



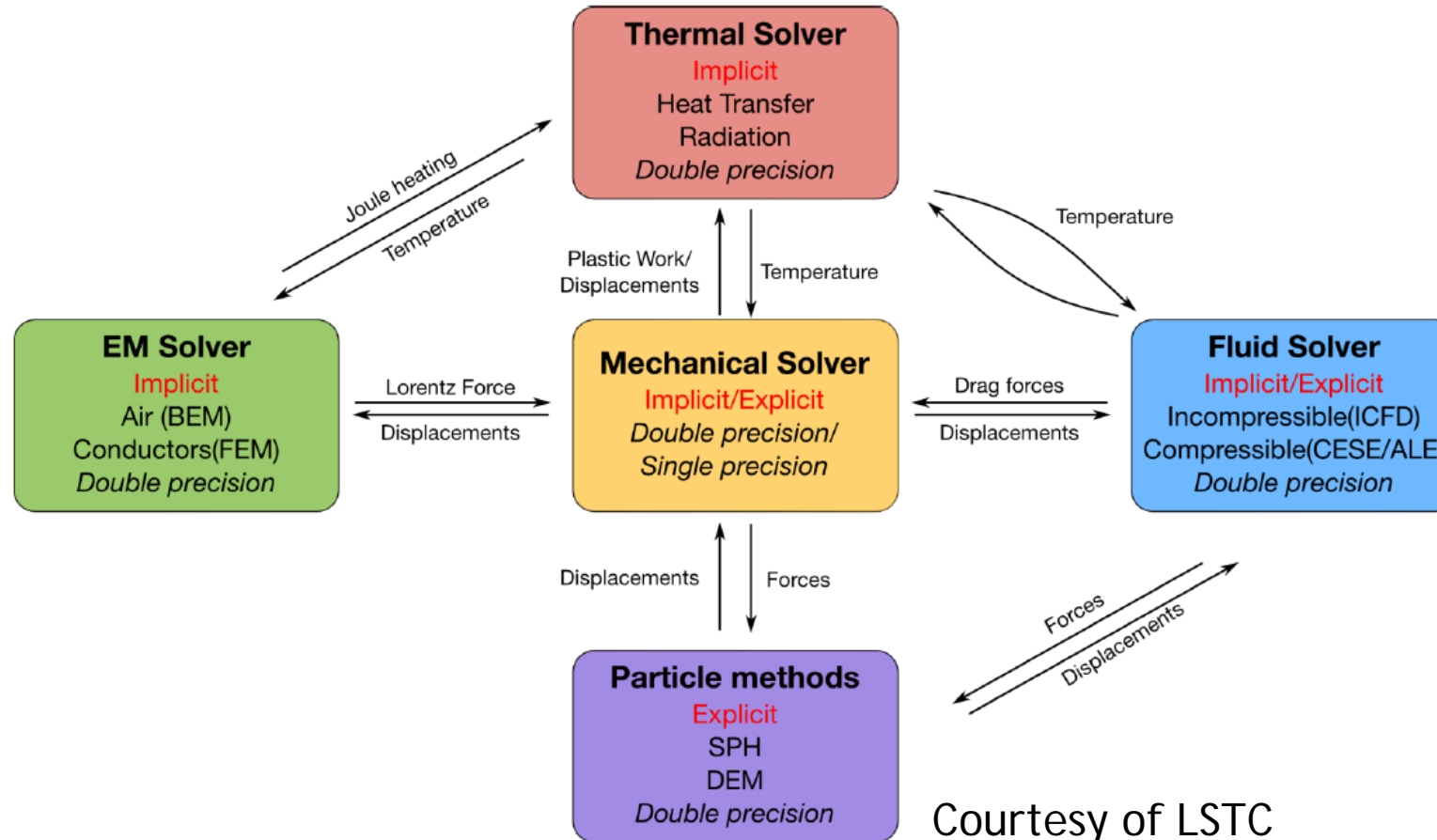
# Fluid structure interactions

Marcus Timgren

[marcus.timgren@dynamore.se](mailto:marcus.timgren@dynamore.se)

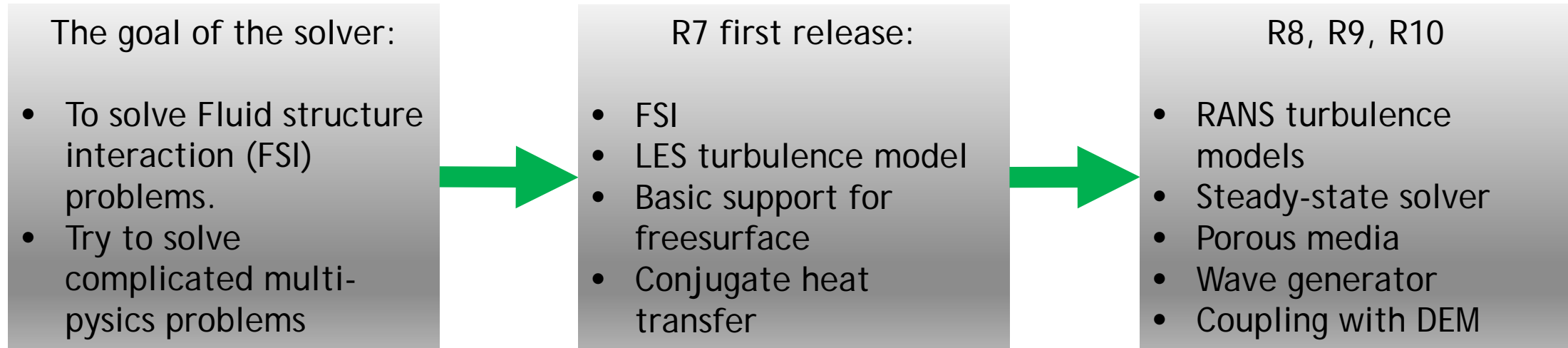
# FSI coupling options

- A general overlook of the coupling capabilities:



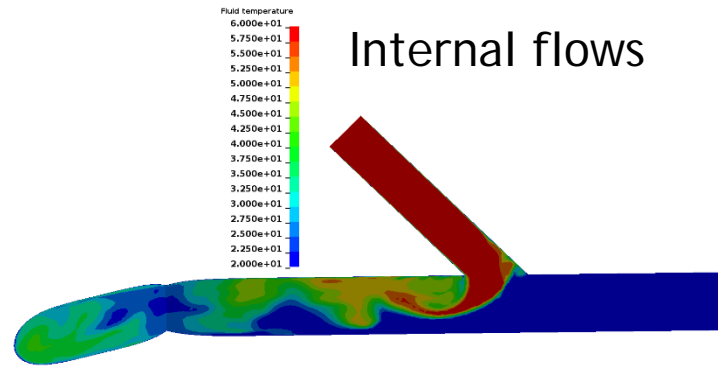
Courtesy of LSTC

# History of the ICFD solver

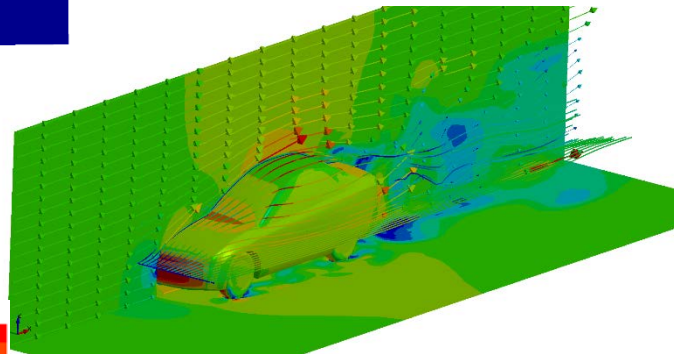


- The goal with FSI problems has a big impact on the development decisions.
- Started as a transient solver but from R10 it has a steady-state solver.
- The development is mostly based on user requests.

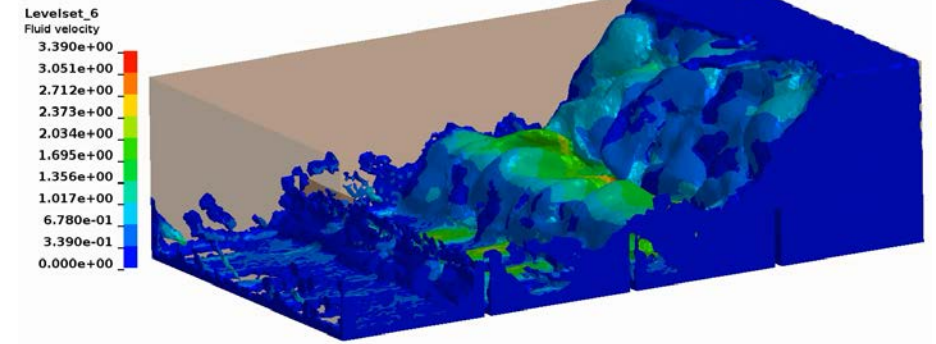
# Classical CFD applications



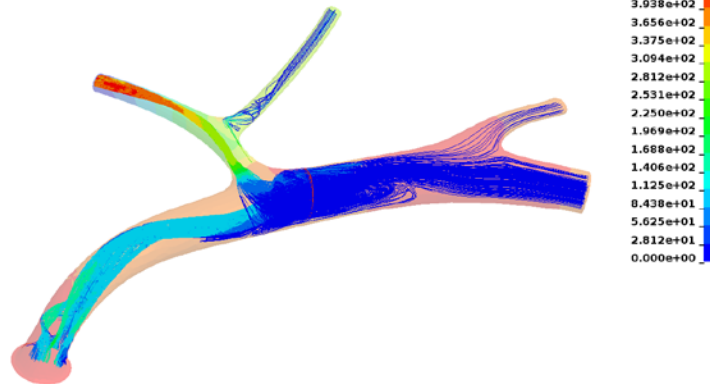
Aerodynamics



Sloshing

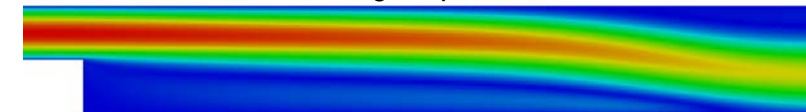


Simulation of human arteries

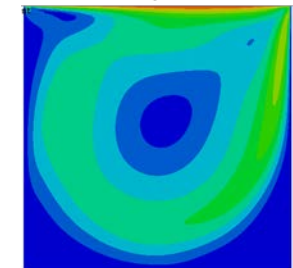


Steady-state simulations

Backward facing step Re 800



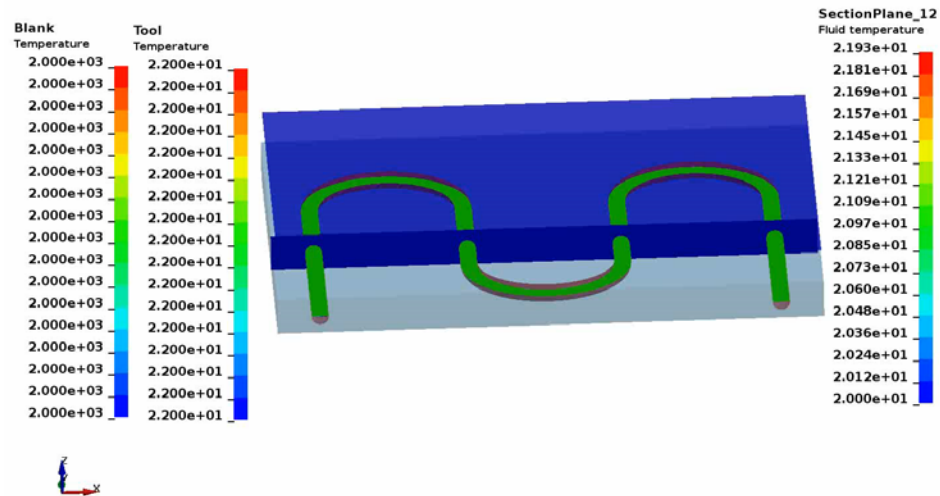
Driven cavity Re 1000



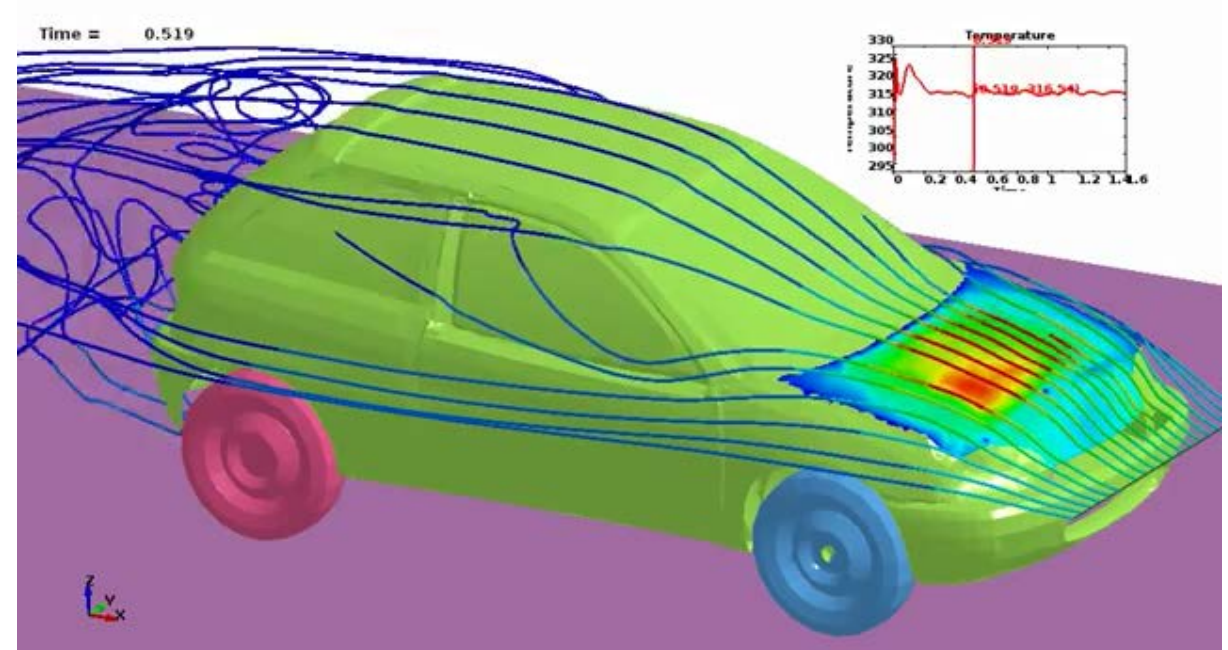
# Conjugate heat transfer examples

## Tool cooling example

LS-DYNA keyword deck by LS-PrePost  
Time = 0

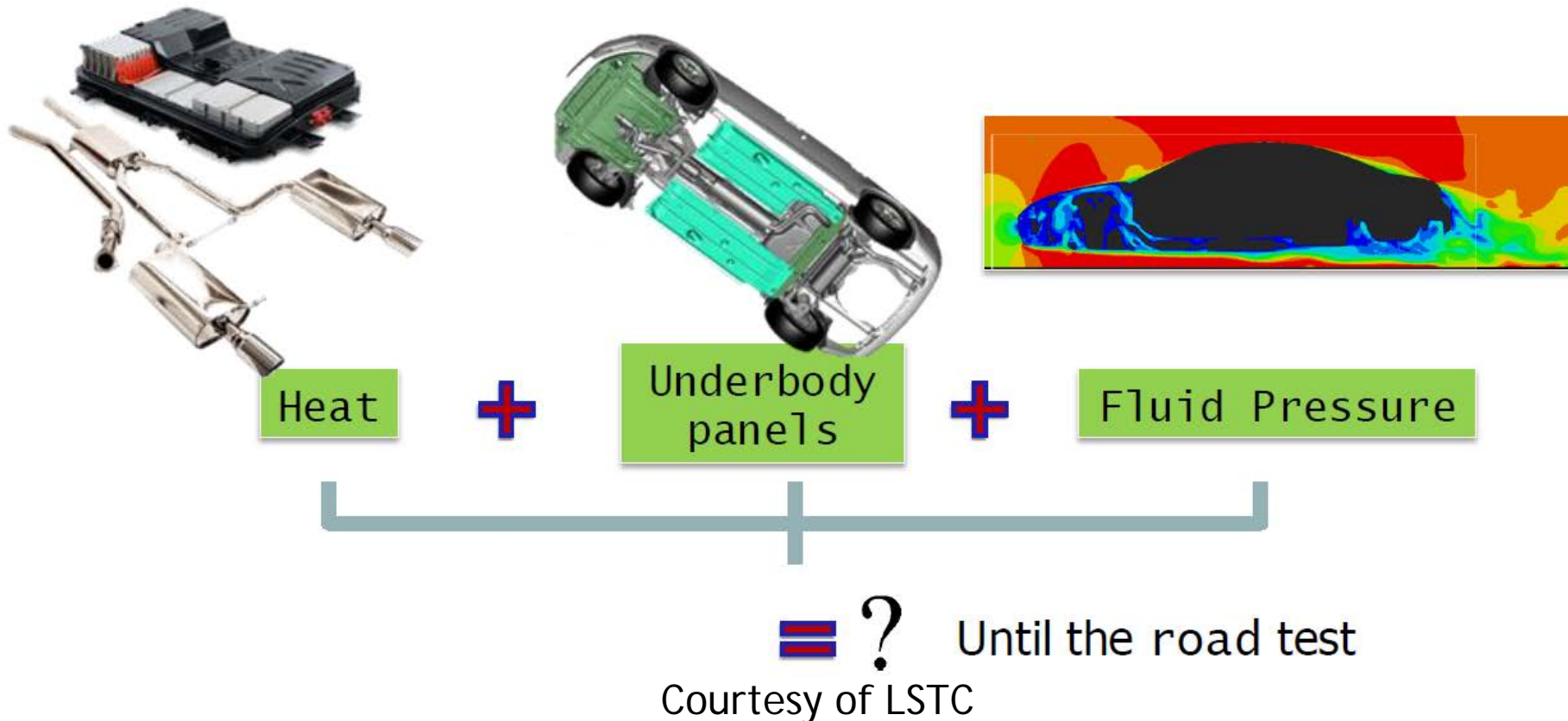


## Cooling of a car hood



# Why the need for FSI simulations?

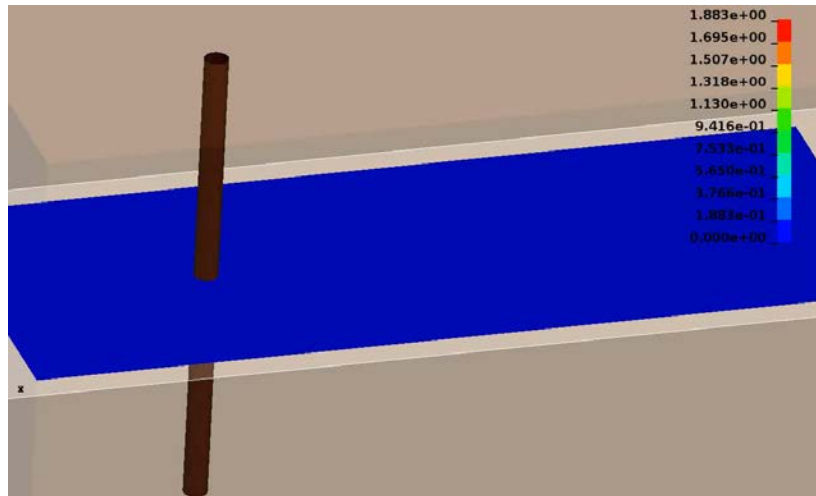
- To get results earlier in the design process, before road tests.
- Unknown boundary conditions.
- The structure and fluid affects each other.



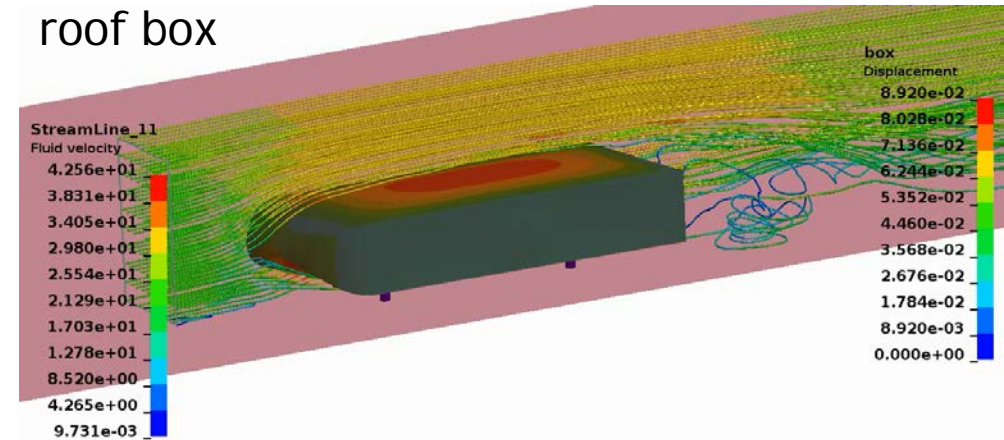


# FSI applications

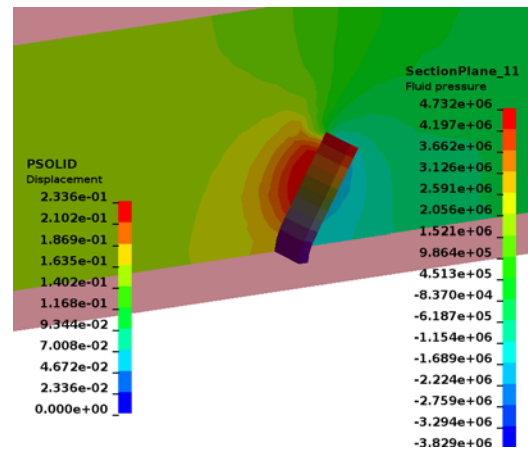
## Vortex induced vibrations



## FSI simulation of a “weak structure”, a roof box



## Flow over a plate



# FSI coupling options

- Type 1: Indirect/linear coupling
  - It is possible to run a CFD simulation and extract the pressure loads.
  - Outputted in a format compatible with a structural input deck.
- Type 2: The same coupling can be used with the steady-state solver
  - This means that a steady-state CFD simulation runs until convergence.
  - Then it continuous with the structural simulation.
- Type 3: The transient/full FSI coupling

ICFD simulation

Structural  
simulation

ICFD simulation  
And  
Structural  
simulation

ICFD simulation

Structural  
simulation





# FSI coupling options

- Coupling options for the full FSI coupling:
  - One way coupling
  - Two way coupling
    - Strong or weak coupling

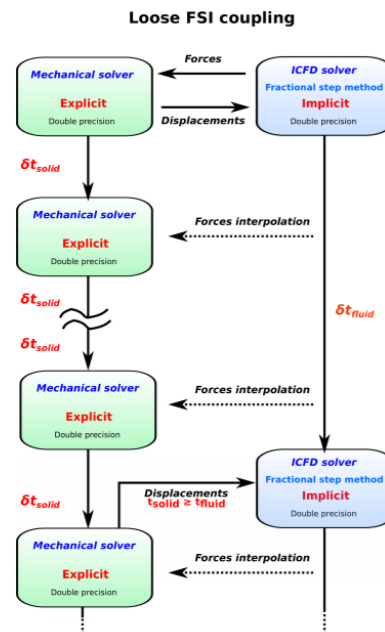


Figure 2: Loose FSI interaction resolution scheme.

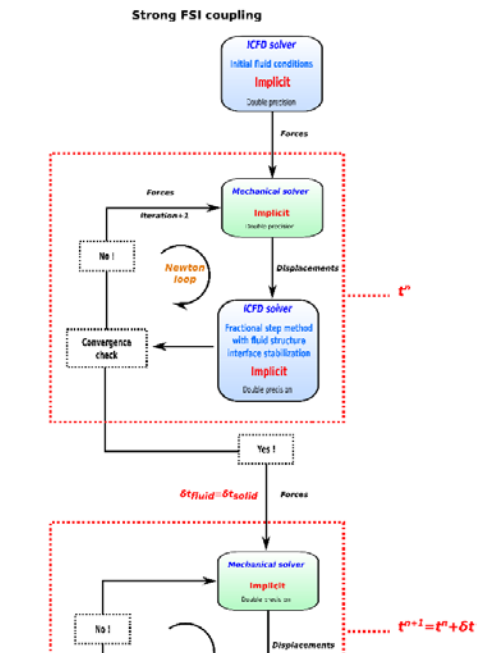
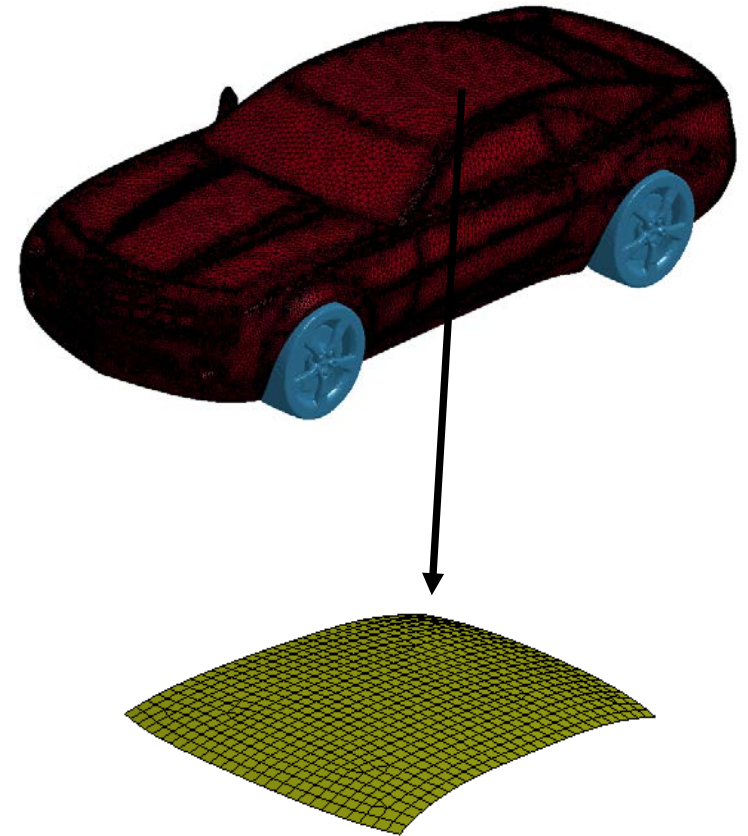


Figure 3: Strong FSI interaction resolution scheme.

Picture from the LSTC theory manual

# FSI setup up

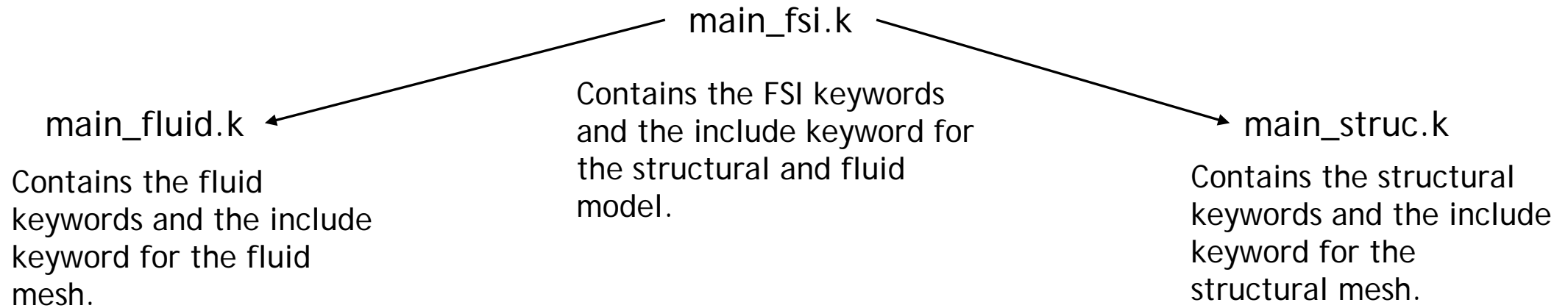
- What are the steps to create a FSI simulation?
  - First create your CFD and structural model.
    - The ICFD solver have an automatic volume mesher, only the surfaces has to be meshed.
  - Determine the area that interacts with the fluid.
    - Create a fluid part for that area or extract that area from the structural model.
  - Activate the FSI coupling by adding \*ICFD\_CONTROL\_FSI and \*ICFD\_BOUNDARY\_FSI.
  - Create a input file that includes the CFD and structural input decks.
  - Run the simulation.



Structural mesh of the roof

# FSI setup up

- Typical FSI input deck



- This makes it possible to collaborate between the structural and CFD group in an easy way.
- Possible to extract certain parts that is affected by the flow.

# Examples and benchmark cases for the ICFD solver

- For more information check these sites out:
  - Dynaexamples has a lot of ready to run examples.
    - <http://www.dynaexamples.com/>
  - LS-DYNA Corporate Tutorial & Content a YouTube channel with a couple of tutorials about the examples at dynaexamples.com.
    - <https://www.youtube.com/user/LSTCandDYNAmore>
  - LSTC also has a multi-physics YouTube channel, it focus more on showing what the different solvers can do.
    - <https://www.youtube.com/user/980LsDyna>
  - LSTC website has some benchmark problems and some other information.
    - [http://www.lstc.com/applications/new\\_multiphysics](http://www.lstc.com/applications/new_multiphysics)

Thank you!



Your LS-DYNA distributor and  
more

