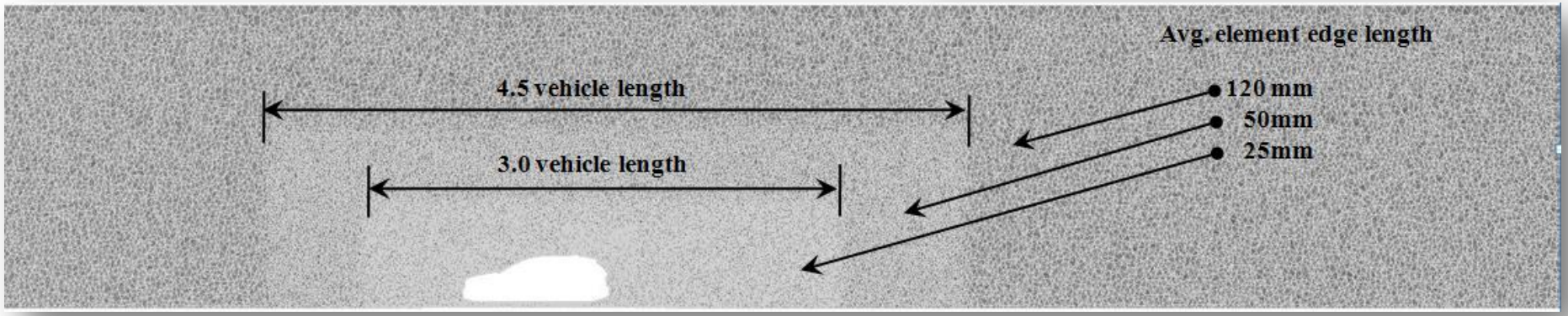
Courtesy of Volvo

- 网格自动生成脚本
- 支持参数化分析ANSYS Workbench

50:50:50 Project

- 网格数量 50.2 M Cells
- 附面层数量 : 10
- 附面层网格数量: 24.4 M Cells
- Skewness < 0.9





ANSYS FLUENT自动化分析流程

(rbf-morph)™

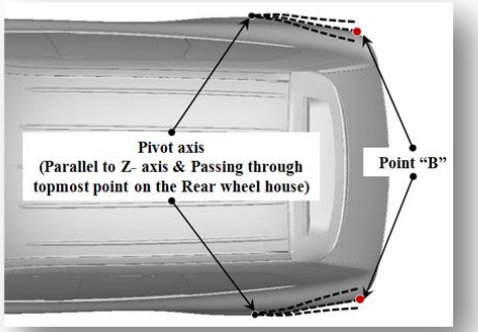
自动化的分析流程

Courtesy of Volvo

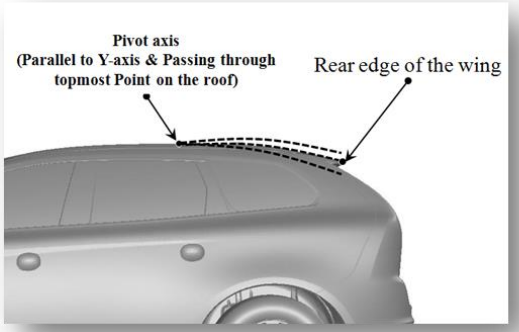
- 网格自动生成脚本
- 支持参数化分析ANSYS Workbench

50:50:50 Project

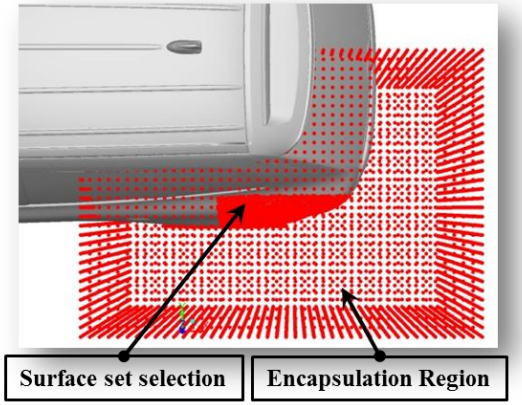
- 指定4个控制参数
- RBF Morph完成网格变形

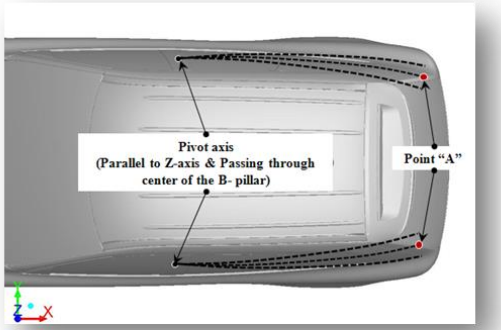


Boat Tail Angle (P2)

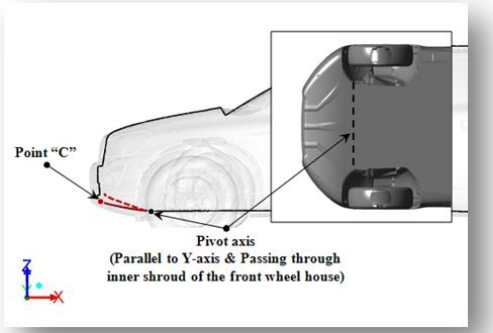


Roof Drop Angle (P3)





Green House (P4)



Front Spoiler Angle (P5)

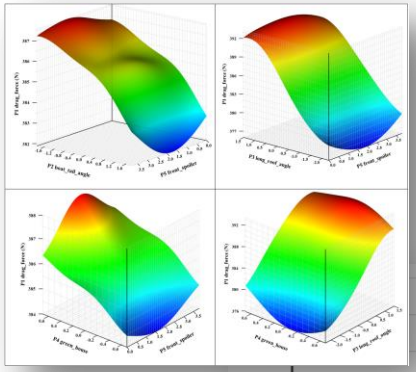
ANSYS FLUENT自动化分析流程

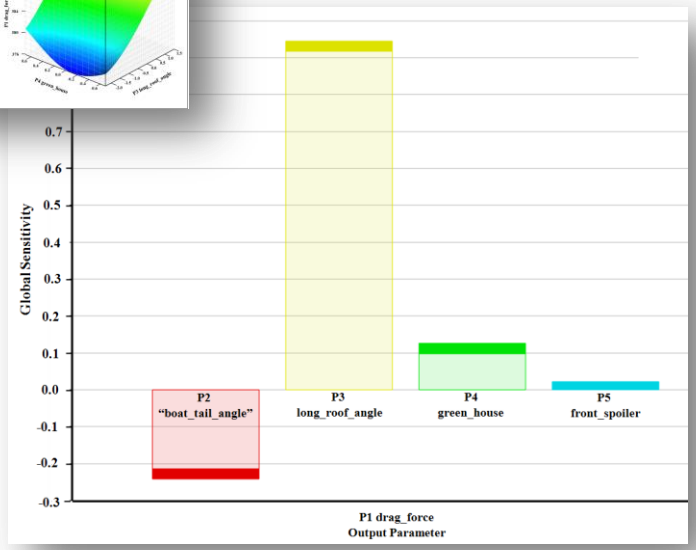
设置/求解过程更加自动化

- 网格自动生成脚本
- 支持参数化分析ANSYS Workbench

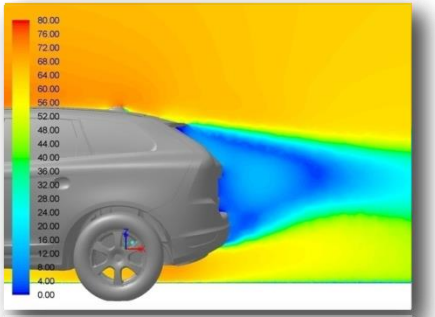
50:50:50 Project

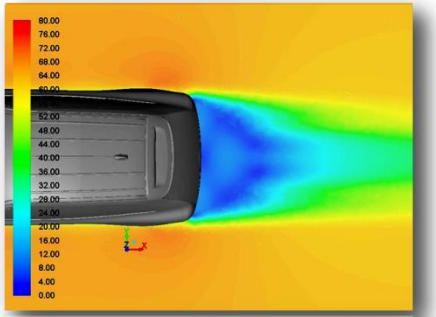
- Design Exploration 驱动算例求解
- 表面响应
- 优化

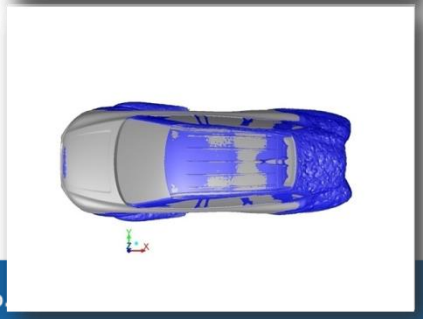


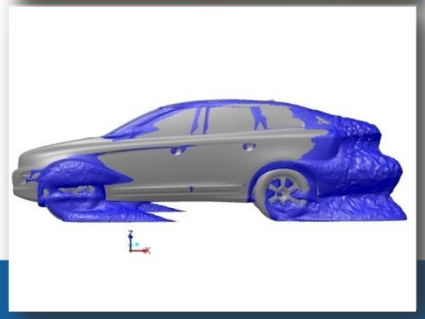


Courtesy of Volvo









-4% Drag

Do not remesh, morph instead !

Highly automated "one-click" proc

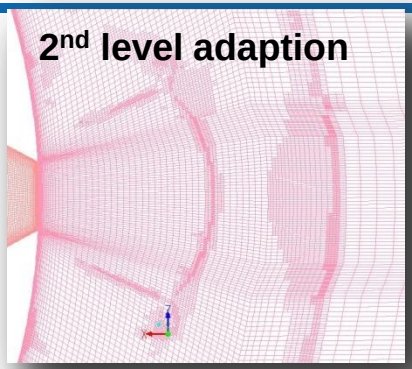
ANSYS FLUENT自动化分析流程

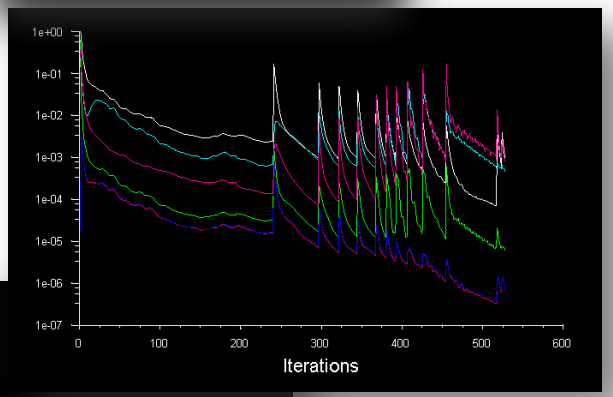
设置/求解过程更加自动化

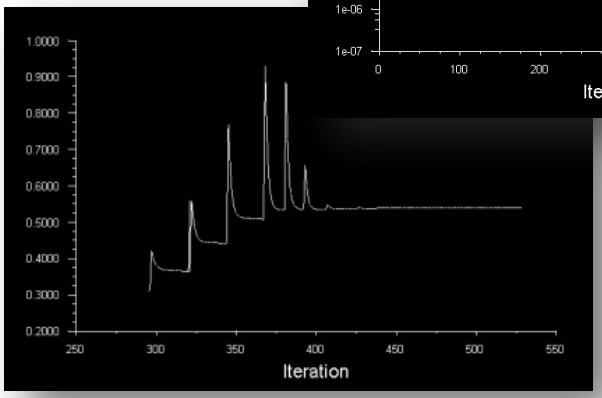
- 网格自动生成脚本
- 支持参数化分析ANSYS Workbench
- 支持自动化网格加密

Tet, Hexa, Prism, Cartesian cells

- 在收敛过程中动态加密
- 各向异性网格自适应
- 基于仿真结果的网格自适应加密：
  - 成功应用于汽车外流及整车热管理分析
  - 附面层数量从2增加到12
  - 网格数量从50增加到72M







Example of automatic 6 levels of grid adaption:
Not change in wall shear stress after 6th adaption

Automotive Simulation World Congress 2016, June 7-8, Munich

Co-located with ANSYS Electronics Simulation Expo

ANSYS

欢迎扫一扫添加IDAJ-China公众账号，
及时获得CAE&CFD最新技术资讯、培训信息！

IDAJ-China
微信公众账号

IDAJ艾迪捷
微博账号

2015年4月3日消息：业界知名CAE&CFD数值仿真软件经销商艾迪捷信息科技(上海)有限公司(IDAJ中国)与美国ANSYS中国公司(ANSYS-China)正式签订ANSYS代理协议。IDAJ中国获得正式授权在中国地区分销ANSYS产品，提供技术支持及培训服务。IDAJ官网：www.idaj.cn

ANSYS产品咨询：

Email: ansys@idaj.cn

Tel: 010-65881497; 021-50588290