CFD Topological Optimization of a Car Water-Pump Inlet using TOSCA Fluid and STAR-CCM+

Dr. Anselm Hopf
Dr. Andrew Hitchings
Les Routledge
Ford Motor Company
CONTENTS

- Introduction/Motivation
  - Optimization of 3D flows (CFD)
  - TOSCA Fluid (CFD topology optimization)
- Water-pump inlet of a car engine
  - Problem description
  - Results of original design
  - CFD Topological Optimization
  - Comparison original vs. optimized design
- Lessons learnt
INTRODUCTION – CFD DESIGN OPTIMIZATION

Optimization of 3D flows (rough classification, incomplete):

1) Parameter optimization
   - Sizing, parameters to be defined: e.g. diameter, radii, cone angle, location of gasket holes
   - Change parameters → 1D optimization problem
   - Numerous runs often solved with DOE
   - e.g. modeFRONTIER, HyperStudy, Optimate, ISight

2) Surface optimization
   - Surfaces have to be identified which should be optimized
   - Move nodes/vertices → 2D optimization problem
   - Often morphing of flow domain surface (again 1D parameter level)
   - Numerous runs often solved with DOE
   - e.g. CFD-code specific morpher, ANSA, Sculptor

3) Topology optimization
   - Design space has to be defined
   - Switch on/off finite volumes → 3D optimization problem
   - Result is a ‘Lego’ brick model which will be smoothed
   - e.g. TOSCA Fluid (first commercial software)
TOSCA FLUID (TOPOLOGY OPTIMIZATION FOR FLUID FLOWS)

- TOSCA Fluid is a new software system for non-parametric fluid flow optimization (topology optimization) producing an optimized 3D design of the flow region as input for a first CAD design.

- Topology optimization is well established and integrated in Ford product development for structural analysis and design.

- In the past no commercial topology optimization tool for fluid flows was available. Now, TOSCA Fluid is the first-in-industry software tool for fluid flow topology optimization.

- Applications:
  - Pressure Loss Minimization (e.g. air ducts, water jacket)
  - **Flow efficiency by uniform velocity distribution** (e.g. pump-inlet, heat exchanger, elbow tubes)
  - Flow balancing tasks (e.g. manifold, HVAC flow splitter)

- Currently restricted to steady-state, single-phase, isothermal, non-compressible flow physics. But this is often sufficient to have a first layout/impression of the new hull of the flow region.

- Currently implemented for STAR-CCM+ and ANSYS Fluent.

- Will reduce development time and improve result quality.

- Optimized fluid topology is essential for low emissions engines.

- Optimality criteria is to avoid flow recirculation

- Eliminate backflow and recirculation by sedimentation

Source pictures: FE Design/Dassault
WATER PUMP INLET OF A CAR ENGINE

Problem:

- Driven by package constraints, the shown coolant pipe has a nearly 180 degree coolant direction change in front of the water pump inlet.
- This will generate an inhomogeneous/non-uniform inlet flow at the water pump entry
- This might reduce the water-pump efficiency and might cause cavitation at the impeller
ORIGINAL DESIGN – CAD AND CFD MODEL

STAR-CCM+:

- CATIA CAD model available, but STAR-CCM+ wrapper may also be used to generate a closed inner hull of the flow region
- Standard meshing with polyhedron cells and two wall-layers
Original Design – Flow Velocity

- **Surface Uniformity**:
  - CFD_designspace_out: 0.868693
  - CFD_impeller: 0.742904
  - CFD_outlet: 0.90741

- **Uniformity**: 74.3%

- **Task**: minimize/avoid recirculation

---

**Notes**: Dr. A. Hopf, Dr. A. Hitchings, L. Routledge | Ford Motor Company

‘CFD Topological Optimization of a Car Water-Pump Inlet using TOSCA Fluid and STAR-CCM+’, STAR Global Conference 2014, Vienna
Workflow:

- Define design space (e.g. with CAD)
- Meshing as usual
- Define your boundary conditions
- Run optimization
- Run smoother
- Run analysis with optimized design
- Export result proposal to CAD
TOSCA FLUID OPTIMIZATION – DESIGN SPACE
TOSCA FLUID OPTIMIZATION – CFD MODEL

- **Inlet max. flow**
- **Outlet pressure outlet BC**
- **Trumpet shape for a smooth outlet flow**
- **Outlet mass-flow inlet BC (const. velocity)**

Recirculation will be eliminated during TOSCA Fluid run.
TOSCA FLUID OPTIMIZATION – POST-PROCESSING

[Image of TOSCA Fluid Optimization interface]

Write STL  ✔  Write VRML  ✔
Output Name  smooth_15000_pnt_spread_0.3_shrink_core.xml
Scale  1000.0
Output Surface  BOTH
Write Groups to STL  ✔

Slices
Generate Slices
Format of Slice Output iges_curves
Normal Vector  0.1
Number of Slices  40
Slice Distance (disables number of slices)  -1.0
Generate Slices along Primary Particle Tracks

Interfaces
Interface Treatment

TOSCA Fluid.post: TOSCA Fluid.smooth

VRML Viewer
user 4.60
sys 1.76
runhost: node name
real 3.88
user 3.46

Fri Nov 29 11:19 AM
COMPARISON OF FLOW DOMAIN – CAD AND CFD MODELS

Original design

TOSCA optimized design
COMPARISON – STREAMLINES

Original design

TOSCA optimized design

\[ v_{\text{max}} = -15.1\% \]

reduced main-jet velocity by larger cross-section

finally lower velocity at inner radius and more uniform flow at outlet

recirculation

impeller plane
COMPARISON – VELOCITY IN SECTION-CUTS

Uniformity = 74.3 %

Original design

improved uniformity!

Uniformity = 94.9 %

(+20.6 %)

TOSCA optimized design

reduced pressure drop!

$\Delta p = -70 \%$

Surface Uniformity CFD_designspace_out: 0.968693
Surface Uniformity CFD_impeller: 0.742904
recirculation

Surface Uniformity CFD_outlet: 0.90741

Surface Uniformity CFD_designspace_out: 0.978573
Surface Uniformity CFD_impeller: 0.949121
impeller plane
LESSONS LEARNT (BY TOPOLOGICAL OPTIMIZATION)

1) Most critical flow region is the inner lying radius in front of the outlet, which may generate a recirculation zone downstream.

2) Use trumpet shaped outlet and radius to keep the flow attached.

3) Increase cross-section/design space (in height and width) in front of outlet radius to reduce velocity intensity of flow jet.

4) Add bulge to set a contrary/compensating momentum to the 90 degree flow.

The final design of the water pump inlet was created using TOSCA Fluid and further fine tuning by surface morphing. The build version shows an improved velocity uniformity of 92% within the impeller plane.

For Ford this new workflow is highly efficient, reduces development time and improves result quality.
Q & A

Ford Research Centre Aachen
Aachen / Germany

Thank you!

Ford Dagenham Diesel Centre
Dagenham / UK
Problem:

Often an engine should fit into several vehicle applications and some parts have to be modified to match all package configurations. Driven by such package constraints, an inlet pipe to a water-pump inlet had to be redesigned resulting in a 180 degree coolant direction change in front of the water pump inlet. In the beginning the U-flow shaped design was generating an inhomogeneous and non-uniform inlet flow at the water-pump entry. This will reduce water-pump efficiency and might cause cavitation at the impeller.

Solution:

For this case the first-in-industry software tool for fluid flow topology optimization called TOSCA Fluid has been used, coupled with STAR-CCM+. TOSCA Fluid is a new software system for non-parametric fluid flow topology optimization producing an optimized 3D design of the flow region as input for a first CAD design. A possible flow region (design space) has to be defined which connects the inlet and outlet cores and is usually limited by clearance planes to other parts. After standard meshing with STAR-CCM+ the optimization is setup to minimize the backflow intensity by sedimentation and to maximize the velocity magnitude uniformity in the impeller plane. After the optimization the critical recirculation zone in front of the water-pump impeller and other backflow zones were eliminated and the pressure drop has been also reduced. Furthermore a more homogeneous flow distribution at the water-pump inlet has been reached by the optimization tool.

Benefit for Ford:

TOSCA Fluid is the first commercial software for topological optimization of CFD problems. Now, the CFD engineer may generate an initial design of new CFD optimized parts and give it to the designer. With this first proposal the flow improvements can be investigated helping to understand the local flow physics for further possible fine tuning of the flow with parametric and shape optimization methods. This new workflow is highly efficient, reduces development time and improves result quality. Optimized fluid topology is essential for low emissions engines.