## ANSYS Meshing & ICEM/CFD

- Meshing Overview
- ANSYS ICEM/CFD Meshing on RVS
- ANSYS Workbench Meshing on RVS

### Meshing Overview

The ANSYS software suite is providing a large variety of different meshing applications, methods and approaches. The following table is an attempt to provide to LRZ users a short outline of the available meshing capabilities:

<table>
<thead>
<tr>
<th>ANSYS Application</th>
<th>Meshing Methods</th>
<th>Suitable for...</th>
<th>Recommended Computer System for Meshing</th>
</tr>
</thead>
<tbody>
<tr>
<td>ANSYS ICEM/CFD</td>
<td>ICEM/CFD Hexa - structured hexahedral meshes</td>
<td>ANSYS CFX, ANSYS Fluent, ANSYS Mechanical</td>
<td>own laptop or workstation (non-LRZ)</td>
</tr>
<tr>
<td></td>
<td>ICEM/CFD Tetra - unstructured tet/prism meshes</td>
<td></td>
<td>LRZ Remote Visualization Systems</td>
</tr>
<tr>
<td>ANSYS Workbench</td>
<td>ANSYS Meshing (incomplete list):</td>
<td>ANSYS CFX</td>
<td>ANSYS Fluent</td>
</tr>
<tr>
<td>-----------------</td>
<td>---------------------------------</td>
<td>------------</td>
<td>--------------</td>
</tr>
<tr>
<td></td>
<td>• Patch Conforming Tetrahedron</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>• Patch Independent Tetrahedron</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>• Mesh Sweeping</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>• MultiZone Meshing</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>• Hex Dominant Meshing</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>• Assembly Meshing</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>• Tet / Prism meshes</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>• CutCell meshes (with hanging nodes)</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>ANSYS CFX</th>
<th>adaptive mesh refinement</th>
<th>ANSYS CFX</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>LRZ Cluster System, Adaptive mesh refinement is a solution-based mesh adaption, which is carried out during parallel solution run. Adaptive mesh refinement requires CFD setup on some initially provided mesh from other external source.</td>
<td></td>
</tr>
</tbody>
</table>
ANSYS Fluent
- adaptive mesh refinement
- tet/prism Fluent meshing
- polyhedral mesh conversion
- native polyhedral meshing (MOS AIC technology)

ANSYS Fluent
- LRZ Cluster System
  As for ANSYS CFX, adaptive mesh refinement is a solution-based mesh adaption, which is carried out during parallel solution run. Adaptive mesh refinement requires CFD setup on some initially provided mesh from other external source.
  ANSYS Fluent Meshing can be applied in a script-based manner, so that it can be used for script-driven, automated mesh generation during the start-up of a massively parallel ANSYS Fluent simulation. This is in particular useful for extremely large meshes for massively parallel computations, where it might be difficult to find a single suitable machine with large enough memory to create the mesh in advance prior to its usage in a parallel simulation run. ANSYS Fluent Meshing is based on so-called "water-tight" CAD geometries, where the geometry is then initially represented by a triangular tesselation.
- LRZ Remote Visualization Systems
  If the ANSYS Fluent Meshing is used in an interactive manner, than it is recommended to use the RVS. Nevertheless the meshing process allows the application of certain degree of parallelism.

ANSYS ICEM/CFD Meshing on RVS
Since meshing of a geometry is a memory intensive application, it is highly recommended to use the LRZ Remote Visualization Systems for running ANSYS ICEM/CFD. Furthermore, most meshing applications require hardware (OpenGL) graphics support for a reasonable user experience with the software.

Once you are logged into one of these LRZ Remote Visualization Systems, you can check the availability (i.e. installation) of ANSYS ICEM/CFD software by:

```
> module avail icem
```

Load the preferred ANSYS version environment module, e.g.:

```
> load module icem/2019.R3
```

Run ANSYS ICEM/CFD on the LRZ Remote Visualization System with OpenGL support:

```
> vglrun icemcfd
```

ANSYS Workbench Meshing on RVS
ANSYS Geometry Modeling (SpaceClaim and ANSYS DesignModeler) as well as ANSYS Meshing methods are available only through the platform of ANSYS Workbench. They cannot be run as standalone applications. So, once you are logged into one of these LRZ Remote Visualization Systems, you can check the availability (i.e. installation) of ANSYS Workbench software by:

```
> module avail wb
```

Load the preferred ANSYS Workbench version environment module, e.g.:

```
module load wb/2019.R3
```

Run ANSYS Workbench on the LRZ Remote Visualization System with OpenGL support:

```
> vglrun runwb2
```

From within the ANSYS Workbench schematic launch the ANSYS Meshing application with appropriate geometry import (CAD file, ANSYS DesignModeler or SpaceClaim geometry).