STRENGTHS, CAPABILITIES AND LIMITATIONS OF USING CFD AND REVIEW SOFTWARE SOLUTIONS FOR NUMERICAL MODELING

M.Sc Aleksandar Pavlovic *, M.Sc Aleksandar Djuric, M.Sc Marko Simic, PhD Bogdana Vujic
University of Novi Sad, Technical faculty “Mihajlo Pupin”, Zrenjanin

Workshop ENVIROBANAT
5-6. September 2013, Zrenjanin

• Computational fluid dynamics or CFD is the analysis of systems involving fluid flow, heat transfer and associated phenomena such as chemical reactions by means of computer-based simulation. The technique is very powerful and spans a wide range of industrial and non-industrial application areas.

• CFD codes are structured around the numerical algorithms that can tackle fluid flow problems. In order to provide easy access to their solving power all commercial CFD packages include sophisticated user interfaces to input problem parameters and to examine the results.

•All codes contain three main elements:

1.pre-processor,
2.solver and
3.post-processor.

GENERAL STEPS OF CONDUCTING A NUMERICAL SIMULATION

- Choice of target variables
- Choice of approximate equations describing the physics of the flow
- Choice of geometrical representation of the obstacles
- Choice of computational domain
- Choice of boundary conditions
- Choice of initial conditions
- Choice of computational grid
- Choice of time step size
- Choice of numerical approximations
- Choice of iterative convergence criteria

CROSS BORDER COOPERATION Romania-Serbia Common borders. Common solutions.

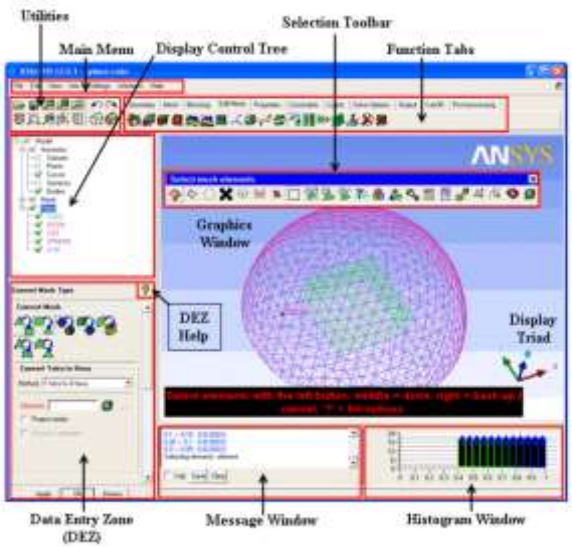
THE ADVANTAGE OF USING CFD SIMULATION

- **Problem solving and visualization.**
- **Reducing the cost of the basic model.**
- **Repeatability**
- **Improvement.**

PRACTICAL EXAMPLES CFD MODEL

ANSYS

- ANSYS is a general purpose software, used to simulate interactions of all disciplines of physics, structural, vibration, fluid dynamics, heat transfer and electromagnetic for engineers
- The ANSYS fluid dynamics solution is a comprehensive suite of products that allows you 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 fluid flow phenomena you are studying – single- or multi-phase, isothermal or reacting, compressible or not – ANSYS fluid dynamics solutions give you valuable insight into your product's performance.



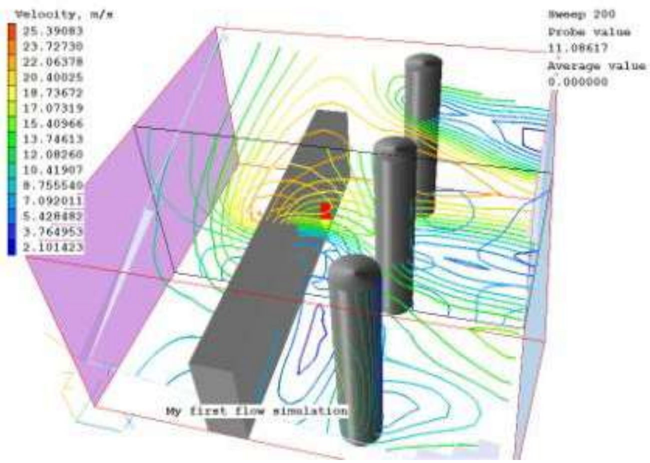
Ansys ICEM GUI Components

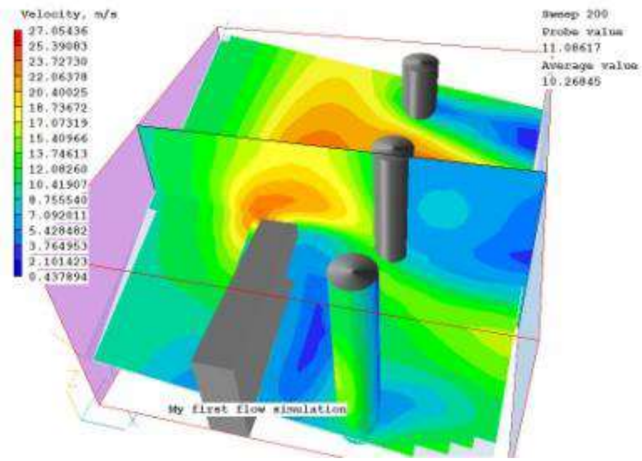
PRACTICAL EXAMPLES CFD MODEL

PHOENICS

• PHOENICS is a general-purpose Computational Fluid Dynamics code. It was launched commercially in 1981 by Concentration Heat and Momentum Limited in Wimbledon, London, UK as the world's first general-purpose CFD code. The name PHOENICS itself is an acronym for Parabolic, Hyperbolic Or Elliptic Numerical Integration Code Series.

• PHOENICS is a general-purpose commercial CFD code which is applicable to steady or unsteady, one- , two- or three-dimensional turbulent or laminar, multi-phase, compressible or incompressible flows using Cartesian, cylindrical-polar or curvilinear coordinates.





View PHOENICS with fill (left) and lines (right)

PROBLEM SOLVING WITH CFD

- In solving fluid flow problems we need to be aware that the underlying physics is complex and the results generated by a CFD code are at best as good as the physics (and chemistry) embedded in it and at worst as good as its operator.

- Typical decisions that might be needed are whether to model a problem in two or three dimensions, to exclude the effects of ambient temperature or pressure variations on the density of an air flow, to choose to solve the turbulent flow equations or to neglect the effects of small air bubbles dissolved in tap water.

- Every numerical algorithm has its own characteristic error patterns. Well known CFD euphemisms for the word ‘error’ are terms such as numerical diffusion, false diffusion or even numerical flow. The likely error patterns can only be guessed on the basis of a thorough knowledge of the algorithms. At the end of a simulation the user must make a judgment whether the results are ‘good enough’.

- Anyone wishing to use CFD in a serious way must realize that it is no substitute for experimentation, but a very powerful additional problem solving tool.

CROSS BORDER COOPERATION
Romania-Serbia
Common borders. Common solutions.

CONCLUSION

- The use of numerical modeling software solutions or CFD has almost unlimited possibilities for the design and testing of certain systems.

- The complexity of the model is determined by the complexity of the system being designed.

- During the design and testing of the system may be to reduce the costs that would be incurred if they were producing prototypes and they solved certain disadvantages. This paper presents two leading software currently in the world, which differ in small details and terms and usually support themselves and improve existing software applications

CROSS BORDER COOPERATION
Romania-Serbia
Common borders. Common solutions.

Thank you for your attention