

ANSYS Release 19.2 - Fluids Update

ANSYS EnSight

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

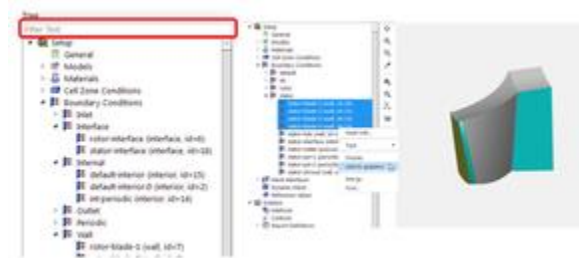


CFD postprocessing with ANSYS EnSight

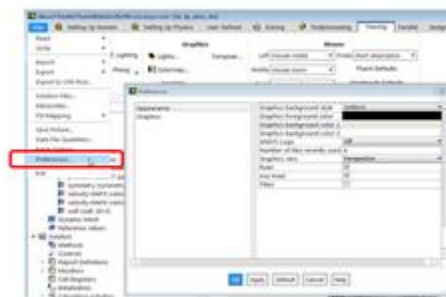
ANSYS CFD / Fluent

- The new patented Mosaic™ technology automatically combines different meshes with the help of polyhedron elements, enabling fast resolution of flow problems with high accuracy:
 - Surfaces: triangular, quadrilateral and polygon elements [19.2]
 - Volume: hexahedrons, tetrahedrons, pyramids, prisms [19.2]
 - Compound of elements is always conforming [19.2]
- Efficient preprocessing:
 - Target-oriented workflows for "watertight" geometries completely within Fluent allow efficient preprocessing [19.2]
 - Significantly faster than before [19.2]
 - Less "clicks" and interactions required [19.2]
- Expressions in Fluent: no more UDFs required for eg the definition of profiles or transient boundary conditions [19.2]
- User interface - efficient pre- and postprocessing through:
 - Display of boundary conditions via the structure tree

- Filter structure tree content



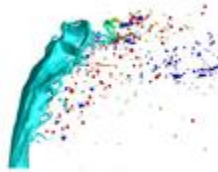
Filter structure tree content



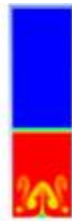
Storage of custom settings

- 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
- Faster to the Result - Solver:
 - Acceleration of transient simulations with rotating fluid areas
 - Improved parallel scalability for simulations with heat transfer
- 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)

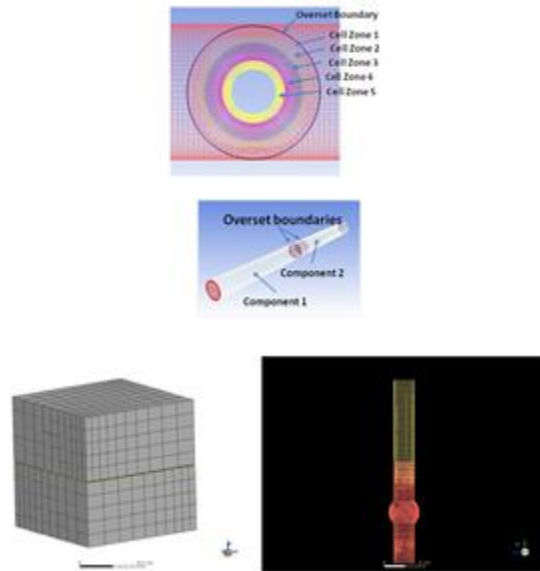


Multiphase Flow



Sharp and dispersive 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

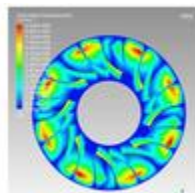


ANSYS CFD / CFX

- 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

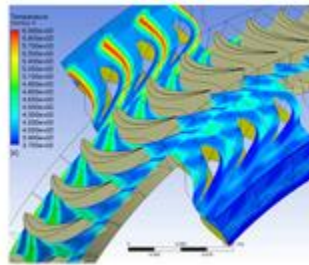


- Modeling of blade flutter:
 - Now also possible for entire wheels
 - Workflows for complex eigen modes available



- 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]

```

Completed Solution

Releasing connections from coupling participants... Done

Participant Build Information
-----
| Fluid Flow (Fluent)
| ANSYS Fluent 19.1.0, Build Year: Feb 19 2019 11:28:37, Build ID: 94_00
| Windows 64-bit
| WPP33 Steady-State Thermal
| Subprocess: ANS Solver 32 1085 102114014 317610000 610000 and
| Yastool
|
| ANSYS CFX solver Build: Tue Feb 20 01:14:37 2007 2018 Release:
| simple-64bit-6432-arch64-optimized-std-10mp

-----
| CONNECTION 0000 = 1
|
| Name          | Type
|-----|-----
| CONNECTION 0000 = 1
|
| MAPPING SUMMARY
|-----|-----|-----|
| Interface 1
| Mesh Name          | 100 | 100 |
| Region Element (E) | 100 | 97 |
| Region Area (A)    | 100 | 100 |
| Interface 1
| Region Element     | 100 | 100 |
| Region Area (A)    | 95 | 100 |
| Interface 2
| Mesh Name          | 100 | 100 |
| Region Area (A)    |
    
```

System Coupling Console [19.1]

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