PROGRAM COMPLEX ANASYS

R. V. Shulga – *group KM-71*

Program Complex ANSYS includes four main softwares. They are: 1) Fluid Dynamics; 2) Structural Mechanics; 3) Electromagnetic; 4) Systems and Multiphysics.

ANSYS Fluid Dynamics consists of ANSYS CFX and ANSYS Fluent. This software can be useful for:

1) Improving aerodynamics and ventilation in aircraft, cars and buildings; cuting energy costs and improving comfort and safety;
2) Designing more-efficient and longer-lasting turbines, from huge hydroturbines to turbochargers and heart pumps;
3) Creation better-functioning solutions in alternative energy such as wave power, wind turbines and fuel cells.

ANSYS structural mechanics software brings together the largest elements library with the most advanced structural simulation capabilities available. This unified engineering environment help you streamline processes to optimize product reliability, safety and functionality.

ANSYS Electromagnetics software can help you to predict the behavior of complex electrical and electromechanical systems — from mobile communication and internet devices to automotive components and electronics equipment. This software can be useful for improving equipment performance of smartphones, satellites, batteries and hybrid vehicles.

ANSYS Systems &Multiphysics can be useful for accurate tracking of the interactive effects of components and detailing how they will perform as a whole; modeling scalability for evaluating entire systems that include any combination of high-fidelity 3-D and reduced-order models and others.

Now to describe ANSYS software because I used it in my bachelor work. ANSYS CFX software is fully integrated into the ANSYS® Workbench™ environment, the framework for the full suite of engineering simulation solutions from ANSYS. Its adaptive architecture enables users to easily set up anything from standard fluid flow analyses to complex interacting systems with simple drag-and-drop operations. If you want to solve complicated problems in ANSYS CFX, you need to create geometry of the body and mesh of the body. The next step is to set up entry and boundary conditions in ANSYS Pre-processor; then you need to choose the way your computer will solve the problem (or the task) in ANSYS CFX Solver and after that you can look through the results.

D. O. Marchenko – *EL adviser*