

# ANSYS 19.1 Update

---

## Fluid Dynamics

### ANSYS EnSight

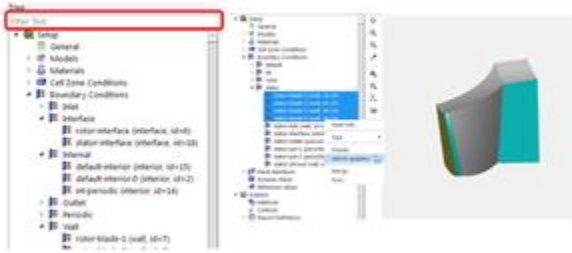
- High-end postprocessing with ANSYS EnSight R19.0 or higher included in
- ANSYS CFD Premium
- ANSYS CFD Enterprise
- ANSYS CFD PrepPost



*CFD postprocessing with ANSYS EnSight*

### ANSYS CFD / Fluent

- User interface - efficient pre- and postprocessing through:
- Display of boundary conditions via the structure tree
- Filter structure tree content
- Storage of user-defined settings (colors of the graphics window, mouse button assignment, etc.)
- Monitoring simulations at runtime in batch mode with the help of a visualization client
- Improvement in post-processing: Faster creation of isosurfaces, significant reduction in rendering time for streamlines and vectors



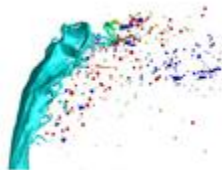
Filter structure tree content



Storage of custom settings

- Faster to the Result - Solver:
  - Acceleration of transient simulations with rotating fluid areas
  - Improved parallel scalability for heat transfer simulations
- New Applications - Applications, Modeling:
  - Multi-phase flows: coupled joint calculation of free surfaces (modeling as "Volume of Fluid") and distributed drops (modeling by Lagrange-Tracking) allows more efficient and accurate simulation of, for example, sputtering processes
  - Consideration of wall deformations due to wear caused by the impact of particles.
  - SCR (Selective catalytic reduction): Risk assessment for urea deposits possible
  - Wall film:
    - Mixing ratios in the wall film,
    - larger possible time steps
    - Propagating particles across multiple cell facets
  - Phase change: model extended to Euler wall-film model
  - Sharp and dispersive phase boundaries simultaneously within a model

- Variable time steps for all multiphase models
- Alternative modeling of heat transfer during mass transfer (explicit calculation of the heat of vaporization)

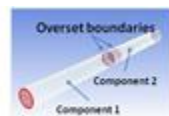
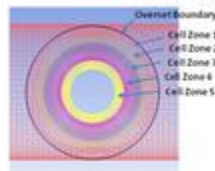


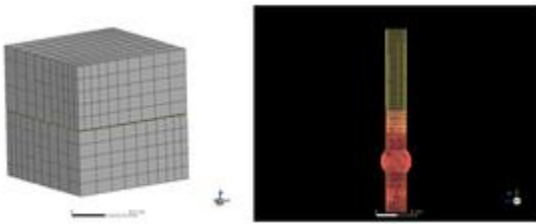
*Multiphase Flow*



*Sharp and dispersing phase boundaries simultaneously*

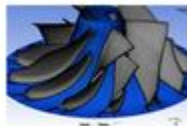
- Form finding using the adjoint solver:
  - Integrals over iso-intersections can be used as target size
  - Design tools allow the direct definition of shared parameters
  - Rigid body boundary conditions possible
- Adapted networking for individual tasks:
  - Overlapping and nested meshes with several internal areas possible
  - Overlapping as interfaces between different components
  - FSI or moving nets: deformation of mesh interfaces with or without overlap



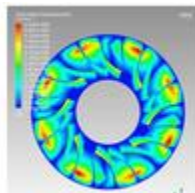


## ANSYS CFD / CFX

- Fast to the model with the turbo tools:
  - "Dual splitter" template available
  - Modeling vane fillets directly in the turbo tools
  - Modeling of variable edge gap thicknesses

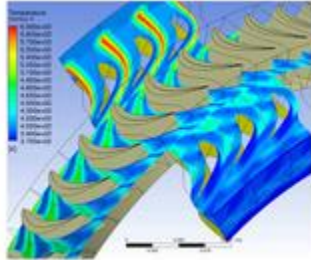


- Modeling of blade flutter:
  - Now also possible for entire wheels
  - Workflows for complex eigenmodes available



- Interaction of blade rows:
  - Harmonic analyzes for
    - FT-ID ("Fourier Transformation - Inlet Disturbance")
    - Equal pitch, PT-TRS ("Profile Transformation - Transient")
  - Improvement in the mixing-plane approach
  - Improved approach to the analysis of cavitation: Simulation of cavitation behavior without empirical approximation models [19.1]

- Consideration of solid domains in harmonic analyzes (blade row method, CHT) [19.1]



### System coupling (fluid-structure interaction) [19.1]

- Command line driven workflow for setup of FSI analyzes [19.1]
  - Focus on definition in the Linux environment [19.1]
  - Based on new Systems Coupling 2.0 engine [19.1]
  - Setup and start of the analysis outside the Workbench environment Command lines based (cmd, shell) [19.1]
  - Definition of the Mechanical / CFD setups from the respective simulation environments [19.1]

```

System Coupling Console [19.1]
=====
Coupled Solution
=====
Resolving connections from coupling participants... done

Participant Build Information
-----
Fluid Flow (Fluent)
ANSYS Fluent 19.1.0, Build Year: Feb 19 2018 15:05:57, Build No: 14, OS:
Windows 64-bit
ANSYS Mechanical (Mechanical)
ANSYS Mechanical APDL Release 19.0.0.0, Build No: 190000, OS: Windows 64-bit
ANSYS CFX (ANSYS CFX)
ANSYS CFX Release 19.1.0, Build No: 191000, OS: Windows 64-bit

=====
Coupling Iteration = 1
=====
Mapping Summary
-----
Participant 1
Block Mesh: 100, 100
Mapped Nodes (N): 100, 97
Mapped Elements (E): 100, 100
Participant 2
Mapped Nodes (N): 100, 100
Mapped Elements (E): 95, 100
Mapped Nodes (N): 100, 100
Participant 3
Block Mesh: 100, 100
Mapped Nodes (N): 100, 100
    
```

System Coupling Console [19.1]

All information has been prepared to the best of our knowledge.  
Information provided without guarantee.