QFINSOFT

ANSYS STUDENT

Frequently Asked Questions

COPYRIGHT

Qfinsoft (Pty) Ltd was founded in 1999 and now nearly twenty years later, Qfinsoft represents the combined expertise of engineers with years of practical and consulting experience in computer-aided engineering in one exceptional company.

Qfinsoft has partnered with world-leading CAE companies in order to serve as their Southern African representative. These partnerships allow us to distribute, train and support local users of the software products.

We have established many relationships through training, engineering consulting, and technical and software solutions. With our strong academic presence and involvement in engineering faculties across the country, Qfinsoft has helped to transfer industry knowledge to students, lecturers and prospective engineers through training and technical support.

Software that we re-sell and support:

EXPERTS IN COMPUTER AIDED ENGINEERING

The following link can be used to engage in the ANSYS student community and find answers to questions that other students have asked:

https://www.ansys.com/en-gb/academic/free-student-products/support-resources

Alternatively, please make use of the "ANSYS Help" that the software provides once it is installed – there are many useful examples and the information given is extensive.

Should you require a download link for the ANSYS Student version:
https://www.ansys.com/en-gb/academic/free-student-products

Please also find the Student FAQ from ANSYS attached.

EXPERTS IN COMPUTER AIDED ENGINEERING

There are three different downloads available for students, namely [1]:

1.1 Discovery Live Student

With ANSYS Discovery Live Student you can now conduct simple simulations without having to understand the complexity behind the simulation. It makes use of a brand-new instantaneous simulation technology and works on the basis of integrated geometry modeling based on the already existing ANSYS SpaceClaim technology. The integrated solvers create an environment where a simulation can be solved instantaneously for thermal, structural and fluid simulations.

1.1.1 System and Browser Requirements

Supported Platforms and Operating Systems:

- Microsoft Windows 7, 64-bit (Professional and Enterprise)
- Microsoft Windows 8, 64-bit (Professional and Enterprise)
- Microsoft Windows 10, 64-bit (Professional, Enterprise and Education), supported builds: LTSB and CBB (CB not supported)

Minimum Hardware Requirements:

- CPU: Pentium® 4 2.0 GHZ or Athlon® 2000+ or faster, 64-bit (x64) processor
- 8 GB RAM
- 4 GB hard drive space
- ANSYS Discovery Live Student relies on the latest GPU technology to provide it's computation and visual experience. To run the software, you will require:
  - Graphics Card: A dedicated NVIDIA GPU card (Quadro recommended) based on the Kepler, Maxwell or Pascal architecture. Maxwell 2000 or better strongly recommended. Most dedicated NVIDIA GPU cards produced in 2013 or later will be based on one of these architectures.
  - At least 4GB of video RAM (8GB preferred) on the GPU. Please ensure you have the latest driver for your graphics card, available from NVIDIA Driver Downloads. The Discovery Live Compatibility Utility can be used to determine if your current graphics hardware is capable of supporting ANSYS Discovery Live. Performance of Discovery Live is less dependent on machine CPU and RAM. A recent generation 64-bit CPU running Windows, and at least 4GB of

RAM will be sufficient. If you do not have a graphics card that meets these specifications, the software will not run.

1.2 ANSYS AIM Student

ANSYS AIM is suggested if you are unfamiliar with simulation tools and need an intuitive, cutting-edge tool to provide you with the solutions required.

1.2.1 System and Browser Requirements

Supported Platforms and Operating Systems:

- Microsoft Windows 7, 64-bit
- Microsoft Windows 8, 64-bit
- Microsoft Windows 10, 64-bit

Minimum Hardware Requirements:

- Processor(s): Workstation class
- 4 GB RAM
- 25 GB hard drive space
- OpenGL-capable graphics card

1.2.2 Problem Size Limits

Structural Physics: 100K nodes/elements

Fluid Physics: 1 Million cells

Electromagnetics: 50 000 elements

1.3 ANSYS Student

ANSYS Student is a product that is a workbench-based bundle of ANSYS Mechanical, ANSYS CFD, ANSYS Autodyn, ANSYS SpaceClaim and ANSYS DesignXplorer.

1.3.1 System and Browser Requirements

Supported Platforms and Operating Systems:

- Microsoft Windows 7, 64-bit
- Microsoft Windows 8, 64-bit
- Microsoft Windows 10, 64-bit

Minimum Hardware Requirements:

- Processor(s): Workstation class
- 4 GB RAM
- 25 GB hard drive space
- Computer must have a physical C:\" drive present
- Graphics card and driver: Professional workstation class 3-D
- OpenGL-capable

1.3.2 Problem Size Limits

Structural Physics: 32K nodes/elements

Fluid Physics: 512K cells/nodes

Electromagnetics: N/A

1.4 SpaceClaim Limitations

SpaceClaim has a limitation of max 50 bodies and max 300 faces once imported into Workbench. You will receive the following error if this limitation should be exceeded:

"*You have exceeded the allowable geometry limits for ANSYS Academic. You cannot save or export. See Help for more information.*"

Computational Fluid Dynamics – ANSYS Fluent

"My simulation does not converge"

Before you reach out to Qfinsoft for support on this question, please check the following options that could be the reason why your simulation is not converging:

- Meshing
    - o It is always sensible having a proper mesh on the model. A proper mesh means:
        - All parts of the geometry must be meshed (you cannot solve a model if the mesh has failed on any given part of the geometry)
        - The minimum orthogonal quality is greater than 0.1
        - The mesh is not too course (sometimes models will struggle to converge because the mesh is so coarse that it can not capture all the physics)
        - The mesh is not too fine/dense (try to make use of a Mesh Dependency Study). A very dense mesh will cause large computational time. Therefore, it is advisable to always ensure that you use the most optimal size/mesh settings for the model in order to still obtain accurate results.
        - Are there small edges and vertices present? This can cause a bad quality mesh (and as a result non-convergence). Small edges and vertices can occur during Multi-Body Part operations and bad geometry setup. It is easiest to either set the Mesh Defeaturing tool to an accurate size that will defeature the small edges away or cleaning the geometry prior to meshing.

- Were the boundary conditions specified in a correct manner?
  - Boundary Conditions include:
    - Pressure, outflows, velocities, mass flow rates, turbulence, walls, temperatures and volume fractions (multiphase) to names a few.
  - Is the magnitude of the boundary condition sensible? Sometimes simulations will struggle to converge when the boundary values are too large.
  - The stability of convergence can be dependent on combinations of boundary conditions used. The following combinations are listed from the most robust option to the least robust:
    - Velocity/mass flow at the inlet and static pressure at the outlet
    - Total pressure at the inlet and velocity/mass flow at the outlet
    - Total pressure at the inlet and static pressure at the outlet (sensitive to initial conditions.
  - Certain boundary conditions are not compatible and will cause stability issues. The following boundary conditions are not compatible:
    - Inlet velocity for compressible flow
    - Total pressure at the inlet and outflow at the outlet.
    - Outflow boundary when using ideal gas law for density
- Solver Settings
  - Playing around with the solver settings can sometimes help convergence. Depending on the type of simulation and the complexity of the model/simulation, the default solver settings are not always optimal. Sometimes it helps to:
    - Compute the initial solution using first order discretization schemes and later use 2nd order once the solution has stabilized (some default settings are 2nd order which is more difficult to converge).
    - Reduce the under-relaxation factors (pressure-based solver) or Courant number (density-based solver)

- When performing a transient simulation, reduce the time step or increase the number of iterations per time step.

Finite Element Analysis – ANSYS Mechanical

"My simulation does not solve"

Before you reach out to Qfinsoft for support on this question, please check the following options that could be the reason why your simulation is not solving:

- Geometry
  - o Is the CAD "clean" and simple? A clean CAD/geometry for simulation purposes comprises of the following:
    - All welding rounds and gaps have been removed (welding will be accounted for by bonded contacts in ANSYS Mechanical)
    - All fillets and unnecessary holes have been removed.
    - Small edges and slivers have been removed (or accounted for in meshing by the defeature tool (refer to the meshing section)
    - If a lot of thin, parallel plates are present in the geometry it is better and easier for meshing & solving purposes to midsurface these (creating a shell geometry)

- Meshing
  - o It is always sensible having a proper mesh on the model. A proper mesh comprises of:
    - All parts of the geometry must be meshed (you cannot solve a model if the mesh has failed on any given part of the geometry)
    - The minimum orthogonal quality is greater than 0.1
    - The mesh is not too course (sometimes models will struggle to converge because the mesh is so coarse that it can not capture all the data)
    - The mesh is not too fine/dense (try to make use of a Mesh Dependency Study). A very dense mesh will cause large computational time.

Therefore, it is advisable to always ensure that you use the most optimal size/mesh settings for the model in order to still obtain accurate results.

- Are there small edges and vertices present? This can cause a bad quality mesh (and as a result non-convergence). Small edges and vertices can occur during Multi-Body Part operations and bad geometry setup. It is easiest to either set the Mesh Defeaturing tool to an accurate size that will defeature the small edges away or cleaning the geometry prior to meshing.

- <u>Were the boundary conditions specified in a correct manner?</u>
  - o Boundary Conditions include:
    - Any kind of support (fixed, remote displacement, cylindrical support etc…)
    - Any kind of pressure, load, temperature
    - Acceleration, velocity, gravity
  - o Is the magnitude of the load/pressure/temperature sensible? Sometimes simulations will struggle to converge when the load applied is too large.

- <u>Contacts</u>
  - o If automatic contact detection is used, please ensure that all contacts are defined properly. Take out contacts that are not necessary and add contacts where necessary. The model will solve incorrectly if the contact settings are applied wrong. When setting up the contacts, remember that the contacts that ANSYS makes use of are defined as follows:

Contact Type	Iterations	Normal Beahviour (Seperation)	Tangential Behaviour (Sliding)
Bonded	1	No Gaps	No Sliding
No Seperation	1	No Gaps	Sliding Allowed
Frictionless	Multiple	Gaps Allowed	Sliding Allowed
Rough	Multiple	Gaps Allowed	No Sliding
Frictional	Multiple	Gaps Allowed	Sliding Allowed

EXPERTS IN COMPUTER AIDED ENGINEERING

- Bear in mind that frictionless, rough and frictional contacts imply that the simulation will result in a non-linear simulation due to the contact type used.

Meshing – ANSYS Mechanical

If you get mesh related errors, your model does not produce expected results or your model takes a ridiculously long time to solve, please consider the following hints:

- First and foremost, check the mesh quality. The minimum orthogonal quality should be ≥ 0.1.
- A mesh that is too coarse will not produce accurate results. Ensure that the mesh is fine enough to capture all the relevant data.
    - If a geometry with a high amount of bodies is present with many areas that are of no concern, consider using a sub-model to only mesh areas of interest very finely (reduces solving time and increases accuracy in the specific areas).
- A very dense (fine) mesh can result in high computational time. Consider a mesh dependency study in order to ensure that the correct mesh sizing is used to still produce accurate results.
- Meshing is simplified if shell models are used (refer to the FEA section on geometry)
- Check defeaturing settings. If important parts of the geometry are defeatured away and thus not capturing all the data at the location of interest your defeaturing value might be too high (the default setting should however not cause any issues).
- If bodies are too complex or not connected properly (large gaps in between "connected bodies") the mesh might fail. Consider splitting/cleaning the geometry in such a way as to simplify the model for meshing purposes (pull edges that are too far apart closer together, use the splitting tool to split up a body into several parts etc).
- Is your mesh continuous across the geometry? If not, you might want to consider using the shared topology option or defining contacts between the bodies of interest.

Shared Topology – ANSYS Mechanical & ANSYS Fluent

Shared topology can be defined in SpaceClaim and/or Design Modeller. Shared topology allows for a continuous mesh across common faces/regions where bodies are in contact. This is specifically helpful when dealing with multi-body geometries which consist of a large amount of bodies. Once shared topology is activated for the bodies of interest, a contact region no longer has to be defined in ANSYS Mechanical for that set of bodies that were shared.

Shared Topology is also required for a CFD analysis in order to ensure mass and heat transfer between volumes.

Before the geometry is imported into Workbench, you should decide which parts should be in shared topology and group them accordingly. Remember to define contacts on the parts that are not in shared topology, once imported into Workbench.

Additional Resources

Should you require specific training material, please feel free to send us a request as follows:

E-Mail: support@qfinsoft.co.za

Subject: Student Request: Training Material [*fill in the type of training that you require, for instance "FEM - Linear", "CFD-Fluent", "Meshing"*]

Please also find the Student FAQ from ANSYS attached.

REFERENCES

[1] ANSYS (2018). *Free Student Software, ANSYS Student*. [online]. Available at:

https://www.ansys.com/academic/free-student-products [Accessed 2 August 2018]