International Refereed Journal of Engineering and Science (IRJES)
ISSN (Online) 2319-183X, (Print) 2319-1821
Volume 13, Issue 5 (Sep. – Oct. 2024), PP. 156-158

Workflow Analysis of Ansys CFX in Ansys Workbench

Krupamayee Mahapatra[1], Bharati Nayak[2], Dillip Kumar Lenka[3]
**Department of Computer Science & Engineering, Swami Vivekananda School Of Engineering and Technology, BBSR, ODISHA, INDIA, PIN-752054*
***Department of Computer Science & Engineering, Swami Vivekananda School Of Engineering and Technology, BBSR, ODISHA, INDIA, PIN-752054*
**** Department of Computer Science & Engineering, Swami Vivekananda School Of Engineering and Technology, BBSR, ODISHA, INDIA, PIN-752054*

ABSTRACT:
*Ansys Inc. is an American public company based in Canonsburg, Pennsylvania. It develops most Ansys simulations are performed using the **Ansys Workbench** software, which is one of the company's main products. The **ANSYS Workbench** platform is the backbone for delivering a comprehensive and integrated simulation system. Using Workbench for your product development sim ulations will result in higher productivity from integrated applications and access to multiphysics and systems level capabilities. The platform is based around an innovative project schematic view which ties together the entire simulation process.*
***Keywords**: ANSYS Workbench, CFX, CFD*

I. INTRODUCTION

The ANSYS Workbench environment is an intuitive up-front finite element analysis tool that is used in conjunction with CAD systems and Design Modeler [7]. ANSYS Workbench is a software environment for performing structural, thermal, and electromagnetic analyses. The class focuses on geometry creation and optimization, attaching existing geometry, setting up the finite element model, solving, and reviewing results. The class will describe how to use the code as well as basic finite element simulation concepts and results interpretation. The ANSYS Workbench represents more than a general purpose engineering tool. It provides a highly integrated engineering simulation platform. Supports multi physics engineering solutions. Provides bidirectional parametric associativity with most available CAD systems[6]. Also provides access to a range of ANSYS Engineering Simulation solutions with designed to handle a limited set of relatively simple engineering solutions.

II. ANSYS Work-Bench Platform

Engineering Simulation Platform

Today's businesses operate in a highly competitive environment where every company strives to innovate faster while reducing their product development and maintenance costs. Every function within an organization faces tremendous pressure to deliver more with less. Engineering operations are no exception. That's why the world's leading companies have adopted the ANSYS simulation platform to transform the way they innovate and engineer their next-generation products and maintenance services.

The ANSYS simulation platform delivers the broadest suite of best-in-class simulation technology and unifies it with your custom applications, CAD software and enterprise business process tools such as PLM [7]. Its open and flexible framework connects engineering teams, tools and data. A common platform facilitates the efficient and reliable sharing of engineering information across an organization, its supply chain and field operations. making your engineering operations more agile.

The platform is built on three pillars that enable you to apply simulation effectively from concept to detailed design, resulting in productivity gains, faster design decisions and accelerated innovation.

Comprehensive simulation

With the ANSYS simulation platform, you can produce high-fidelity virtual prototypes to simulate the behavior of complete products in their actual working environments[8]. It delivers the most advanced technologies for 3D physics simulation, embedded software design and system simulation.

Salable solutions

The platform adapts to any IT infrastructure from desktop to cloud, enabling you to collaborate and share simulation data and processes across multiple sites.

Extensible ecosystem
With the ANSYS simulation platform's open framework, you can create custom workflows or download ready-to-use apps from the ANSYS App store. The platform gives you access to our thriving partner network of industry-leading companies, which ensures easy integration and management of additional capabilities for your challenging and evolving needs

III. ANSYS CFX/CFD

ANSYS CFX is a high performance, general purpose CFD program that has been applied to solve wide-ranging fluid flow problems for over 20 years. At the heart of ANSYS CFX is its advanced solver technology, the key to achieving reliable and accurate solutions quickly and robustly. ANSYS Computational Fluid Dynamics (CFD)[3] simulation software allows to predict, with confidence, the impact of fluid flows on your product throughout design and manufacturing as well as during end use. The software's unparalleled fluid flow analysis capabilities can be used to design and optimize new equipment and to troubleshoot already existing installations. Whatever phenomena you are studying single or multi-phase, isotherm or reacting, compressible or not ANSYS fluid dynamics solutions give you valuable insight into your product's performance. ANSYS renowned CFD analysis tools include the widely used and well-validated ANSYS Fluent and ANSYS CFX[4], available separately or together in the ANSYS CFD bundle. Because of solver robustness and speed, development team knowledge and experience, and advanced modeling capabilities,

The academicians and researchers are utilizing CFD to investigate and simulate the combination of fluid flow and chemical processes. From equipment and processes to chemical and petrochemical refining to glass and metals manufacturing. ANSYS simulation provides solutions for the chemical & process industries to reduce overall costs, conserve energy, minimize environmental impacts, meet higher regulatory standards, and streamline product development and operations through a range of initiatives.

IV. ANSYS Fluent

Fluent software contains the broad, physical modeling capabilities needed to model flow, turbulence, heat transfer and reactions for industrial applications. These range from air flow over an aircraft wing to combustion in a furnace, from bubble columns to oil platforms, from blood flow to semiconductor manufacturing and from clean room design to wastewater treatment plants. Fluent spans an expansive range, including special models, with capabilities to model in-cylinder combustion, aero-acoustics, turbomachinery and multiphase systems.

Fluent also offers highly scalable, high-performance computing (HPC) to help solve complex, large-model computational fluid dynamics (CFD) simulations quickly and cost-effectively [1]. Fluent set a world supercomputing record by scaling to 172,000 cores.

New Fluent Experience: A whole new way to enjoy CFD simulations.
Fluent already solves the toughest design challenges with well-validated results across the widest range of CFD and multiphysics applications. But today, engineers need to accomplish more, in less time and with less training than ever before.
Introducing the New Fluent Experience, a whole new way to enjoy CFD simulations.
Built on top of the proven Fluent solver, this new experience:
. Provides a complete, single-window solution within Fluent.
. Streamlines the Fluent workflow for generating a mesh from imported CAD.
. Removes barriers for common tasks that frustrate users.

Common Workflow

The ANSYS CFD product provides access to both ANSYS FLUENT and ANSYS CFX software. ANSYS Workbench technology, these two well-known products share a common workflow. CAD connectivity, geometry modeling, meshing, and post-processing are shared by both. Furthermore, engineers can now use the familiar ANSYS Workbench applications to perform parametric analysis and design optimization.

Comprehensive Physics

The combination of ANSYS FLUENT and ANSYS CFX software provides users with an unprecedented level of fluid dynamies modeling capabilities. ANSYS CFD technology provides engineers with a wealth of physics models. This includes internal and external flow, single-phase and multiphase flow, chemical reaction, combustion, noise prediction, heat transfer, radiation, turbo machinery flows, and many more fluid flow related models.

Complex turbulent flow structures modeled with ANSYS CFD software

The extensive capabilities of ANSYS CFD software make it the simulation tool of choice for designers and analysts world-wide. With a large commercial and academic user base, along with a fluids simulation product line that is both broad and deep, ANSYS, Inc. is an industry leader and a technological champion for fluid dynamics simulation.

V. CONCLUSION

ANSYS software played a critical role in creation of rocket launch, flown on an airplane, driven a car, used a computer, touched a mobile device, crossed a bridge or put on wearable technology. ANSYS is the global leader in engineering simulation. Through our strategy of Pervasive Engineering Simulation, we help the world's most innovative companies deliver radically better products to their customers. By offering the best and broadest portfolio of engineering simulation software, we help them solve the most complex design challenges and create products limited only by imagination. At ANSYS, we are committed to fostering a culture of diversity and inclusion. We strive to create a workplace where people from distinct backgrounds can come together to support each other and solve our customers' problems. We respect, value, embrace and are strengthened by the unique perspectives that our employees bring to work each day to make us ONE ANSYS.

REFERENCES

[1]. Nuclear Systems Material Handbook. Vol. 1: Design Data, Part 1: Structural Materials, Group 1: High Alloy Steels. U. S. Department of Energy, Office of Scientific and Technical Information. Oak Ridge, TN.
[2]. Nuclear Systems Material Handbook. Vol. 1: Design Data, Part 1: Structural Materials, Group 2: Low Alloy Steels, Section 2-2 1/4 CR-1 Mo.. U. S. Department of Energy, Office of Scientific and Technical Information. Oak Ridge, TN.
[3]. F. Barlat and J. Lian. "Plastic Behavior and Stretchability of Sheet Metals. Part I: A Yield Function for Orthotropic Sheets Under Planc Stress Conditions". Int. Journal of Plasticity, 5. pg. 51-66.
[4]. F. Barlat, D. J. Lege, and J. C. Brem. "A Six-Component Yield Function for Anistropic Materials". Int. Journal of Plasticity, 7. pg. 693-712.
[5]. R. Hill. "A Theory of the Yielding and Plastic Flow of Anisotropic Metals". Proceedings of the Royal Society of London, Series A., Vol. 193. 1948. [
[6]. F. K. Chang and K. Y. Chang. "A Progressive Damage Model for Laminated Composites Containing Stress Concentration", Journal of Composite Materials, 21. pg. 834-855. 1987a.
[7]. R. G. Dean. Evaluation and Development of Water Wave Theories for Engineering Application. Volume 2, Tabulation of Dimensionless Stream Function Theory Variables, Special Report No. 1,. U. S. Army Corps of Engineers, Coastal Engineering Research Center. Fort Belvoir, VA. November 1974.
[8]. Michael E. McCormick. Ocean Engineering Wave Mechanics. Wiley & Sons. New York. 1973.