Latest Development in OpenFOAM® (v1906/v1912)

4th German OpenFOAM User meetiNg - GOFUN

Dr. Sebastien Vilfayeau
(CFD Team Leader, ESI Gmbh)

22/04/2020

get it right®

www.esi-group.com

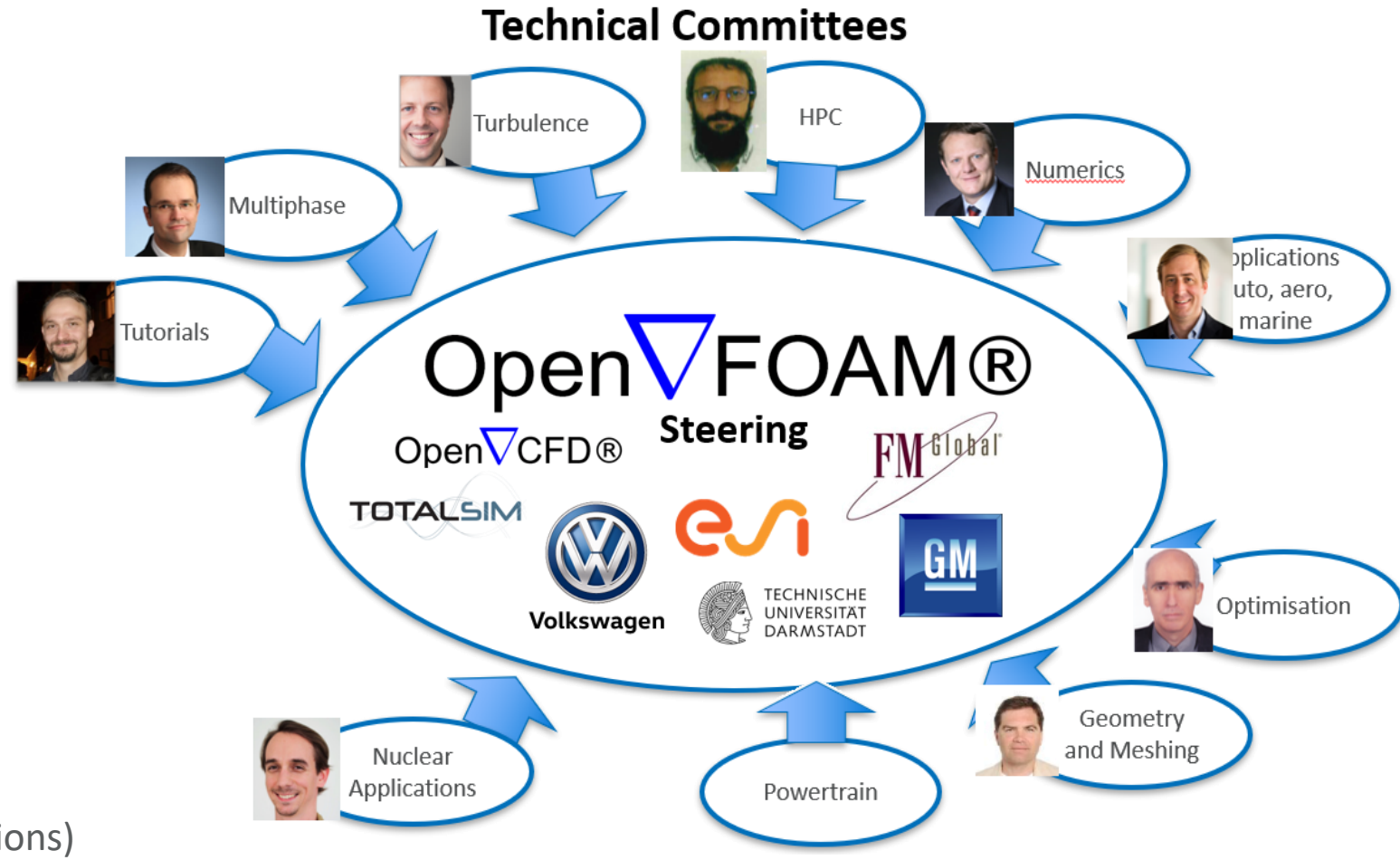
 Document Class: INTERNAL

Latest Development in OpenFOAM® (v1906/v1912)

Governance

- **Steering Committee**
  - (6) OEMs and Support Consultants
  - (3) Interests of Release Authorities
  - (2) Research Organizations
  - (1) Release and Maintenance
  - (1) Permanent Chair

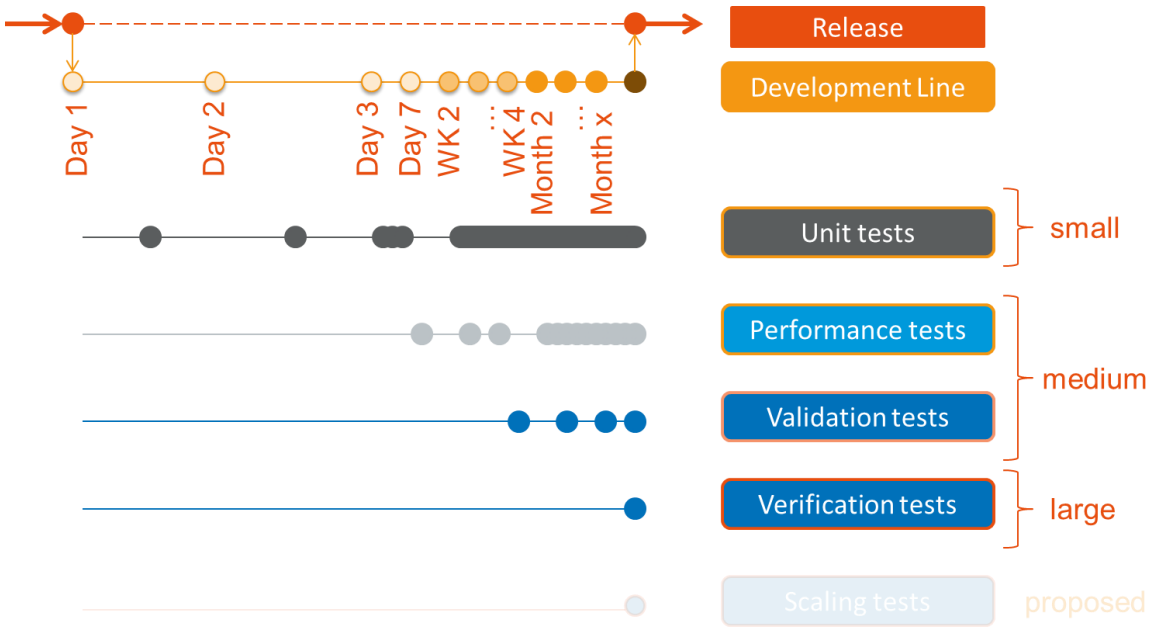
- **Technical Committees**
  - Documentation
  - Marine Applications
  - Meshing
  - Multiphase
  - Numerics
  - Optimisation
  - Turbulence
  - Nuclear
  - HPC
  - … others (core technologies and applications)



Latest Development in OpenFOAM® (v1906/v1912)

Quality Assurance

- Quality Assurance:
  - Small (unit) test loop
    - ***Nightly*** tests to ensure no cross-feature breakage
    - Approximately 750 feature-by-feature tests
    - Execution time ~ 4 hours (nightly)
  - Medium test loop
    - Tutorials and small validation tests
    - Approximately 450 tests
    - Execution time ~ 2 days (***weekly***)
  - Large test loop
    - ~25 Client cases
    - Several million steady and transient cases
    - Execution time ~ 1 week (***once per release***)



ISO 9001 BUREAU VERITAS Certification

ISO 9001 Certified Company

Latest Development in OpenFOAM® (v1906/v1912)

Highlights

Preprocessing: Integration of swak4Foam Functionalities

Mesh: snappyHexMesh automatic gap closure

Solver: adjoint shape optimization

Performance: GAMG, mixed-precision

Application: paint dipping, Solar Load Modeling (Electrical Vehicules),

Virtual Wind Tunnel for WLTP aerodynamics certification

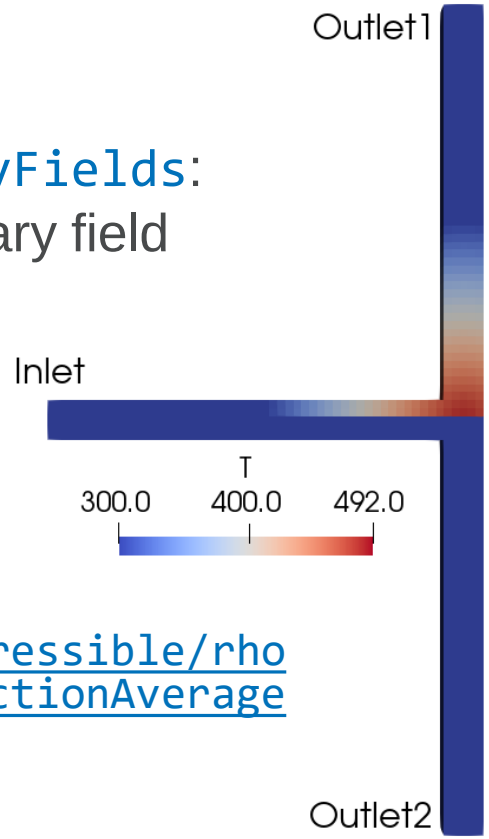
Latest Development in OpenFOAM® (v1906/v1912)

Integration of swak4Foam Functionalities

Expressions-based setFields utilities

↔ swak4foam *funkySetFields, groovyBC*

- `setExprFields`:
  for custom fields
- `setExprBoundaryFields`:
  for custom boundary field entries



[$FOAM_TUTORIAL/compressible/rhoPimpleFoam/RAS/TJunctionAverage]

e.g. `setExprFields` to adjust the temperature within a defined zone

```
expressions
(
    T
    {
        field       T;
        dimensions  [0 0 0 1 0 0 0];
        constants
        {    centre (0.21 0 0.01);    }

        variables
        (    "radius = 0.1"    );

        condition
        #{
            // Within the radius
            (mag(pos() - $[(vector)constants.centre]) < radius)
            // but only +ve y!
          && pos((pos() - $[(vector)constants.centre]).y()) > 0
        #};

        expression
        #{
            300
          + 200*
            (1 - mag(pos() - $[(vector)constants.centre])/radius)
        #};
    }
);
```

Latest Development in OpenFOAM® (v1906/v1912)

Integration of swak4Foam Functionalities

Inline expressions: **#calc ⇒ #eval**

- Faster & convenient replacement for #calc
- No compilation, inline interpreter

…to conveniently recalculate things

```
angle    37.5;
angle    #eval{ degToRad($angle) };
```

…to embed mathematical evaluation into regular string expansions

```
#include "<system>/sampling${{ round(${flowRate:-5} * 100) }}";
```

More info

v1912 User Upgrade Guide:
https://develop.openfoam.com/Development/openfoam/wikis/guides/upgrade/v1912-User-Upgrade-Guide

Latest Development in OpenFOAM® (v1906/v1912)

Integration of swak4Foam Functionalities

Inline expressions: **#calc ⇒ #eval**

- Faster & convenient replacement for #calc
- No compilation, inline interpreter

…to change the way you view dictionaries!

```
kgh     700;
split   0.4;

inlet1
{
    type            flowRateInletVelocity;
    massFlowRate    constant #eval{ $split * $kgh / 3600 };
}

inlet2
{
    type            flowRateInletVelocity;
    massFlowRate    constant #eval{ (1-$split) * $kgh / 3600 };
}
```

More info

v1912 User Upgrade Guide:
https://develop.openfoam.com/Development/openfoam/wikis/guides/upgrade/v1912-User-Upgrade-Guide

get it right®

Latest Development in OpenFOAM® (v1906/v1912)

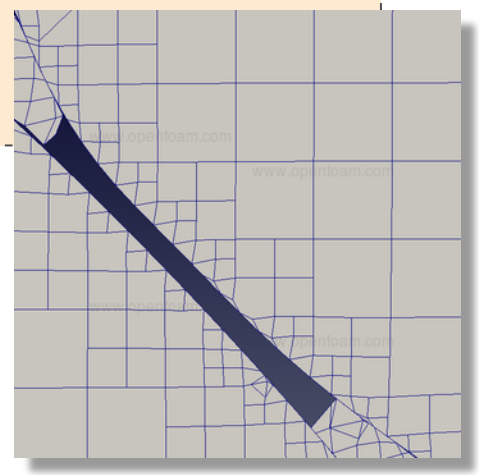
SnappyHexMesh (automatic gaps closure)

- Sometimes gaps need to be resolved (not meshed)
- New control in surface refinement section

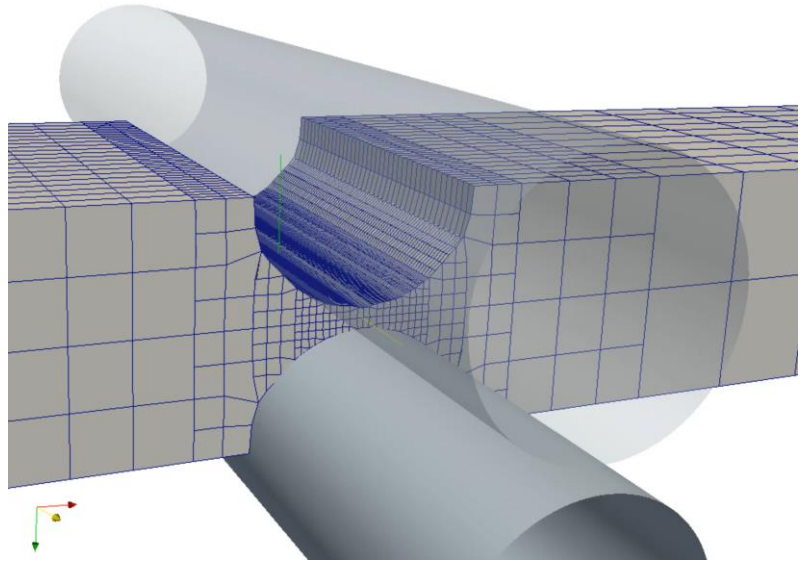
<system>/snappyHexMeshDict

```
refinementSurfaces
{
    "gap.*"
    {
        // Surface-wise min and max refinement level
        level       (2 2);

        // From cell level 2 onwards start checking
        // for opposite surfaces
        blockLevel  2;
    }
}
```



Automatic gap refinement is in OpenFOAM for some time



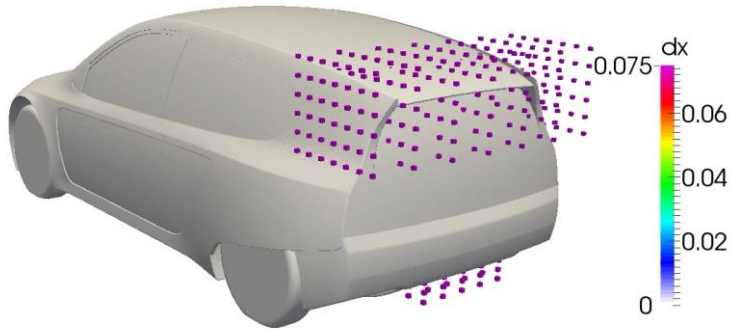
<system>/snappyHexMeshDict

```
gapLevel (2 0 10);
gapMode outside;
```

Latest Development in OpenFOAM® (v1906/v1912)

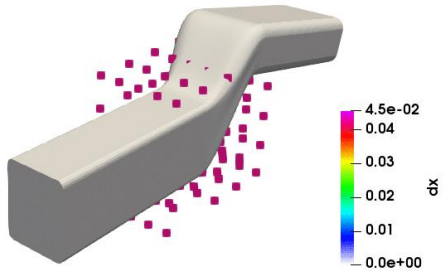
Adjoint shape optimisation

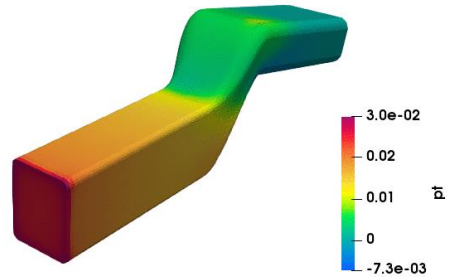
- `adjointOptimisationFoam` solver runs entire loop
  - solution of the flow
  - adjoint equations
  - computation of sensitivity derivatives
  - update of the design variables and mesh
- Implementation is contributed by OpenFOAM Governance
- The software was developed by PCOpt/NTUA and FOSS GP



To master the adjoint shape optimisation we do recommend users to take a 2 days training lead by two of the developers of the method and implementation: Prof. Giannakoglou and Dr. Papoutsis-Kiachagias

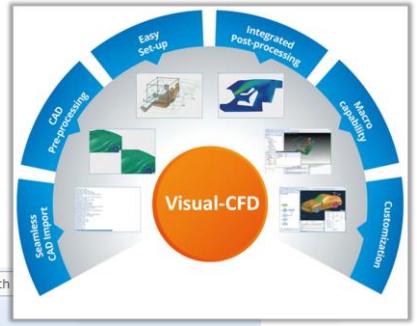
Next training will be virtual
3.-4. June 2020
See www.openfoam.com/training





Latest Development in OpenFOAM® (v1906/v1912)

Adjoint shape optimisation – **Visual-CFD**



Visual-CFD 15.5.0 - naca0012

Applications File Edit View Model Tools Window Help

Explorer

naca0012

- naca0012
  - Model Options
    - Multiphase
    - Discrete Phase Modeling
    - Scalar
    - Radiation
    - Adjoint Optimisation
      - Optimisation Parameters
      - Solver Parameters
      - Control Shape
  - Mesh
  - Surface Parts (4)
  - Interface Parts
  - Primitive Parts

Simulation Control

Solve Options	
Solve	
No. of Cycles	5
Save options	
Write Interval	1
Parallel Settings	

Save Project Save VDB

Preserve Boundaries

Surface Parts

Parallel Settings (Machines):

Machines: Local Host

Hosts File

Command:

Abort Run Close

P1W1 P1W2

Spalding wall function is recommended with all Spalart Allmaras models
Reading project... <D:\V-CFD\Testing\naca0012>
Model Name = naca0012
Read and displaying mesh from constant/polyMesh
Model unit system is set to M
Project <D:\V-CFD\Testing\naca0012> read successfully
Spalding wall function is recommended with all Spalart Allmaras models
Please select at least one Surface Part

Console Command

Ready

Latest Development in OpenFOAM® (v1906/v1912)

Improved GAMG solver controls

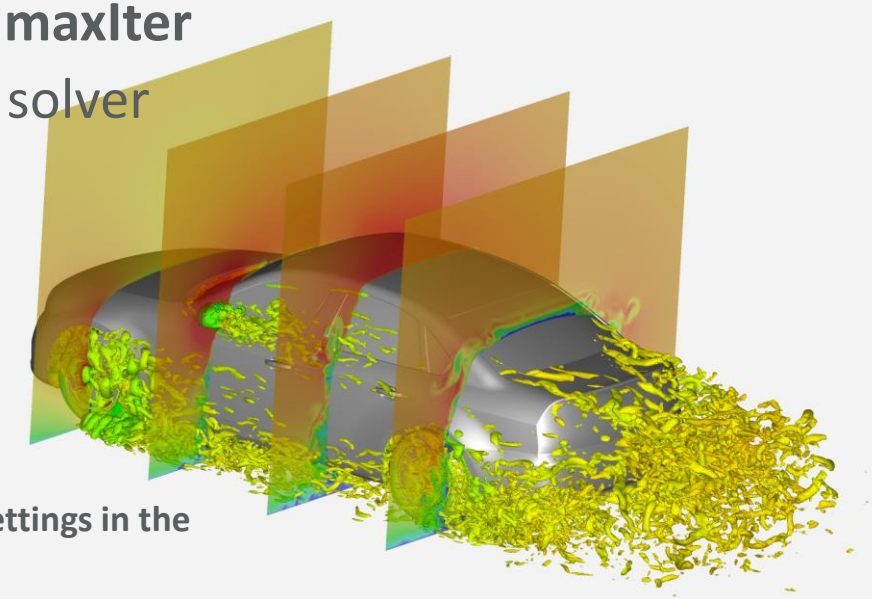
- In parallel runs communication on the coarsest level could be bottleneck
  - each cell at different processor
- User can now lower the number of sweeps by using **relTol** and **maxIter**
- Example shows drastic reduction in number of sweeps for PCG solver

```
pFinal
{
    solver          GAMG;
    tolerance       1e-6;
    relTol          0;
    smoother        GaussSeidel;

    coarsestLevelCorr
    {
        solver          PCG;
        preconditioner  DIC;
        relTol          0.05;
    }
}
```

Global solver settings

Solver settings for coarsest level. If not defined here, the global settings in the parent-dictionary is taken



For unsteady external aero simulation with 100M cells,
5% performance improvement is observed

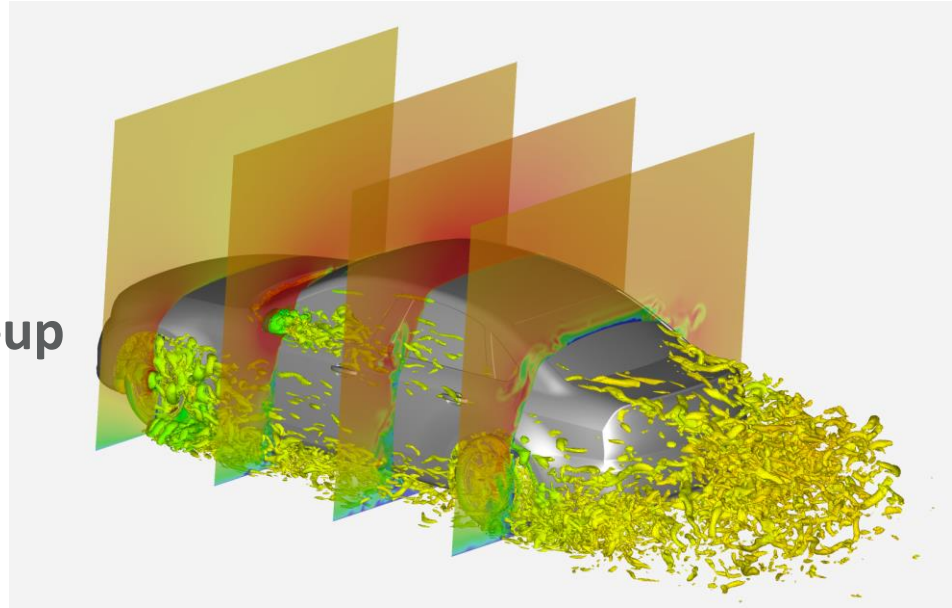
Latest Development in OpenFOAM® (v1906/v1912)

Mixed precision

- New compiling option to compile linear solvers in double precision and rest of the code in single precision

```
wmSPDP
```

```
environment variable WM_PRECISION_OPTION=SPDP
compilation flag WM_SPDP
```



- Single precision runs need **40% less memory**
- Mixed precision needs **30% less memory,** offers **20% speed-up**
- Using mixed precision was tested for robustness on external aerodynamics cases
- Files are written in single precision
- **N.B.:** doesn't work yet with AMI for rotating wheels

Testing on a driveAir model with 64 million cells,
Transient run resulted in ~ 20-25% speed-up

Latest Development in OpenFOAM® (v1906/v1912)

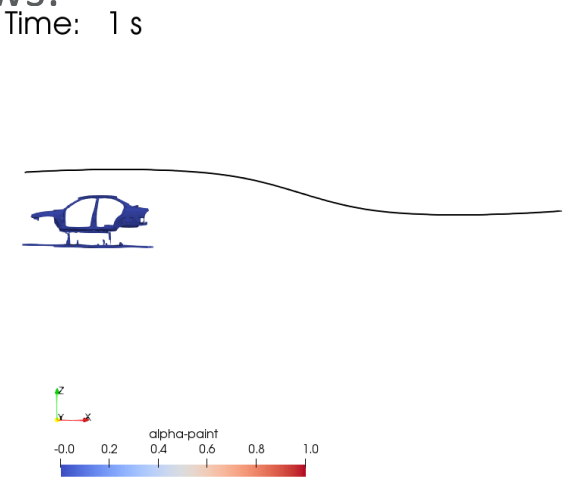
Application: Paint Dipping

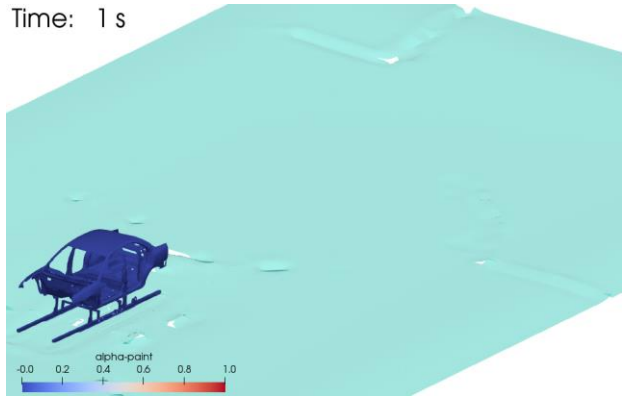
- **VOF** - New restrains in 6DoF
  - `linearSpringDamper` restrain
  - Acts as soft-rope model when a distance between `anchor` and `refAttachmentPoint` exceeds the `restLength`
  - When the rope is slack there is no force applied to body
- The specification in dynamicMeshDict is as follows:

```
restraints
{
    chain
    {
        sixDoFRigidBodyMotionRestraint  linearSpringDamper;
        refAttachmentPt     (3.7 4.2 4.5);
        anchor              table
        (
            (0          (3.7 4.2 6.5))
            (2          (4.7 4.2 6.5))
            (6          (5.7 4.2 4.5))
            (8          (6.7 4.2 4.5))

        );
        psi                 1;
        wn                  8.28;
        numberOfChains      4;
        restLength          2;
    }
```

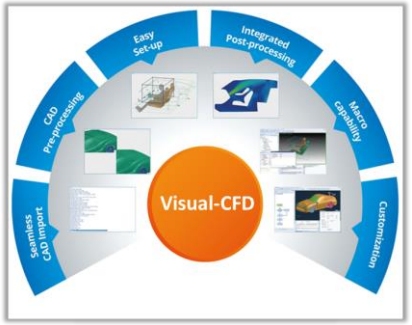
Mechanically driven dip coating with an chain drive (Source: https://www.durr.com)





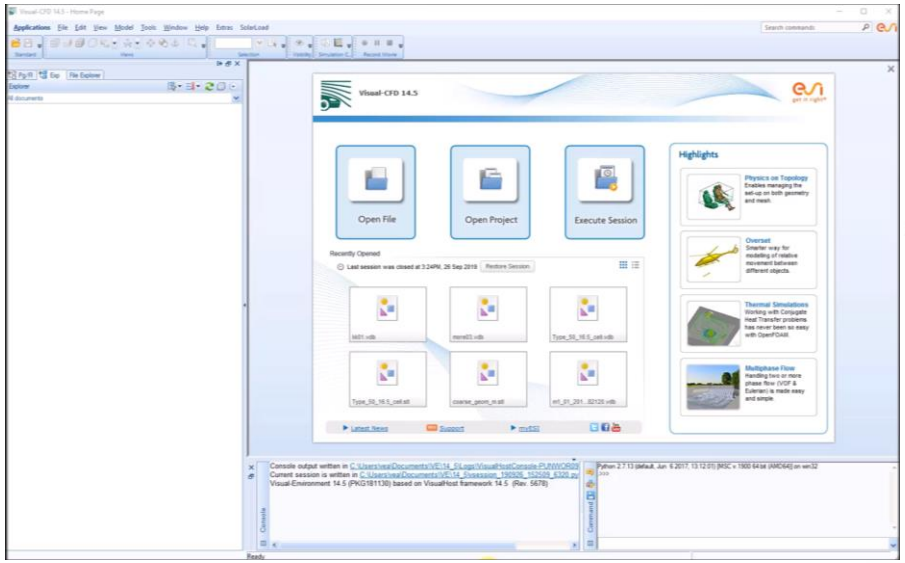
Latest Development in OpenFOAM® (v1906/v1912)

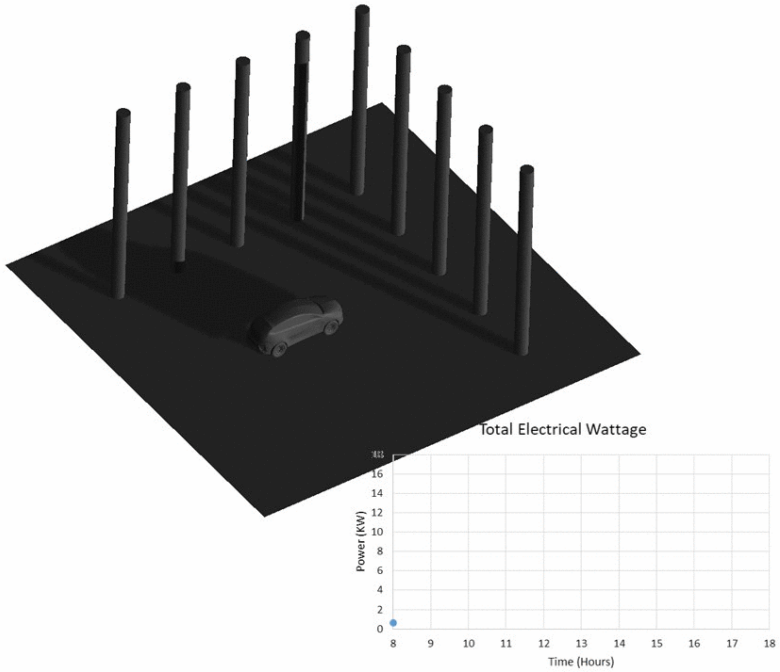
Application: Solar Load Modeling (Electrical Vehicules)



- Solution to leverage the maximum power of solar energy into electrical energy
  - **New collimated solar beam model for fvDOM**
    - The fvDOM radiation model now includes support for external beams from a solar external load to provide a straightforward means to include solar load in the domain.
  - **New multi band zone absorption-emission model**
    - enables the use of non-uniform absorptivity and emissivity fields in the domain (trees and background)

New Hyundai Sonata with solar panel (2019)

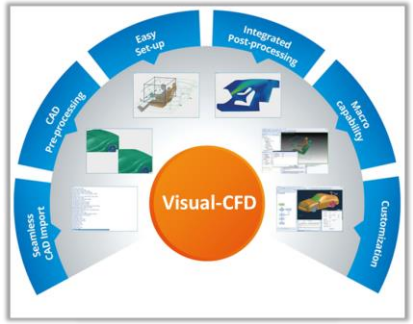




get it right®

Latest Development in OpenFOAM® (v1906/v1912)

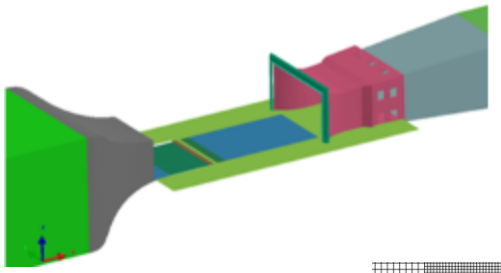
Application: Virtual Wind Tunnel for WLTP aerodynamics certification

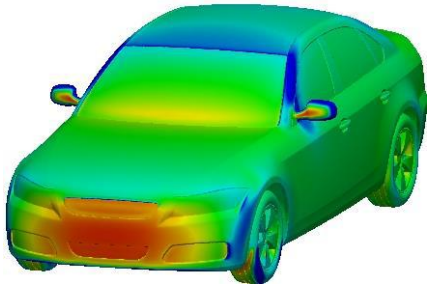


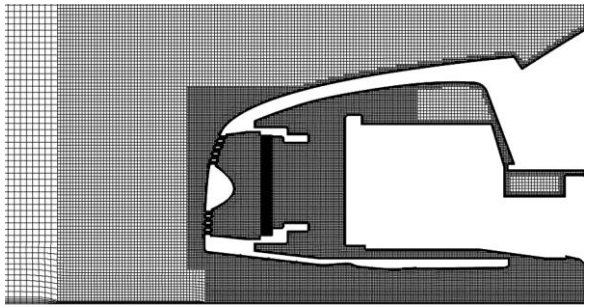
- **Features:**
  - Wind Tunnel Calibration: Virtual Wind Tunnel, rolling belt
  - Baseline versus design variants: spoilers, grills, rims, etc

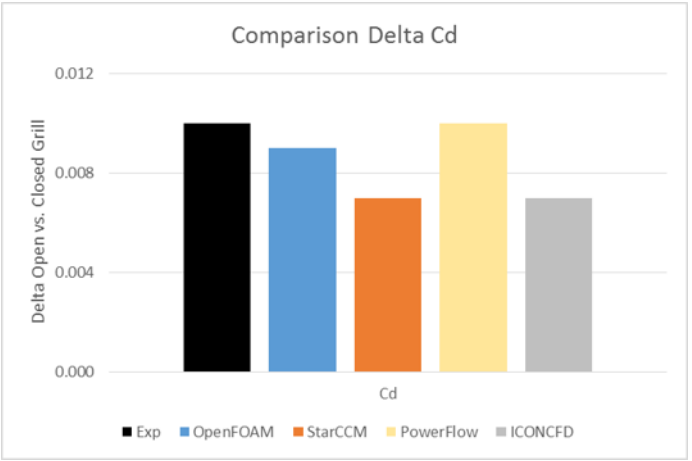
Example of individual vehicle variants

Test configurations per vehicle					
	Wheel A			Wheel B	Wheel C
Baseline	OPEN GRILLE	50% OPEN	CLOSED	OPEN GRILLE	OPEN GRILLE
Optional Equipment A	OPEN GRILLE	-		-	-
Optional Equipment B	OPEN GRILLE	-		-	-









- Automated process from CAD clean-up to results
- HPC Parallel scalability to several 1000s cores
- Post-processing automatically done in the cluster
  - Mesh pictures, section cuts, contours, plots.
  - Automated report in pdf or xml format

http://ecara.org/

Latest Development in OpenFOAM® (v1906/v1912)

Upcoming events

- 8th OpenFOAM Conference
  - 13-15th October 2020 in Berlin
    - 2 days Keynote and Technical Presentation
    - 1 day Workshops
  - Call for papers will open soon, until 31st M
  - www.esi-group.com/openfoam2020

THANK YOU!

SVU@ESI-GROUP.COM