

ANSYS Airpak Technical Specifications

ANSYS® Airpak® software is a quick and easy-to-use design tool that simplifies the application of state-of-the-art airflow modeling technology for the design and analysis of ventilation systems used to deliver indoor air quality (IAQ), thermal comfort, health and safety, air conditioning and contamination control solutions. The ability to rapidly create and automatically mesh ventilated spaces is coupled with the fast, accurate and well-proven unstructured solver engine of ANSYS® FLUENT® software. Additionally, post-processing features essential to the heating, ventilating and air conditioning (HVAC) industry allow ANSYS Airpak software users to effectively visualize ventilation system design performance and communicate analysis results to customers, internal team members, government agencies and prospective clients. ANSYS Airpak technology enables users to complete work in the shortest amount of time, compared to other airflow modeling software tools.

Model Building Features

- Mouse-driven selection, placement and sizing of pre-defined objects for use in model building; fast and easy, even for complex geometries
- Object-based model building with predefined objects including rooms, people, blocks, fans, partitions, vents, openings, sources, walls, ducts, resistances, heat exchangers and hoods
- Comprehensive object shapes including rectangular blocks, cylinders, ellipsoids, elliptical cylinders, concentric cylinders, prisms of polygonal cross section, prisms of varying cross section and ducts of arbitrary cross section
- Rectangular or circular 3-D/2-D fans with hubs, guards and power specifications; ability to dial-in operating and nominal RPM and fan curve changes automatically
- Polygonal and circular shapes allowed for fans, vents, resistances, partitions and openings
- Library functions to store or retrieve groups of objects in a parts library
- Inclined rectangular partitions, fans, vents, openings and resistances
- ModelManager: a graphical Explorer tree-style model management tool to quickly create, edit and manage objects, assemblies, libraries, projects, configurations and settings

- Mouse-driven interactive GUI controls including:
 - Mouse or keyboard control for placing, moving and sizing objects
 - Ruler option to guide in sizing/resizing objects
 - 3-D mouse-based view manipulation with dynamic rotation, translation and zoom
 - Undo/redo functions
 - Active/inactive option for all objects
 - Error and tolerance checking
 - Flexible/customizable units for all input fields
 - ModelManager entities
 - Graphical alignment tools
- Geometry inputs using local coordinates
- Geometry import using IGES, DXF or DWG file formats
- Capability to import and export model geometry information to spreadsheets
- Comprehensive model summaries in HTML format
- Comprehensive material property database
- Capability to set up user-defined macros

CFD Pre-Processing in CFX-Pre

The ANSYS® CFX® physics pre-processor is a modern and intuitive interface for the setup of CFD analyses. In addition to a general mode of operation, predefined wizards are available to guide users through the setup of common fluid flow simulations. A powerful expression language gives users the ability to customize their problem definition in numerous ways, such as with complex boundary conditions, proprietary material models or additional transport equations. The adaptive architecture of CFX-Pre even allows users to create their own custom GUI panels to standardize input for selected applications, and thereby ensure adherence to established best practices

Meshing Features

- A full range of meshing options:
 - Automatic unstructured body conforming
 - Automatic structured Cartesian meshing for convenient analysis of models with predominantly rectangular geometric shapes
 - Automatic nonconformal (for unstructured and structured) to mesh regions of the model separately, simplify meshing and reduce cell count, thereby increasing speed of solutions