# CFD aerodinamika

UVOD V OPENFOAM IN PARAVIEW

- ▶ Zbirka programov za CFD analize raznih tokov
- Za Windows:
  - https://bluecfd.github.io/Core/Downloads/

- Za Ubuntu:
  - https://openfoam.org/download/7-ubuntu/



- blueCFD-Core 2017-2 blueCFD-Core 2017-1
- blueCFD-Core 2016-2
- blueCFD-Core 2016-1
- blueCFD-Core 2.3-1
- Download counters

#### blueCFD-Core 2017-2

vailable installers for Windows 7 through 10, all 64-bit:

- blueCFD-Core-2017-2-win64-setup.exe
  - FOAM 5.x (19576d14), ParaView 5.4.1 and MS-MPI 7.1 (MS-MPI 8.1 is optional).
  - CLI and development stack included with MSys2. See release notes for more details.
- o 790 MiB installer, SHA1: 4df0b7a38eea5363b66e5f7d0d94f50714d59f1a

- Instructions on how to install and build from source code will be provided in the User Guides page.
- This installer can be used for updating an existing installation of blueCFD-Core 2017-1, since it will load the same installation settings.
- . blueCFD-Core 2017 is installed separately from previous blueCFD-Core versions and should not be used as a direct upgrade.

#### blueCFD-Core 2017-1

Available installers for Windows 7 through 10, all 64-bit:

- blueCFD-Core-2017-1-win64-setup.exe
  - blueCFD-Core 2017-1 provides OpenFOAM 5.x (5f49a693), ParaView 5.4.1 and MS-MPI 7.1 (MS-MPI 8.1 is optional).
    - CLI and development stack included with MSys2.
       See release notes for more details.

#### Izpis rezultatov

Nastavitve v beležkah v "**case**" mapi

OpenFOAM-ovi programi

- **0** začetne vrednosti in robni pogoji
- constant materialne lastnosti, mreža, težni pospešek
- system nastavitve simulacije, nastavitve mreženja, paralelnega procesiranja ...

 Urejamo npr. z notepad++:

https://notepad-plus-plus.org/downloads/v7.7.1/

- ▶ 1. vaja:
  - 1. Odpri blueCFD-Core 2017 terminal
  - Vnesi ukaz: \$ cd blueCFD/ofuser-of5/
  - 3. Vnesi ukaz: \$ git clone https://github.com/jpeter3/0Fkurz.git
  - 4. V terminalu se s pomočjo ukazov Is (pokaži možne direktorije) in d (change directory) premakni v mapo:

\$\rightarrow \cdot \blueCFD/ofuser-of5

Environment is now ready. Notes:

\$ cd blueCFD/ofuser-of5/

- You can change between installed versions by running: ofmenu - You can change to other predefined versions by running: ofmenuNew

jakap@DESKTOP-KKPTPA9 MINGW64 OpenFOAM-5.x ~/blueCFD/ofuser-of5

Addons blueCFD changedefmpi.bat defaultmpi.sh mountBlueCFD README.TXT sourceOF

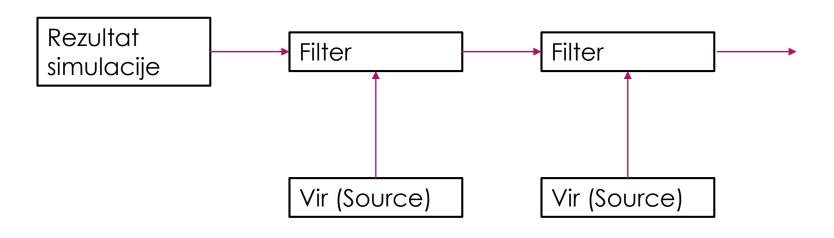
- ~\blueCFD\ofuser-of5\OFkurz\simulacije\_naloge\airFoil2D
- 5. V terminalu poženi program za izračun nestisljivega, časovno ustaljenega turbulentnega toka: \$ simpleFoam

(Dolar znak označuje ukaz in ga ni potrebno pisati v terminal)

Setting environment for OpenFOAM 5.x mingw-w64 Double Precision (of5-64), using MSMPI71 - please wait...

```
\( \sigma \) \( \sigma \) \( \lambda \) \( \lambda \) \( \sigma \) \( \lambda \) \(
smoothSolver: Solving for Ux, Initial residual = 4.92825e-006, Final residual = 1.3327e-007, No Iterations 4
smoothSolver: Solving for Uy, Initial residual = 1.52363e-006, Final residual = 1.13175e-007, No Iterations 4
GAMG: Solving for p. Initial residual = 8.46511e-006, Final residual = 6.70642e-007, No Iterations 5
time step continuity errors : sur
smoothsolver: solving for nutil Ostanek reda veliksoti le-6 je značilen za konvergirano rešitev
ExecutionTime = 10.998 s ClockTime = 11 s
Time = 350
smoothSolver: Solving for Ux, Initial residual = 4.79482e-006, Final residual = 4.77931e-007, No Iterations 2
smoothSolver: Solving for Uy, Initial residual = 1.49668e-006, Final residual = 1.11209e-007, No Iterations 4
GAMG: Solving for p, Initial residual = 7.87121e-006, Final residual = 6.14078e-007, No Iterations 5
time step continuity errors : sum local = 1.01978e-009, global = 2.6872e-017, cumulative = 4.25407e-016
smoothSolver: Solving for nuTilda, Initial residual = 9.92605e-006, Final residual = 8.54953e-007, No Iterations 2
ExecutionTime = 11.268 s ClockTime = 11 s
                                                                                                                                                  Čas preračuna
SIMPLE solution converged in 350 iterations
End
                                         Uspešno izveden preračun
jakap@DESKTOP-KKPTPA9 MINGW64 OpenFOAM-5.x ~/blueCFD/ofuser-of5/prvaSimulacija/airFoil2D
```

- Paraview je bil naložen skupaj z OpenFOAM-om
  - Rezultat simulacije s pomočjo virov in filtrov prikaže v razumljivi obliki

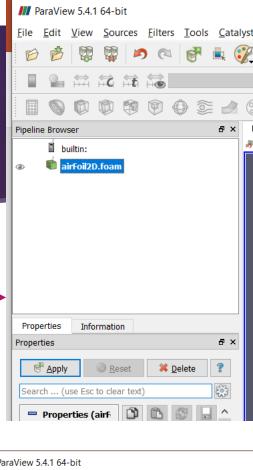


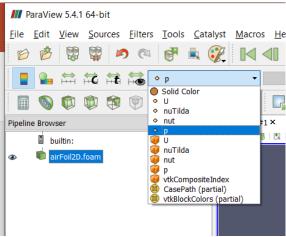
Vir: območje zajema podatkov

Filter: bolj ali manj kompleksna operacija na podatkih

Viri so večinoma že vgrajeni v filtre, zato jih pogosto ni potrebno specificirati

- ▶ 2. vaja:
  - 1. Odpri blueCFD-Core 2017 terminal
  - S pomočjo cd in Is se premakni v mapo ~\blueCFD-Core-2017\ofuser-of5\OFkurz\simulacije\_naloge\airFoil2D
  - Poženi ukaz paraFoam (ta ukaz prikaže rezultate simulacije v programu paraview)
  - 4. Pritisni "Apply", da naložiš izbrane podatke
  - 5. S pomočjo spustnega menija lahko opazuješ različna polja.
    - Izberi in preglej hitrostno polje U
    - 2. Izberi in preglej tlačno polje p



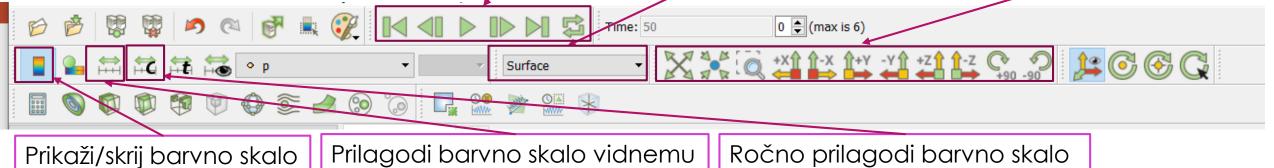


- ▶ 3. vaja:
  - 1. Premakni pogled: srednji miškin klik
  - 2. Zavrti pogled: levi miškin klik
  - 3. Približaj, oddalji: miškino kolo
  - 4. Preizkusi naslednje ukase:

Opazuj rezultate ob različnih časih/iteracijah

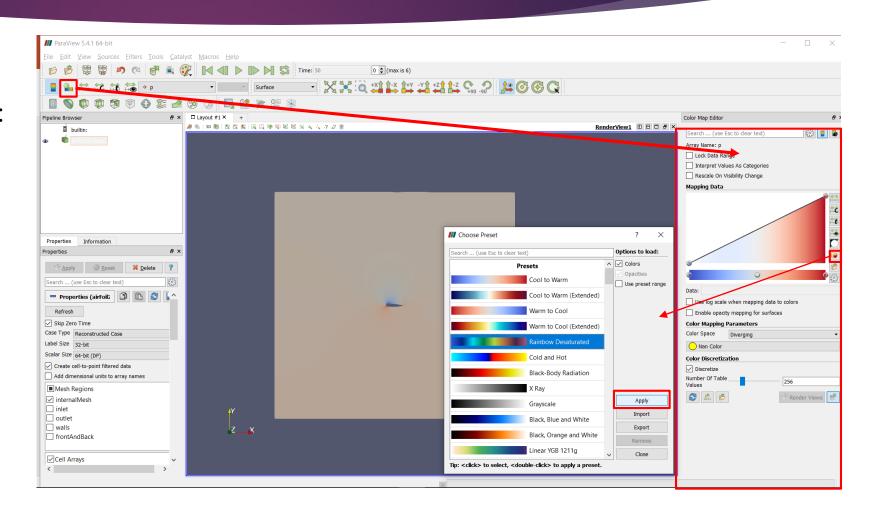
Izberi način prikaza ploskev

Orientiraj pogled

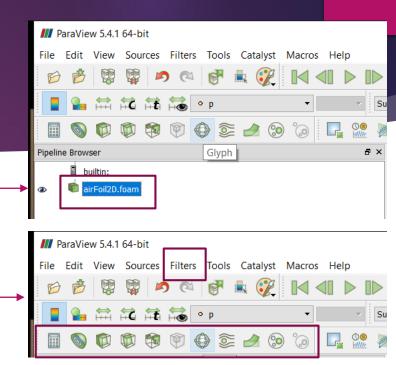


Vaja 4:

Nastavi novo barvno lestvico:



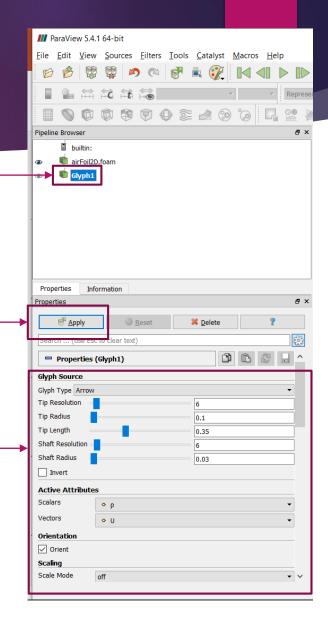
- Filtri:
  - ▶ Izberi željen set podatkov iz "Pipeline Browser"-ja
  - ▶ Izberi željen filter -
    - ▶ Nastavi potrebne nastavitve
    - "Apply"
    - ▶ Generira se nov set podatkov



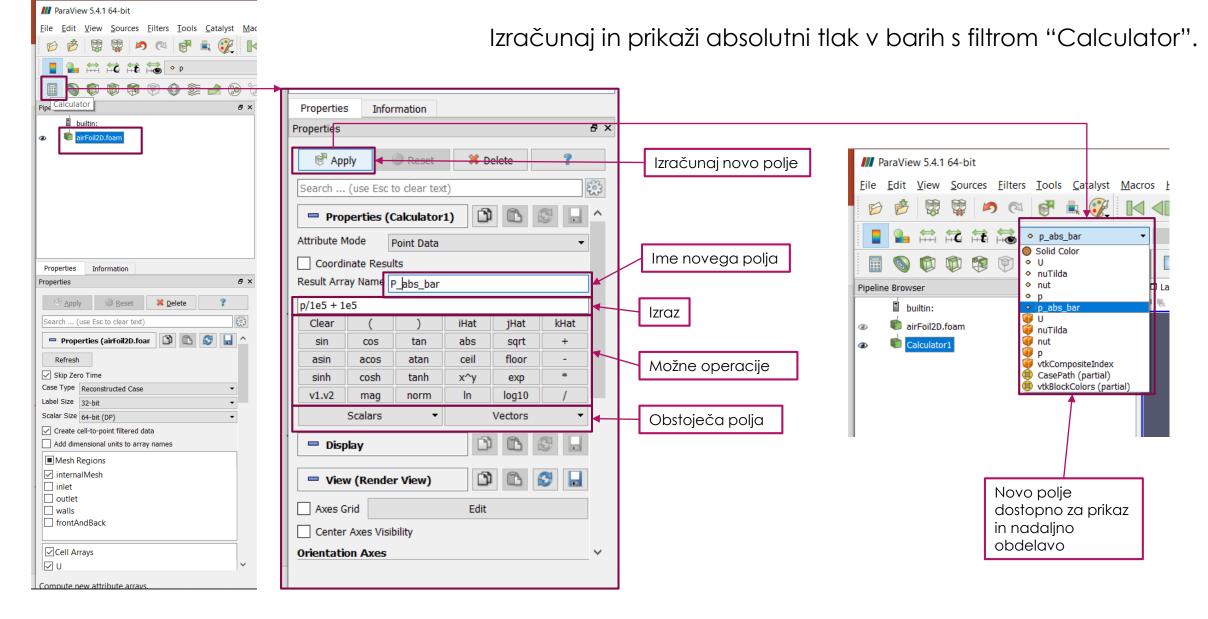
- Filtri:
  - Izberi željen set podatkov iz "Pipeline Browser"-ja (vhodni podatki za filter)
  - Izberi željen filter
    - Nastavi potrebne nastavitve
    - "Apply"
    - Generira se nov set podatkov
  - S filtrom generirani seti podatkov so lahko vhodi v nove filtre
  - Vidnost setov podatkov lahko spremenimo s klikom na





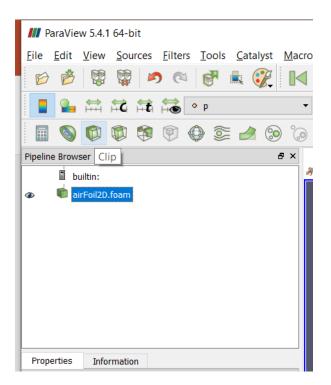


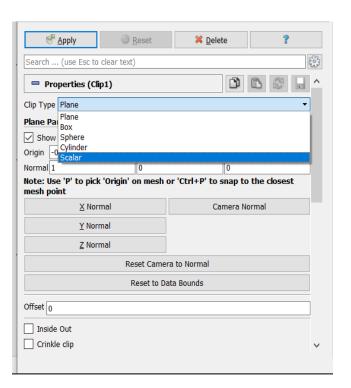
Vaja 5:

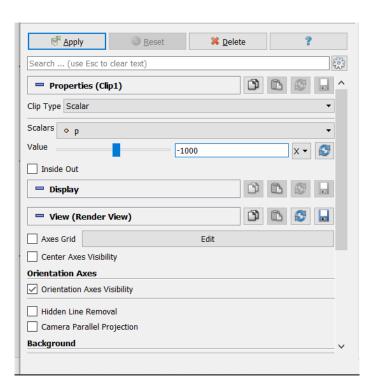


#### Vaja 6:

Prikaži območje, kjer tlak presega 1000 Pa s pomočjo filtra "Clip"

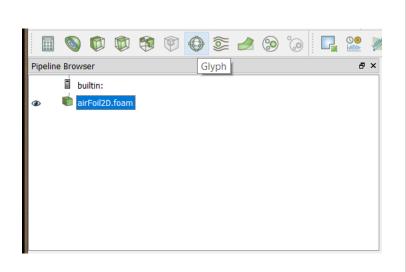


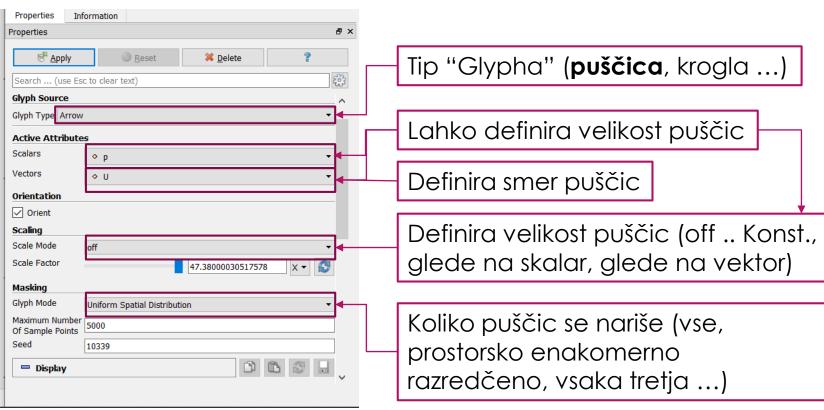




Vaja 6:

S pomočjo filtra "Glyph" izriši vektorje hitrosti.





▶ Filter "Slice": 3D domeno prereže z ravnino in vrne 2D rezino

Filter "Clip", varianta "Plane": 3D domeno razpolovi z ravnino in vrne eno izmed polovic

Filter "Stream tracer": generira točke na črti ali v krogli, nato pa jih propagira v smeri toka, rezultat so tokovnice

- Vaja 5:
  - Ponovi preračun toka okoli krila pri drugačni hitrosti na vstopu v domeno:
    - ▶ Izbriši rezultate s skripto ./Allclean

(Znebiti se moraš vseh map, razen 0, constant in system, to lahko izvedeš tudi ročno v raziskovalcu)

Podvoji hitrostni začetni pogoj in robne pogoje v 0/U in shrani spremenjeno datoteko

**Pozor!** Datotek ne smete urejati z windowsovo beležko. Uporabite ali Notepad ++ ali pa Notepad2.

Ponovno izvedi preračun z ukazom simpleFoam in preglej rezultat s pomočjo ukaza paraFoam

```
dimensions
                [0 1 -1 0 0 0 0];
                uniform (50 3.62 0);
internalField
boundaryField
    inlet
        freestreamValue uniform (50 3.62 0)
    outlet
        freestreamValue uniform (50 3.62 0
    walls
                         noSlip;
        type
    frontAndBack
        type
                         empty;
```