What is the ANSYS Software Package?

The ANSYS Software Package is an engineering and scientific software suite applicable for the simulation of computational fluid dynamics (CFD), acoustics, computational structural mechanics (CSM), electromagnetics (EMAG), thermodynamics and many more applications. The different ANSYS simulation tools and applications can be accessed either via the ANSYS Workbench GUI, the GUI's of standalone applications like ANSYS CFX or ANSYS Fluent and through command line interfaces, e.g. in so-called batch mode on the Linux Cluster and SuperMUC-NG. Furthermore the ANSYS Software Package includes the Postprocessing Tools CFD-Post and ANSYS Ensight.

The ANSYS Applications

- ANSYS CFX (Fluid Dynamics)
- ANSYS Fluent (Fluid Dynamics)
- ANSYS Mechanical (CSM)
- ANSYS Meshing & ICEM/CFD
- ANSYS Postprocessing

Licensing

The Leibniz Supercomputing Center (LRZ) provides access to ANSYS Software which is limited by the number of concurrently available software licenses as well as by the terms and conditions for the usage of the ANSYS Academic Research Licenses. These licenses may only be used for research and teaching by the institutes of the Bavarian Academy of Sciences and the Munich Universities. The licenses allow only the use of ANSYS products for non-commercial purposes.

At this point students are advised to use the freely available ANSYS Student Software edition wherever applicable. This free student software product can be downloaded from the ANSYS website and provides access to most of the ANSYS software products with some limitations on node/element/cell counts in the simulation models.
For academic research purposes there are only a limited number of free floating licenses available at LRZ. Further information on licensing at LRZ is available [here](#). Additional licenses purchased and owned by other institutions or university departments and which are aimed for use at the LRZ supercomputers can be hosted at LRZ.

For local installation of the software please consult our [software download page](#). To access this site you need a password that you will receive by contacting the LRZ user support. Furthermore, the most contemporary version of the ANSYS software (Windows, Linux) can be downloaded directly from the ANSYS Customer Portal after registering with ANSYS, Inc. as a software user.

**License Preference Settings**

ANSYS allows users to make there license preference settings prior to start any of the ANSYS applications by using the following ANSYS license administration GUI, e.g. on a Linux Cluster login node:

```bash
> module load ansys
> anslic_admin
```

in order to adjust their license preference settings. In the GUI under the menu option "Set License Preferences for User XXXXXX" is an overview provided over the ANSYS versions and available licenses. You can set your preferred usage profile there.

**Getting Started**

LRZ users are advised **NOT** to use Linux Cluster or SuperMUC-NG login nodes for any kind of ANSYS simulations which have the potential to put some heavy processor load on these login node systems or which are potentially consuming large amounts of memory in order not to disturb other cluster users. For such purposes large memory nodes are provided e.g. in interactive cluster queues. It is even more recommended to use for e.g. ANSYS meshing, pre- and postprocessing purposes the LRZ Remote Visualization Systems.

Once you are logged into one of these systems, you can check the availability (i.e. installation) of ANSYS software by:

```bash
> module avail ansys
```

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

```bash
> module load ansys/19.2
```

Also this is not recommended, the ANSYS Workbench can be run interactively on a login node (Linux: SSH option "-Y" or X11-forwarding; Windows: using PuTTY and XMing for X11-forwarding) via:

```bash
> runwb2
```

or on the LRZ Remote Visualization System via:

```bash
> vglrun runwb2
```

For parallel execution of the ANSYS individual software applications (ANSYS CFX, ANSYS Fluent, ANSYS Mechanical, ANSYS Maxwell, HFSS, LS Dyna, etc.) in the LRZ Linux Cluster and SuperMUC-NG batch queuing systems (SLURM, LoadLeveler) please refer to the provided application-specific subpages of this documentation.

Since ANSYS Workbench unfortunately does not support SLURM as a scheduler or batch queuing system, it is currently not possible at the compute resources of LRZ to execute ANSYS Workbench projects in distributed parallel mode. This would require an ANSYS Remote Solver Manager (RSM) instance being able to access the SLURM queues, which is currently not a functionality supported by the ANSYS software. The same applies for the currently not possible parametric studies in distributed parallel mode, which makes use of the parameter manager and RSM as well. Nevertheless those simulations can be started from within ANSYS Workbench, if the workbench is executed on a single node in interactive queue by using local parallel mode on the CPU cores of that single node.

**Documentation**

The ANSYS documentation can be accessed from within ANSYS Workbench or from within each individual ANSYS application through the provided help menu. Furthermore ANSYS documentation is provided for registered users through the ANSYS Customer Portal.
On LRZ systems with available ANSYS installation the online documentation can further be accessed by:

```bash
> module load ansys/19.2
> anshelp
```

### Mailing List for ANSYS Users

Users of the LRZ ANSYS licenses may join the mailinglist `ansys@lists.lrz.de`. This list will provide the user with important information about ANSYS software installation at LRZ (e.g. announcement of new versions, system configuration changes or important updates).

More information about ANSYS mailinglist at LRZ may be found [here](#).

### User Support

In case of any observed issues in the usage of the ANSYS software on LRZ managed compute resources or any arising questions, please feel free to contact the LRZ support.