

Customization tools and programming in FLUENT

Dr. Gergely Kristóf
Department of Fluid Mechanics, BME
16-th November 2011

Journaling

- Use text menu commands
- Eg.
su rs x05mm .005 0 .005 .003 20
- Commands can be executed in runtime:
Calculation Activities / Execute Command
- Model settings and calculation activities can be stored in one single journal file, and then be executed on other meshes.
- FLUENT can run in batch mode: executes a single journal file and exits. Text output can be written into a log file.
- More elaborated journal programs can be written in scheme programming language.

Defining point profiles

An easy way without any programming:

- Write point profile from the given boundary zone
- Manipulate the profile in Excel
- Read profile file

User Defined Scalars (UDS)

- Additional transport equations can be solved for UDS-s.
- Physical content of the UDS can be specified through source terms, boundary conditions and the diffusion coefficient of the defining transport equation.
- Eg: Define UDS for calculating the residence time of the fluid!

Customized field functions

- You can define more complex functions of existing field quantities.
- Can be used only for initialization (patching), monitoring, and for post processing.

User Defined Functions (UDF)

- You can access and manipulate almost every solver data and fully control the solution process in UDF-s written in C programming language.
- Some application examples:
 - Defining profiles that depend on any field variables;
 - Defining source terms that describe complex physics;
 - Defining complex material properties such as thixotropy;
 - Defining complex mesh motion;
 - Coupling a CFD model with another CFD model (nested models) or with different simulations (such as solid mechanics);
 - Doing advanced post-processing.
- [ANSYS presentation on UDF-s](#)