Session H3.5

ANSA FOR CFD - OVERVIEW AND OUTLOOK

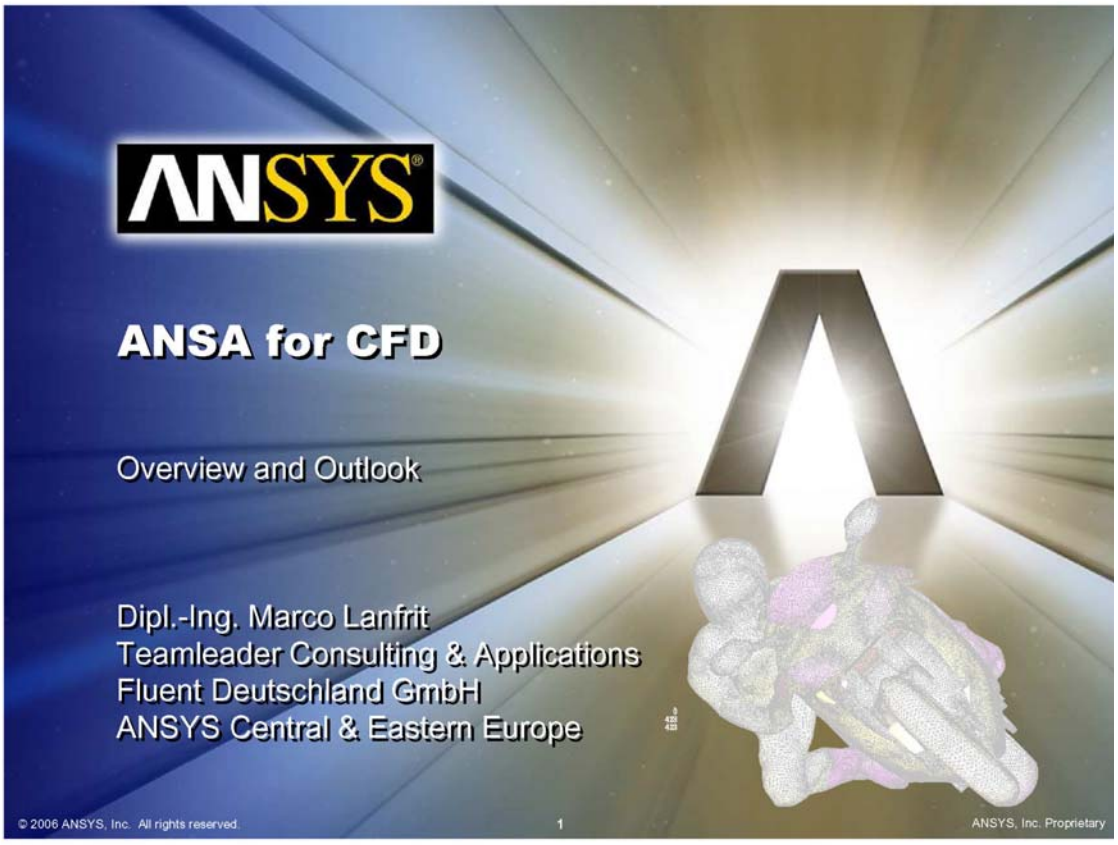
Marco Lanfrit*, Thomas Lehnhäuser
ANSYS Central & Eastern Europe - Fluent Deutschland GmbH, Germany

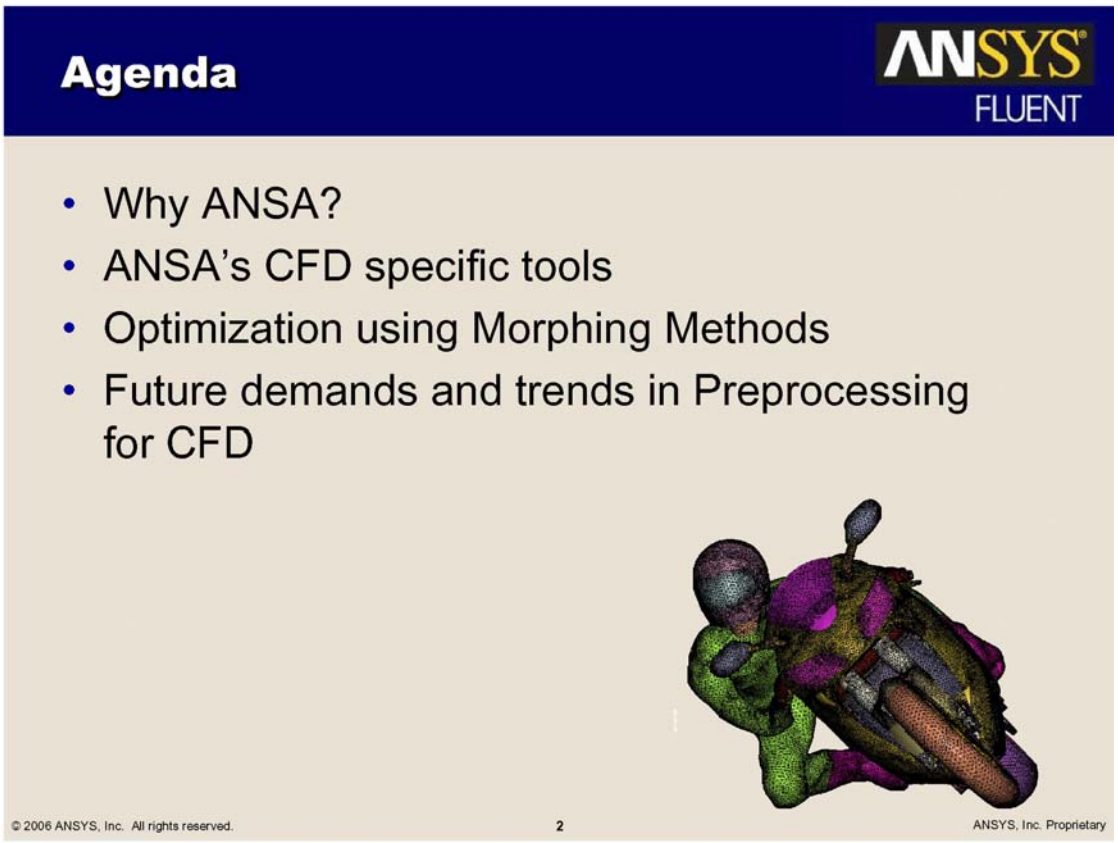
KEYWORDS - CFD, Preprocessing, FLUENT, Morphing, Meshing

ABSTRACT - Preprocessing is the primary and underlying Step in every CFD process. It is mandatory for a good and reliable result of the subsequent simulation. As computer resources grow and get cheaper, Engineers of all industries want to simulate in more detail and thus deal with complex geometries and a variety of CAD and mesh formats.

Based on an existing mesh, mesh deformation in combination with design optimisation methods yield significant gains in term of CFD productivity and turnaround time.

ANSA offers advanced tools to deal with complex geometries and mesh deformation. It is known as one of the most reliable Preprocessing tools in the world.
The application of ANSA in the CFD process is shown for a specific industrial case.
At the end of the presentation, the future demands of CFD users with respect to a Preprocessing tool will be highlighted.

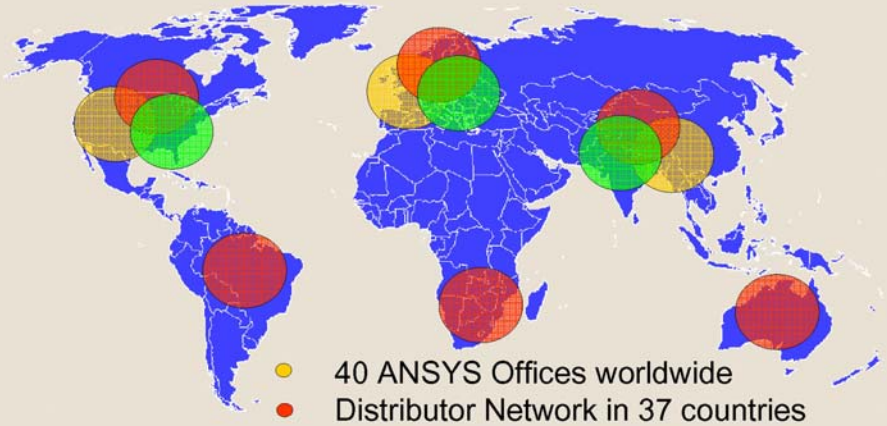

ANSYS
ANSA for CFD
Overview and Outlook
Dipl.-Ing. Marco Lanfrit
Teamleader Consulting & Applications
Fluent Deutschland GmbH
ANSYS Central & Eastern Europe
© 2006 ANSYS, Inc. All rights reserved.
1
ANSYS, Inc. Proprietary


Agenda
ANSYS
FLUENT
• Why ANSA?
• ANSA's CFD specific tools
• Optimization using Morphing Methods
• Future demands and trends in Preprocessing for CFD
© 2006 ANSYS, Inc. All rights reserved.
2
ANSYS, Inc. Proprietary

ANSYS worldwide

ANSYS FLUENT

Headquater: Canonsburg (Pittsburgh), Penn.



- 40 ANSYS Offices worldwide
- Distributor Network in 37 countries
- 17 Software Development Centres

> 1.400 employees
> 280 Mio. $ revenue in 2005

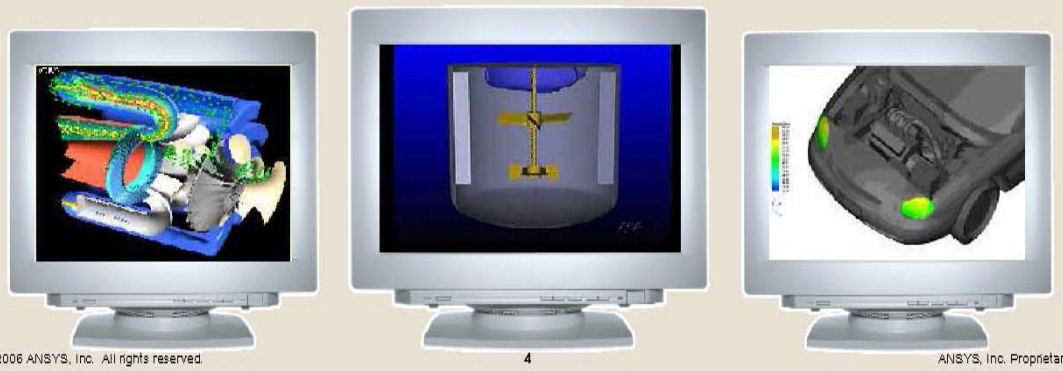
3

ANSYS - Vision

ANSYS FLUENT

- ***Simulation Driven Product Development***

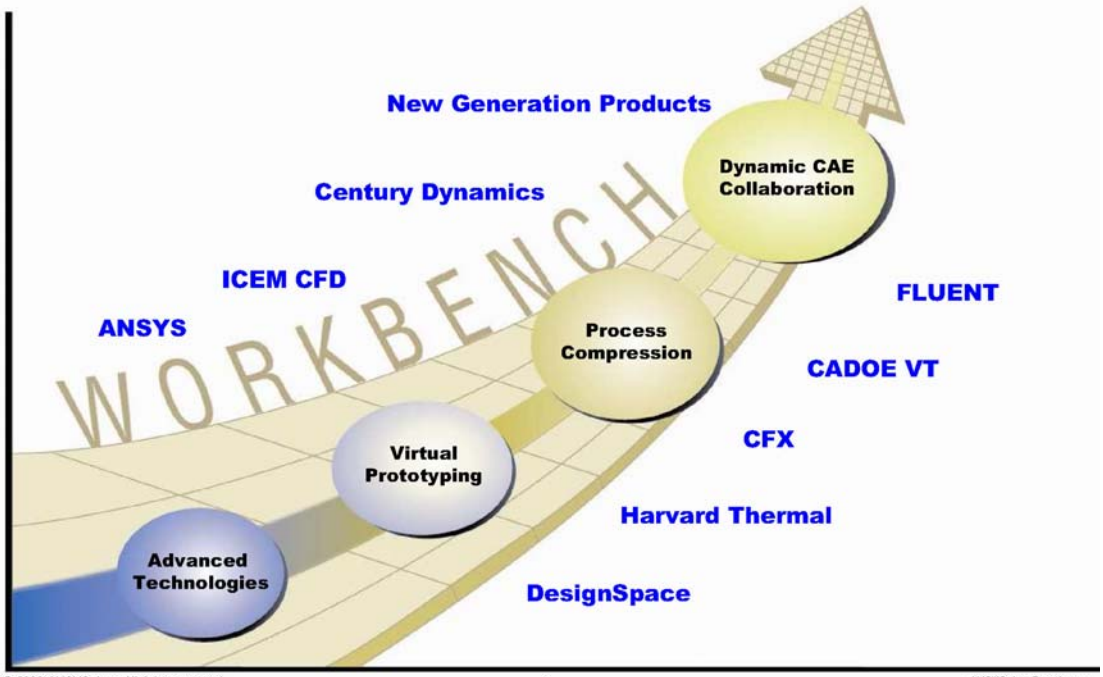
BETTER – *Innovative & higher-quality products*
FASTER – *Dramatic time-to-market improvement*
CHEAPER – *Minimize development, warranty & liability costs*



4

Broad & Integrated Solutions

ANSYS FLUENT

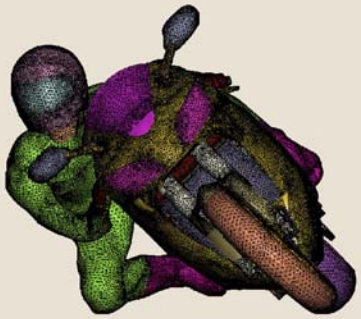


5

Agenda

ANSYS FLUENT

- Why ANSA?
- ANSA's CFD specific tools
- Optimization using Morphing Methods
- Future demands and trends in Preprocessing for CFD



6

Why ANSA ?

ANSYS FLUENT

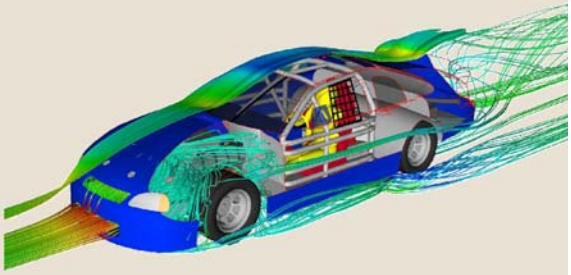
There are two main reasons, why ANSYS engineers are coming in contact with ANSA:

1. A lot of our customers are using ANSA, either in combination with our or other CFD Codes, or for structural analysis
2. ANSA is a very flexible and effective tool for preprocessing, especially for complex geometries

Beta Cae and Fluent a winning team since many years

ANSYS FLUENT

- First combined application of ANSA / TGrid in 1995 at Renault, France (via NASTRAN)
- 1996: FLUENT Interface (surface mesh)
- Highly flexible and efficient approach to create high quality CFD-meshes for complex configurations
- Has become a very successful and efficient combination of tools amongst automotive users

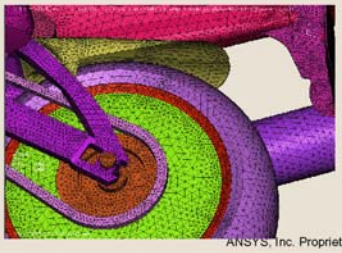


Collaboration

ANSYS FLUENT

Beta Cae and ANSYS/Fluent have a loosely collaboration:

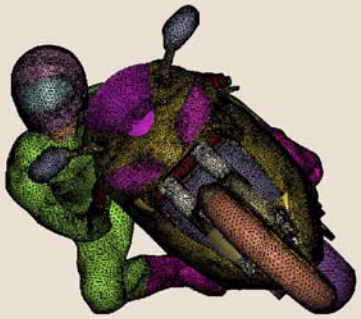
- Software Exchange
- Fluent delivers input for CFD/Fluent specific functionalities
- Fluent provides access to code know-how (e.g. skewness definiton, output formats, …)
- Beta CAE offers tools, which are custom made for FLUENT users (e.g. CFD Mesh)

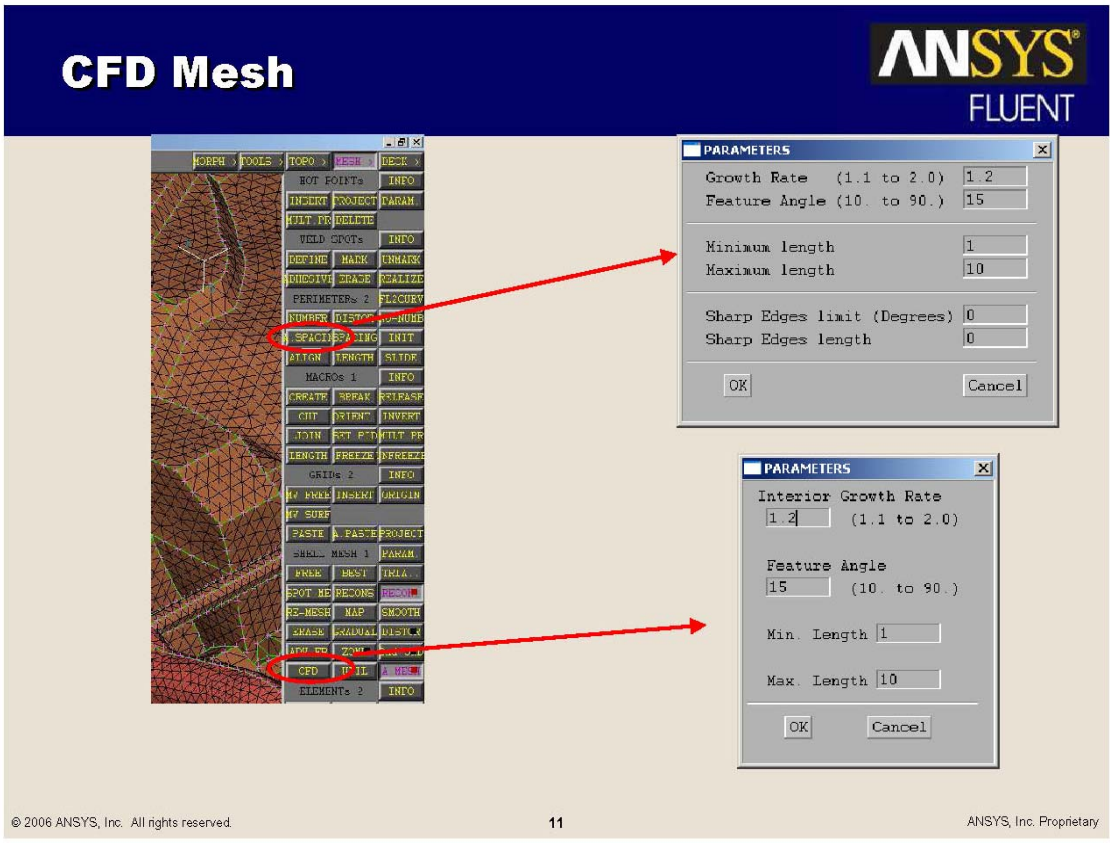


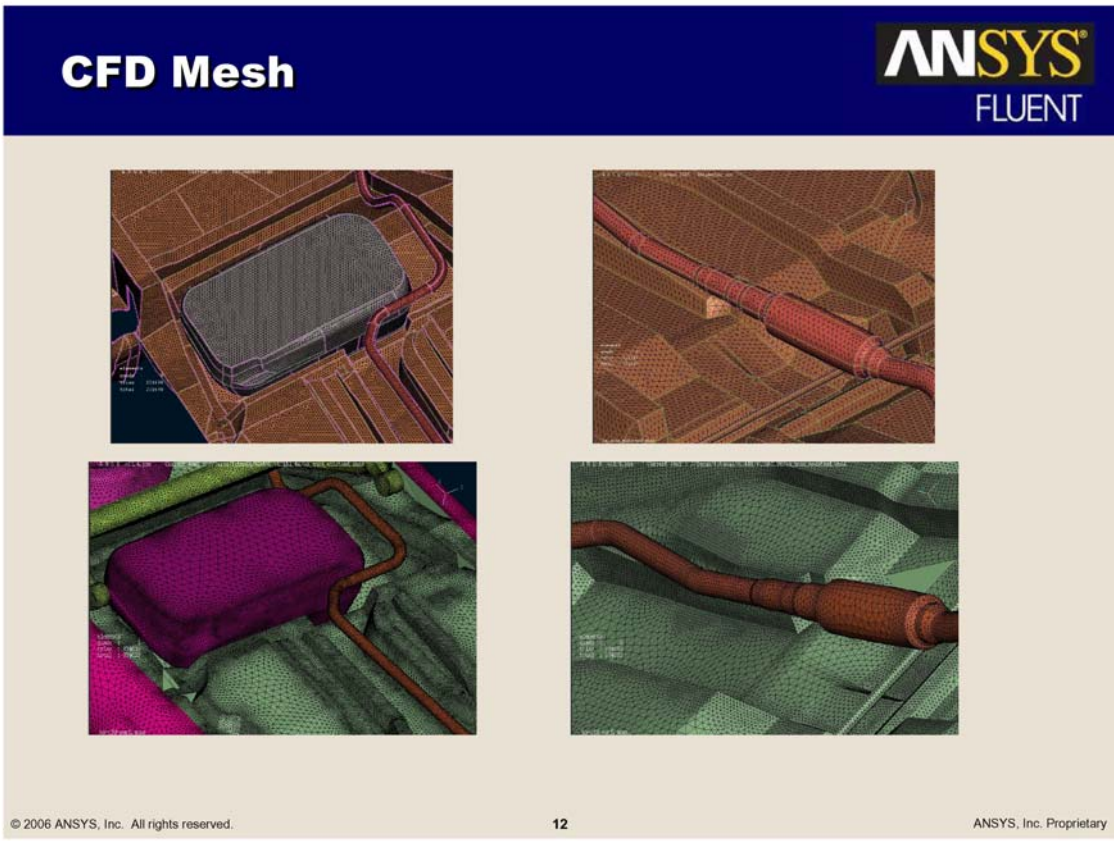
Agenda

ANSYS FLUENT

- Why ANSA?
- ANSA's CFD specific tools
- Optimization using Morphing Methods
- Future demands and trends in Preprocessing for CFD




CFD Mesh
ANSYS
FLUENT
PARAMETERS
Growth Rate (1.1 to 2.0) 1.2
Feature Angle (10. to 90.) 15
Minimum length 1
Maximum length 10
Sharp Edges limit (Degrees) 0
Sharp Edges length 0
OK
Cancel
PARAMETERS
Interior Growth Rate
1.2 (1.1 to 2.0)
Feature Angle
15 (10. to 90.)
Min. Length 1
Max. Length 10
OK
Cancel
© 2006 ANSYS, Inc. All rights reserved.
11
ANSYS, Inc. Proprietary


CFD Mesh
ANSYS
FLUENT
© 2006 ANSYS, Inc. All rights reserved.
12
ANSYS, Inc. Proprietary

Ford Ka – Meshing Benchmark

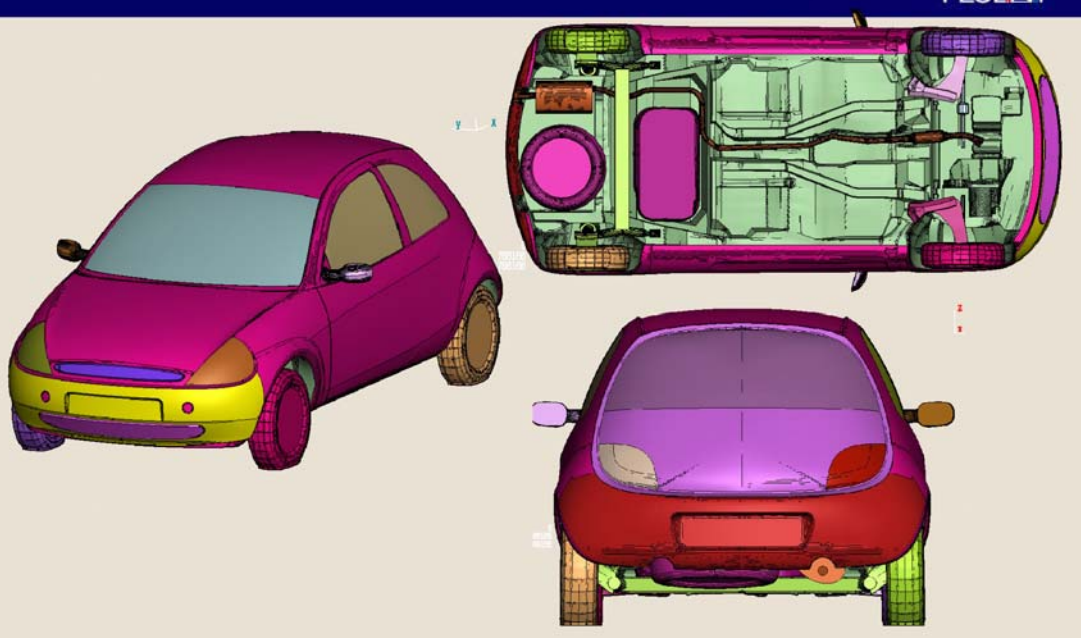
ANSYS FLUENT

- Geometry was used for EADE Benchmark and following studies
- Detailed Underbody, Side Mirrors, Wheelhouses
- „Conventional Way“ took around 2 – 3 weeks for Surface Meshing, because a high resolution mesh with highly non-uniform element sizes was aimed
- Mesh shown in the following took around 1 hour for Meshing in ANSA v.12.0
- Quality is similar

13

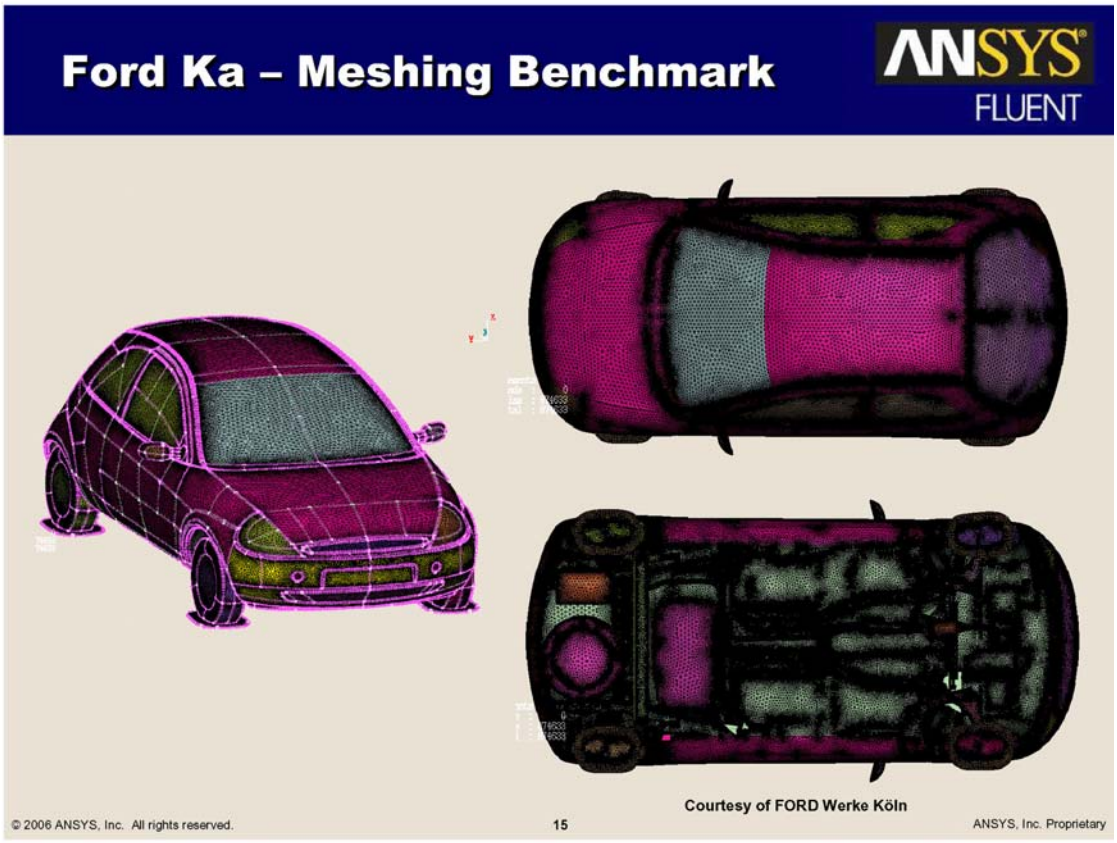
Ford Ka – Meshing Benchmark

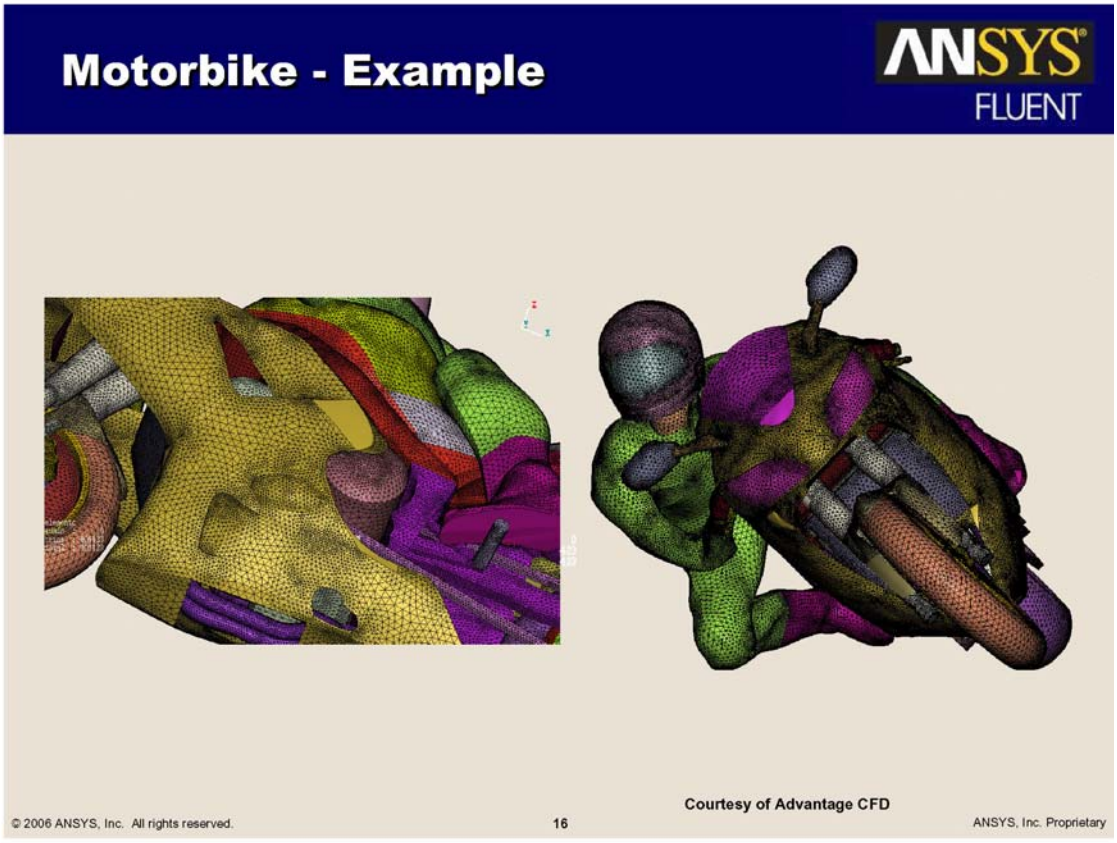
ANSYS FLUENT



Courtesy of FORD Werke Köln

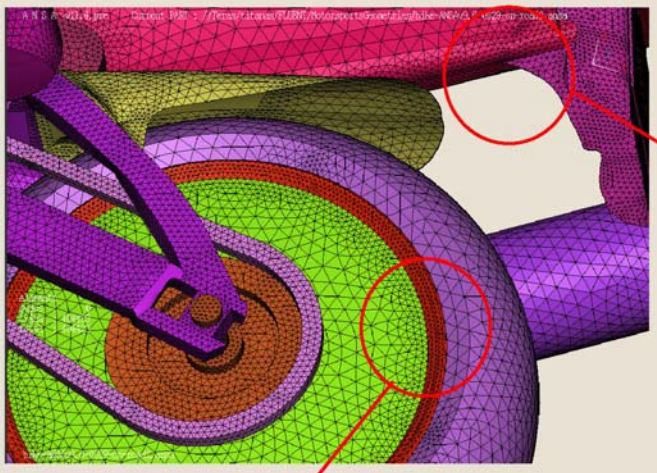
14


Ford Ka – Meshing Benchmark
ANSYS
FLUENT
Courtesy of FORD Werke Köln
© 2006 ANSYS, Inc. All rights reserved.
15
ANSYS, Inc. Proprietary


Motorbike - Example
ANSYS
FLUENT
Courtesy of Advantage CFD
© 2006 ANSYS, Inc. All rights reserved.
16
ANSYS, Inc. Proprietary

Motorbike - Example

ANSYS FLUENT

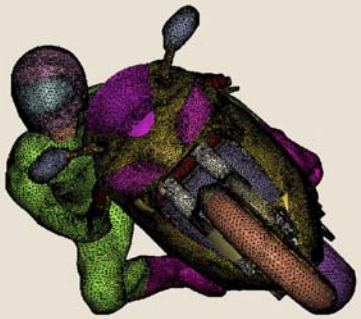


Courtesy of Advantage CFD

Agenda

ANSYS FLUENT

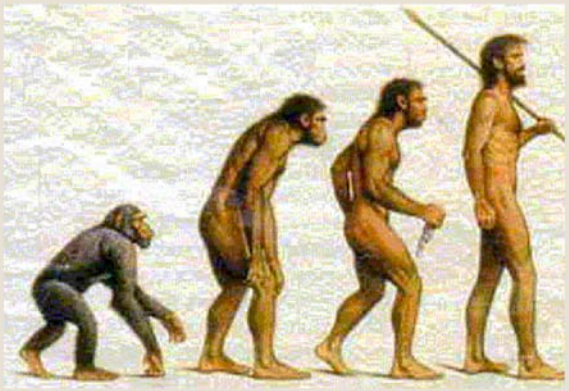
- Why ANSA?
- ANSA's CFD specific tools
- Optimization using Morphing Methods
- Future demands and trends in Preprocessing for CFD



Optimization

ANSYS FLUENT

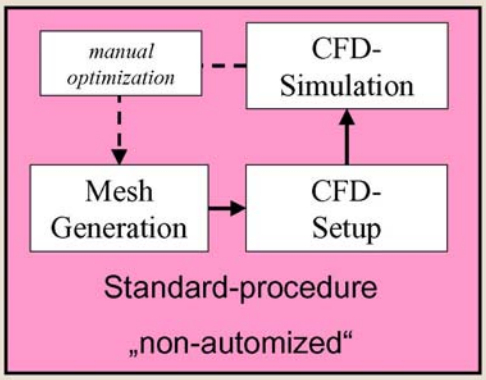
Improvement of specific attributes/qualities through parameter adaption and variation



Global Optimum?

Optimization - process

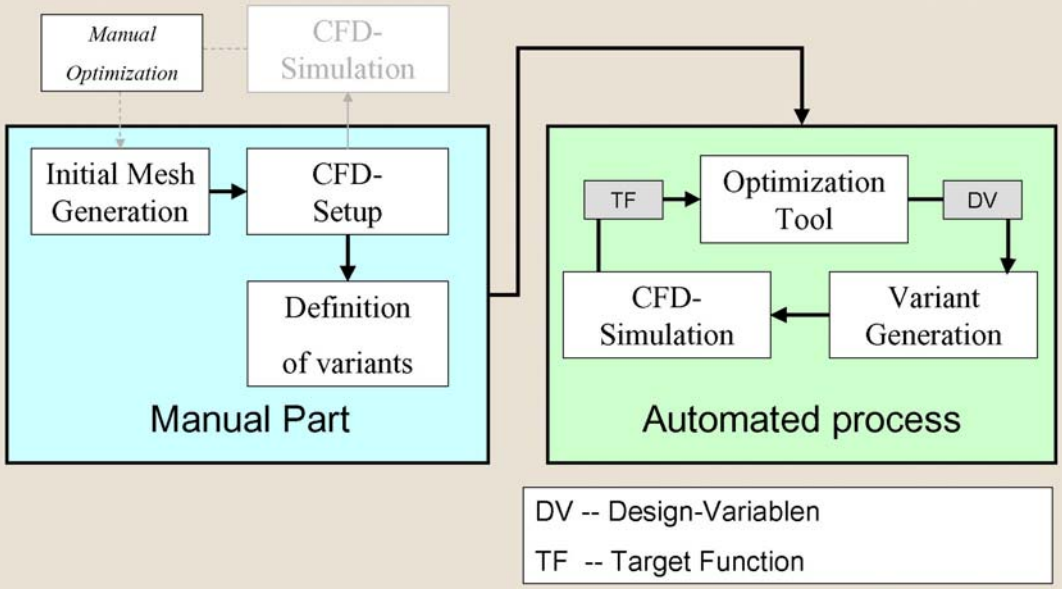
ANSYS FLUENT



- „easy” automizable
  - Mesh generation
  - CFD-Setup
  - CFD-Simulation
- „difficult” automizable
  - manual optimization

Automation

ANSYS FLUENT



21

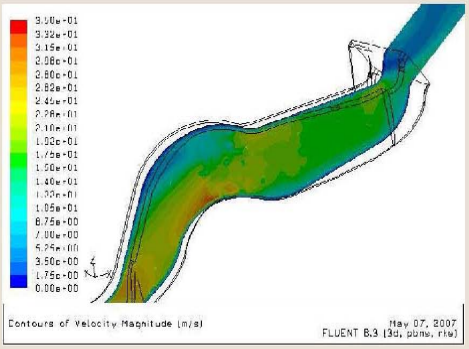
Example – Climate Control Duct

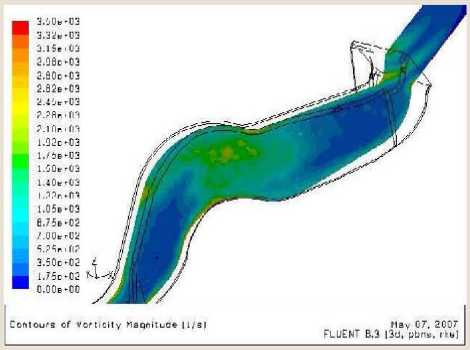
ANSYS FLUENT

22

Example – Climate Control Duct

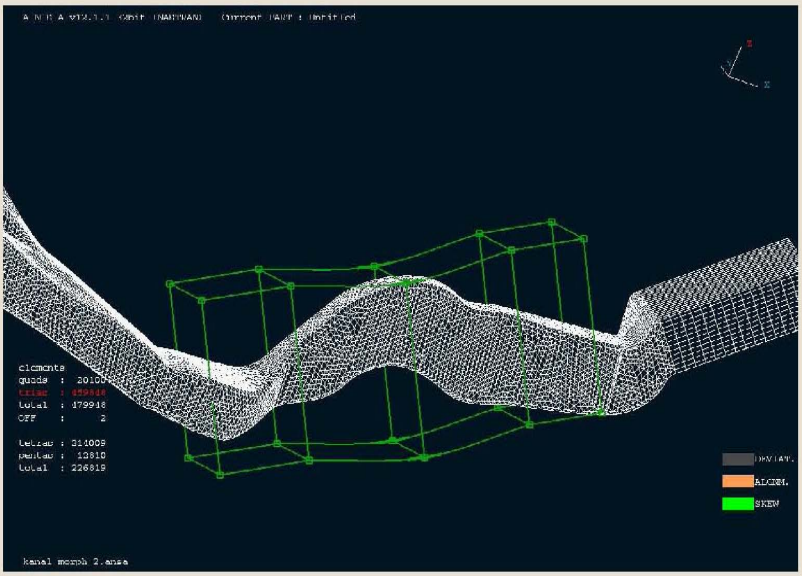
ANSYS FLUENT





Example – Climate Control Duct

ANSYS FLUENT



Example – Climate Control Duct

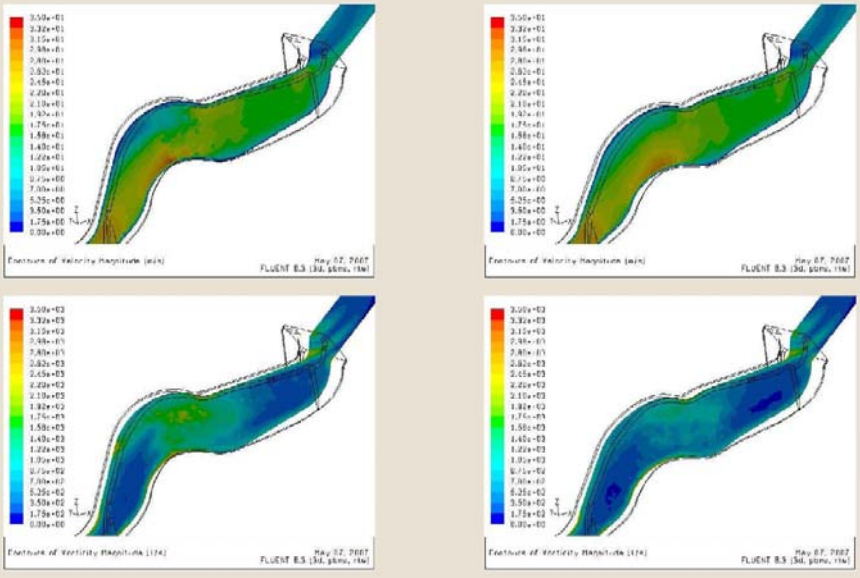
ANSYS FLUENT

Basis Optim.

25

Example – Climate Control Duct

ANSYS FLUENT



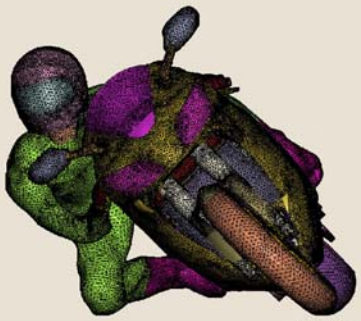
Basis Δp=100% Δp=95% Optim.

26

Agenda

ANSYS FLUENT

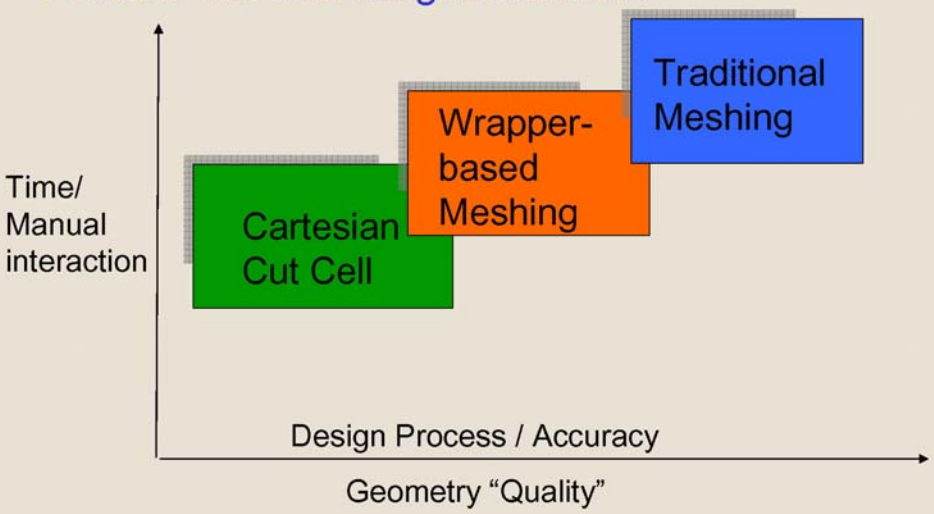
- Why ANSA?
- ANSA's CFD specific tools
- Optimization using Morphing Methods
- Future demands and trends in Preprocessing for CFD



Trends in CFD

ANSYS FLUENT

- Provide the right mesh for the right problem
- Reduce the meshing bottleneck



Trends in CFD Preprocessing

ANSYS FLUENT

- Cavity Remeshing (TGrid Prototype)
- Cut Cell Meshing (TGrid Prototype)
- Surface Wrapping (TGrid Release)
- Immersed Boundaries (Roll out during EACC 2007 in Frankfurt)

29

Cavity Remesh

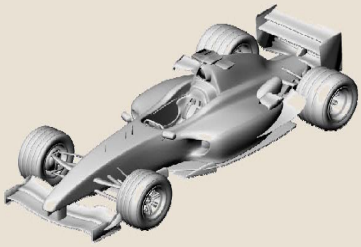
ANSYS FLUENT

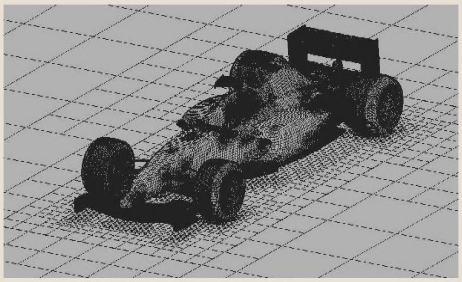
- Exchange (surface) parts in an existing volume mesh
- no need to re-mesh the complete domain
- Saves huge amount of time
  - 100 Mio cell mesh (Formula 1) 1h vs. 18h for volume mesh creation

30

Cut Cell Approach

ANSYS FLUENT





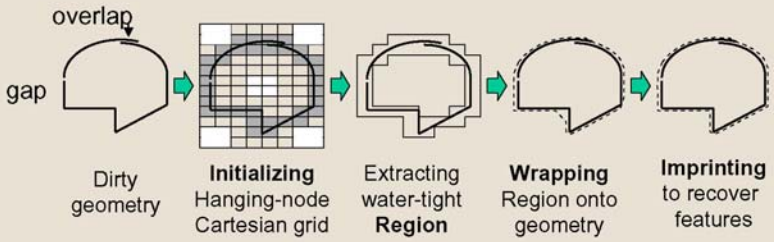
- Push button approach
- Very low turnaround time
- Sufficient accuracy
- Early design evaluation

What does wrapping mean ?

ANSYS FLUENT

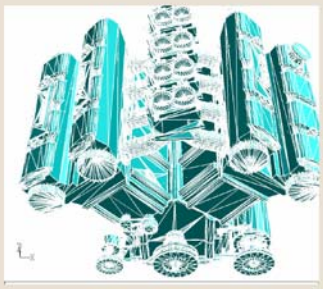
- Main Entry: **1wrap**
  Pronunciation: 'rap
  Function: *verb*
  Inflected Form(s): **wrapped**; **wrap·ping**
  Etymology: Middle English *wrappen*
  transitive senses
  1 a : to cover especially by winding or folding **b :** to envelop and secure for transportation or storage **: BUNDLE c : ENFOLD, EMBRACE d :** to coil, fold, draw, or twine (as string or cloth) around something

- In TGrid 4



Surface Wrapping

ANSYS FLUENT

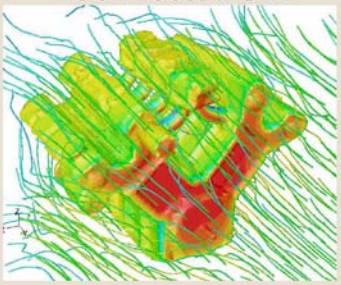


Unconnected STL

Connected STL

CFD triangular mesh

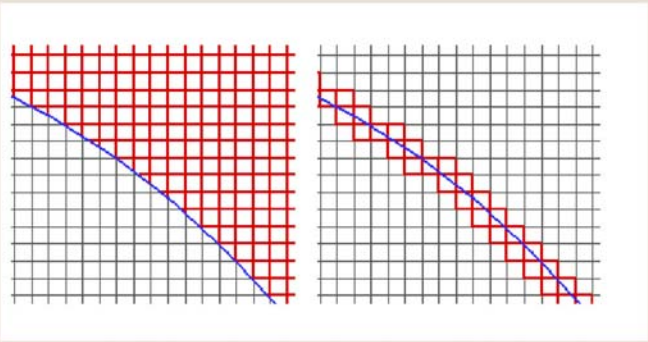
CFD solution



Immersed Boundary Technique

ANSYS FLUENT

Grids DO NOT conform with the boundaries of the CFD domain



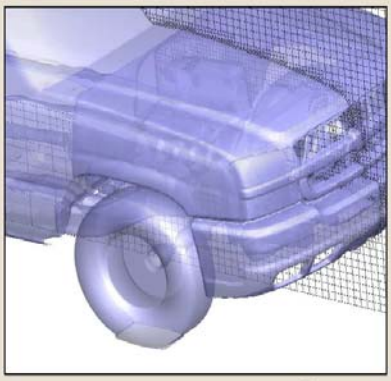
Cell-Cut Approach
Computational stencils are modified to use information on the boundaries.
(Direct BC)

Immersed Boundary
The governing equations are modified in the cells cut by the interface by adding a source term (Indirect BC or Forcing)

Start from STL Geometry

ANSYS FLUENT

- Avoids time-consuming and error-prone CAD to CFD geometry conversion/clean-up issues
- Locally refined, high quality, Cartesian mesh is generated automatically
- Sufficient accuracy for preliminary studies

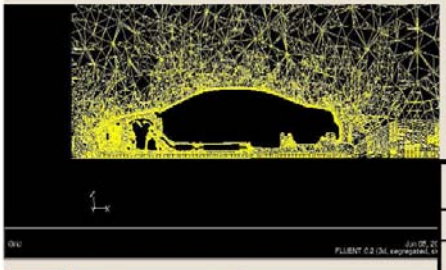


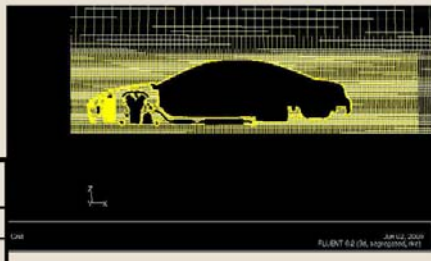
Courtesy of
General Motors Corporation

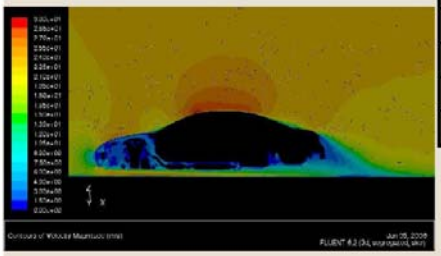
35

IB Validation for a Car

ANSYS FLUENT

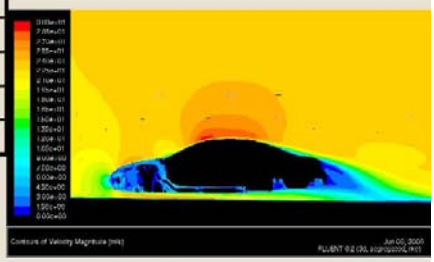






Grand-prix	Drag coefficient
Experiments	100
BF1, 7M cells	95.6
BF2, 10M cells	94.4
IB1, 12M cells	101.1
IB2, 16M cells	99.4
IB3, 17M cells	100.8

Results Courtesy GM

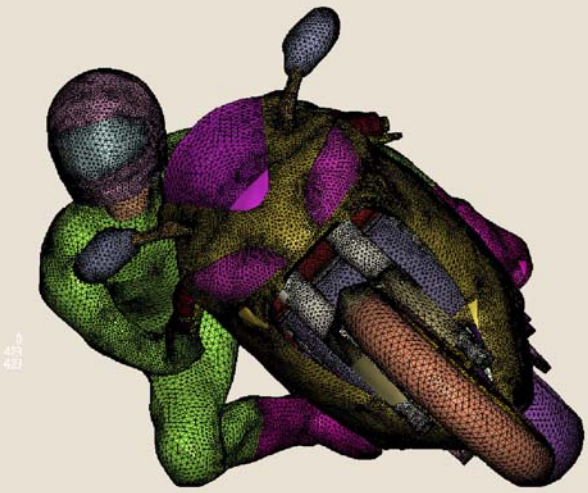


36

ANSYS FLUENT

Thank you very much, and ...

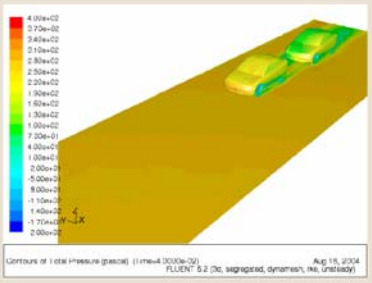
m1

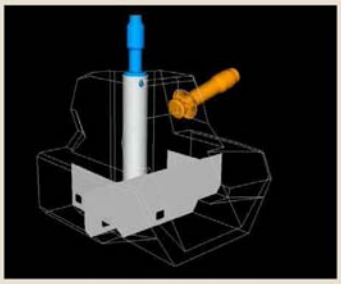


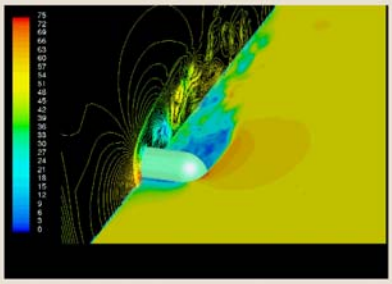
37

Questions ?

ANSYS FLUENT







39