

# CFD aerodinamika

UVOD V OPENFOAM IN PARAVIEW

# OpenFOAM

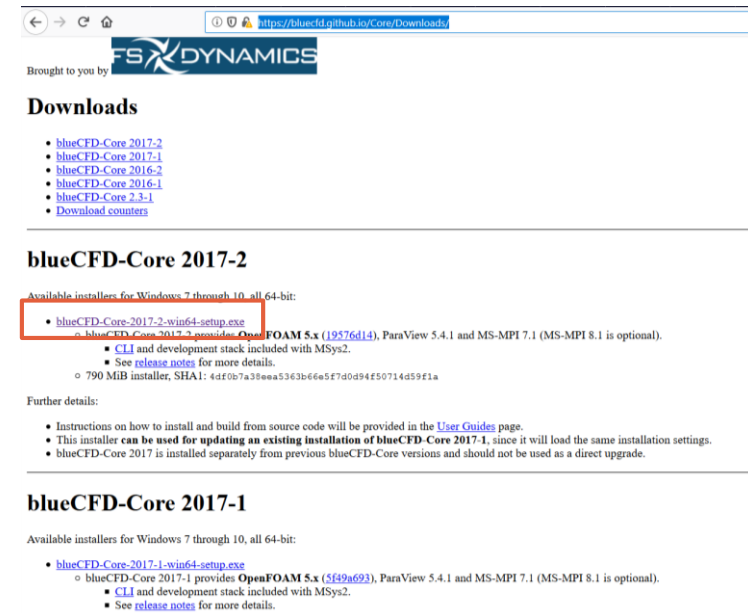
► Zbirka programov za CFD analize raznih tokov

► Naloga 0: inštaliraj OpenFOAM za Windows:

► <https://bluecfd.github.io/Core/Downloads/>

► Za Ubuntu:

► <https://openfoam.org/download/7-ubuntu/>



The screenshot shows a web browser window with the address bar displaying <https://bluecfd.github.io/Core/Downloads/>. The page features the blueCFD logo and a 'Downloads' section. Under 'Downloads', there is a list of links for various versions of blueCFD-Core. The 'blueCFD-Core 2017-2' section is highlighted with a red box, showing the download link for 'blueCFD-Core-2017-2-win64-setup.exe'. Below this, there are instructions for installation, including a note that the installer can be used for updating an existing installation of blueCFD-Core 2017-1. The 'blueCFD-Core 2017-1' section is also visible, showing the download link for 'blueCFD-Core-2017-1-win64-setup.exe'.

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

- [blueCFD-Core-2017-2-win64-setup.exe](#)
  - blueCFD-Core 2017-2 provides **OpenFOAM 5.x** (19576d14), ParaView 5.4.1 and MS-MPI 7.1 (MS-MPI 8.1 is optional).
    - See [release notes](#) for more details.
  - 790 MiB installer, SHA1: 4df0b7a38eaa5363b66e5f7d0d94f50714d59f1a

Further details:

- 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** (5f9a693), ParaView 5.4.1 and MS-MPI 7.1 (MS-MPI 8.1 is optional).
    - See [release notes](#) for more details.

# OpenFOAM

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 ...

- Datoteke z nastavitvami urejamo z Notepad2

# OpenFOAM

## ► 1. vaja:

1. Odpri **blueCFD-Core 2017 terminal (namizje)**

2. Vnesi ukaz: **\$ cd blueCFD/ofuser-of5/**

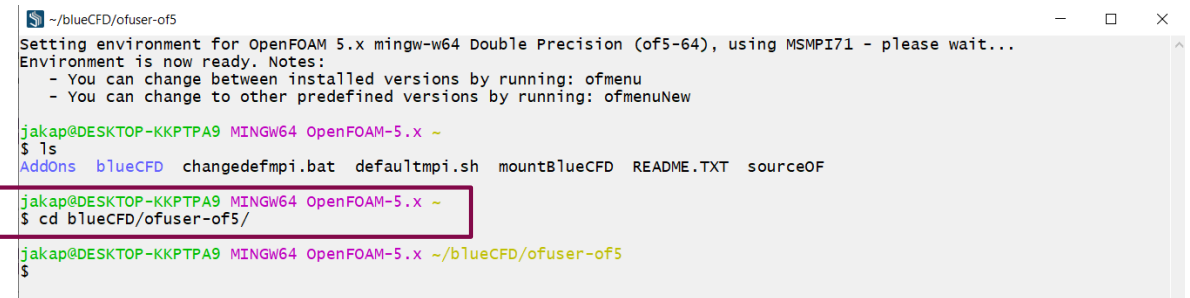
3. Vnesi ukaz: **\$ git clone https://github.com/jpeter3/OFkurz.git**

4. V terminalu se s pomočjo ukazov **ls** (pokaži možne direktorije) in **cd** (change directory) premakni v mapo:

**~\blueCFD\ofuser-of5\OFkurz\simulacije\_naloge\airFoil2D**

5. V terminalu poženi program za izračun nestisljivega, časovno ustaljenega turbulentnega toka:

**\$ simpleFoam**



```
~/blueCFD/ofuser-of5
Setting environment for OpenFOAM 5.x mingw-w64 Double Precision (of5-64), using MSMPI71 - please wait...
Environment is now ready. Notes:
- 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 ~
$ ls
Addons  blueCFD  changedefmpi.bat  defaultmpi.sh  mountBlueCFD  README.TXT  sourceOF

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

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

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

# OpenFOAM

Simulirani čas/iteracija

~/blueCFD/ofuser-of5/prvaSimulacija/airFoil2D

```
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 : 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 = 10.998 s  ClockTime = 11 s
```

Ostanek reda velikosti  $1e-6$  je značilen za konvergirano rešitev

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

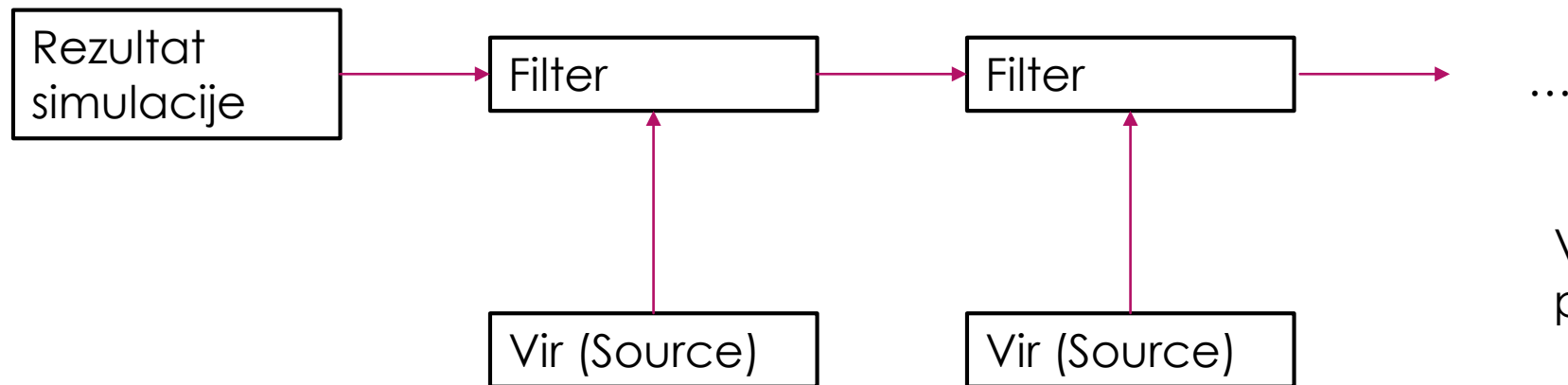
Uspešno izveden preračun

jakap@DESKTOP-KKPTPA9 MINGW64 OpenFOAM-5.x ~/blueCFD/ofuser-of5/prvaSimulacija/airFoil2D

\$

# Paraview

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



Vir: območje zajema podatkov

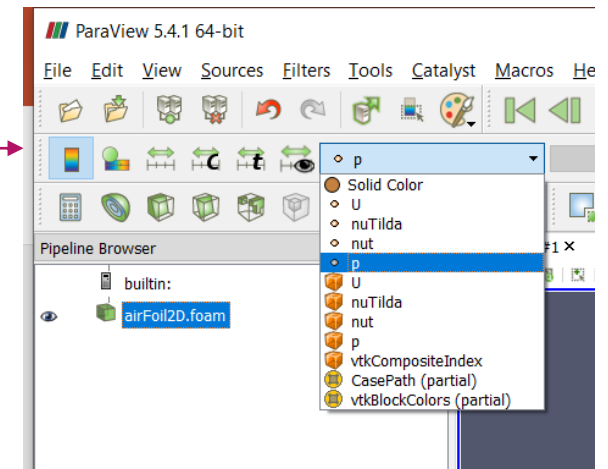
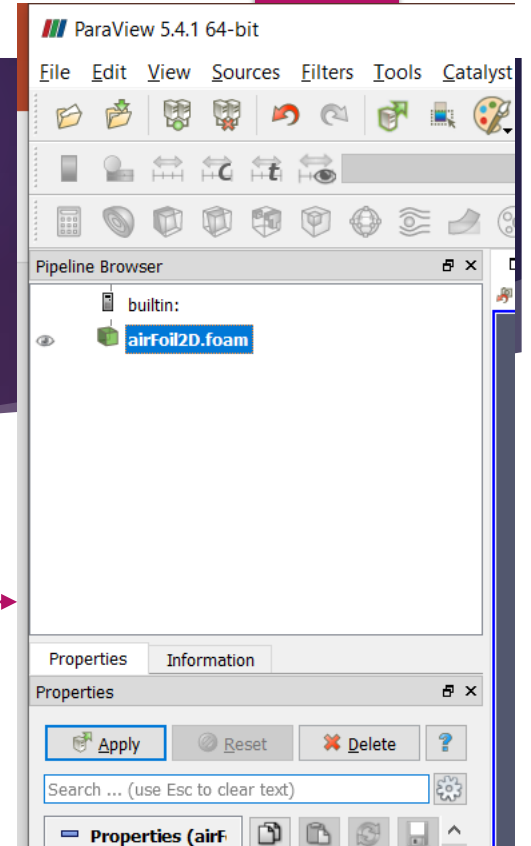
Filter:  
bolj ali manj kompleksna  
operacija na podatkih

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

# Paraview

## ► 2. vaja:

1. Odpri **blueCFD-Core 2017 terminal**
2. S pomočjo **cd** in **ls** se premakni v mapo  
`~\blueCFD-Core-2017\ofuser-of5\OFkurz\simulacije_naloge\airFoil2D`
3. 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
  1. Izberi in preglej hitrostno polje **U**
  2. Izberi in preglej tlačno polje **p**



# Paraview

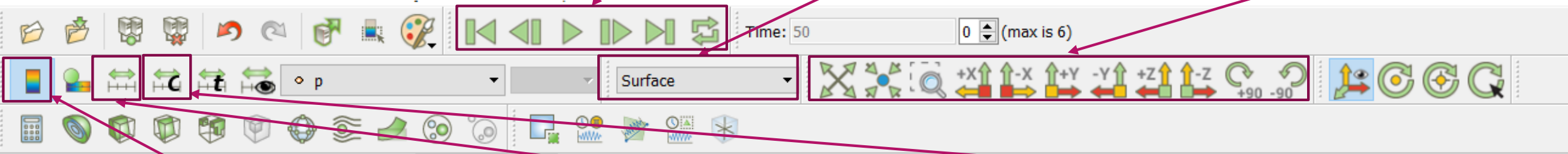
## ► 3. vaja:

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



Prikaži/skrij barvno skalo

Prilagodi barvno skalo vidnemu

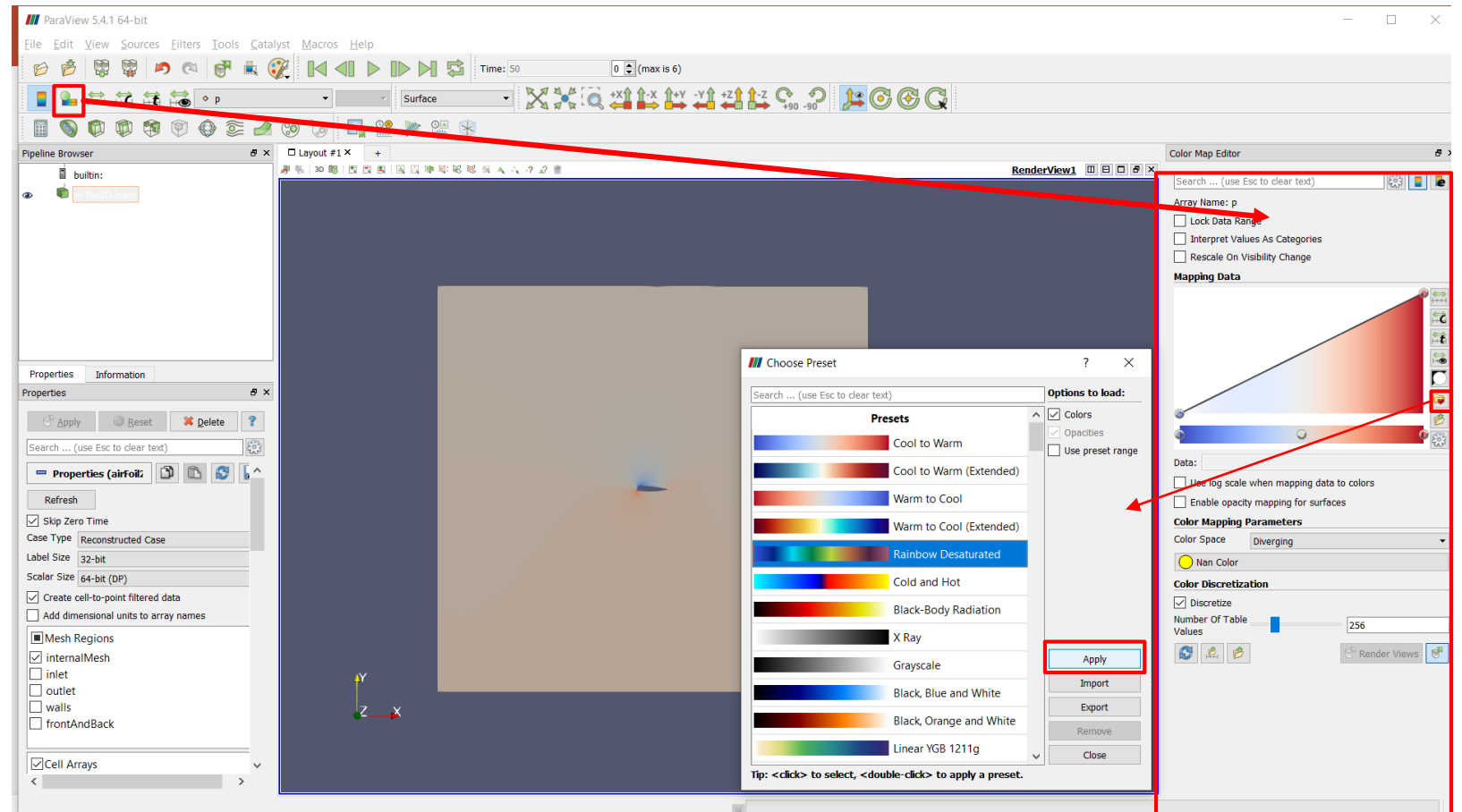
Ročno prilagodi barvno skalo



# Paraview

Vaja 4:

Nastavi novo barvno lestvico:



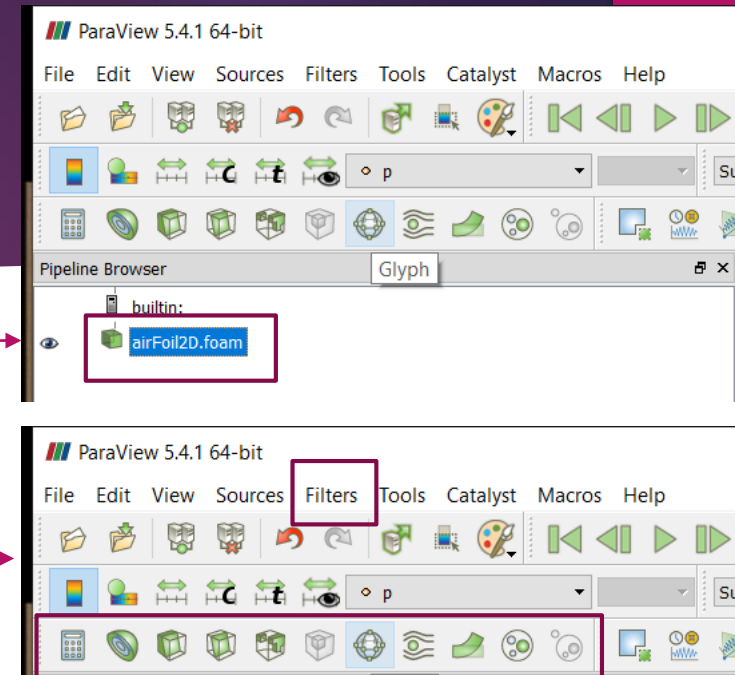
# Paraview

## ► Filtri:

► Izberi željen set podatkov iz "Pipeline Browser"-ja

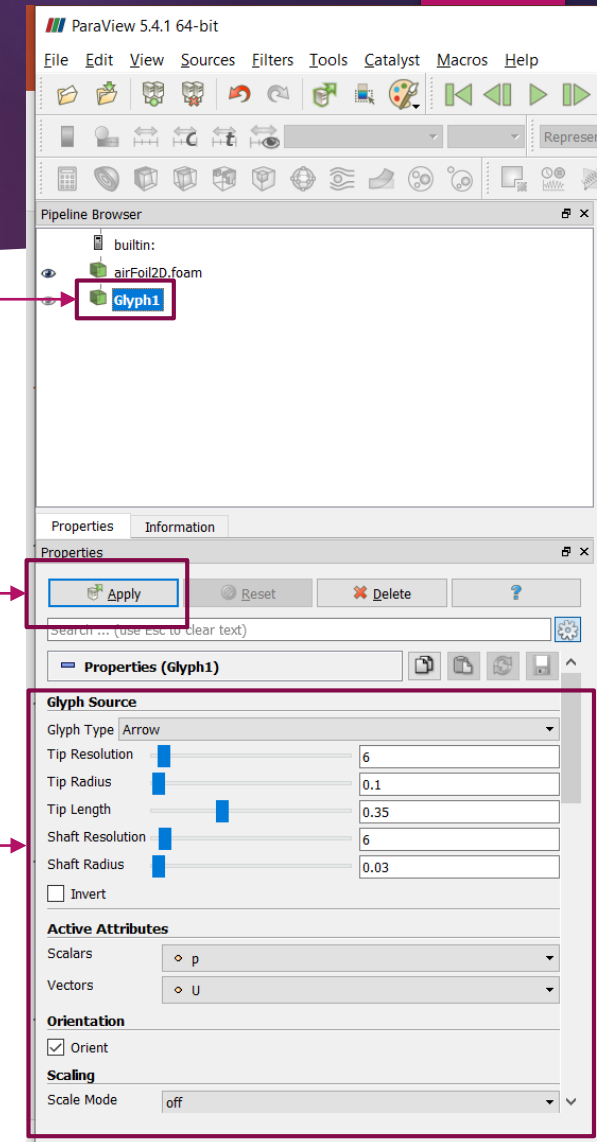
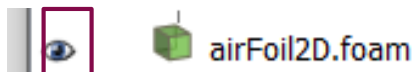
► Izberi željen filter

- Nastavi potrebne nastavitve
- "Apply"
- Generira se nov set podatkov



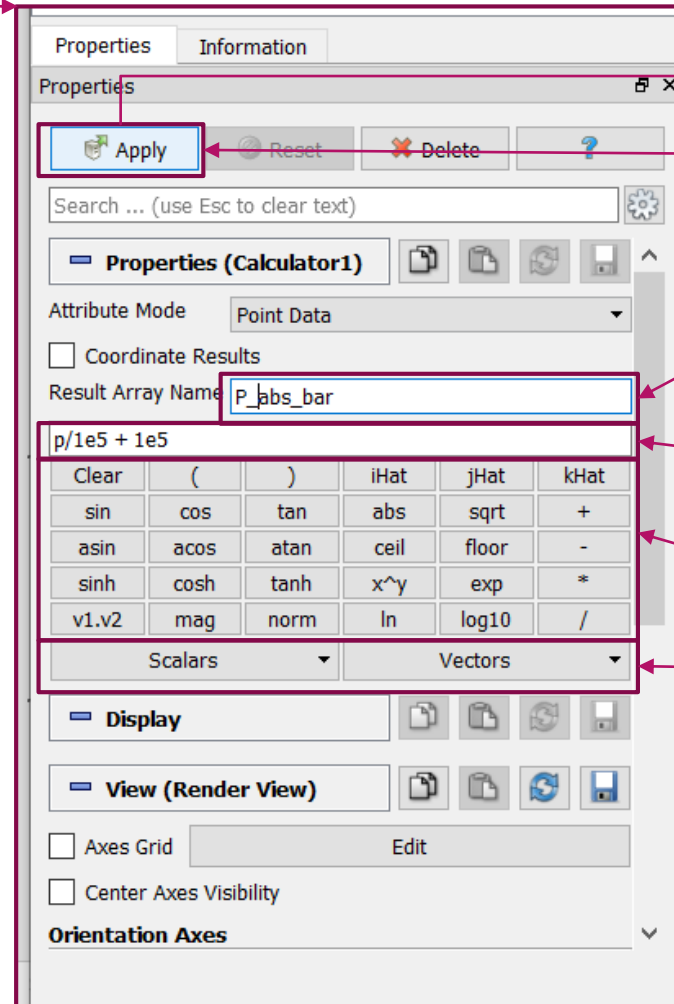
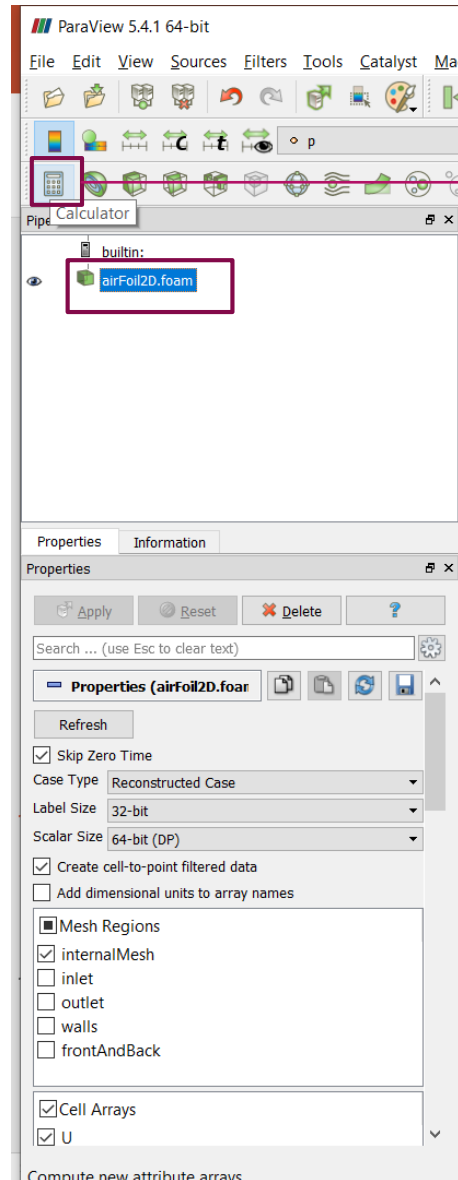
# Paraview

- ▶ 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:

Izračunaj in prikaži absolutni tlak v barih s filtrom "Calculator".



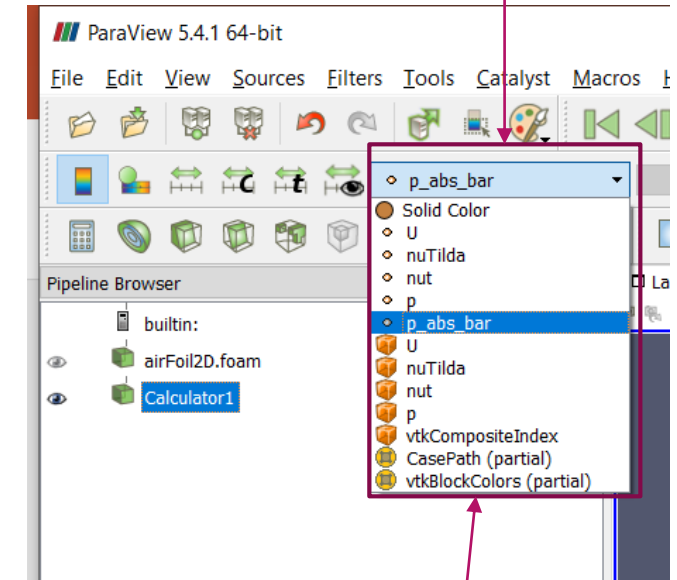
Izračunaj novo polje

Ime novega polja

Izraz

Možne operacije

Obstoječa polja

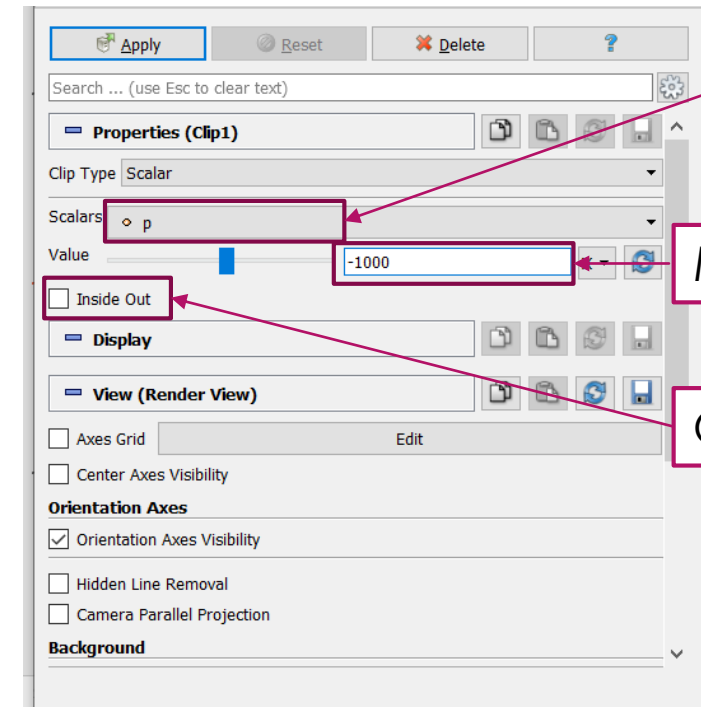
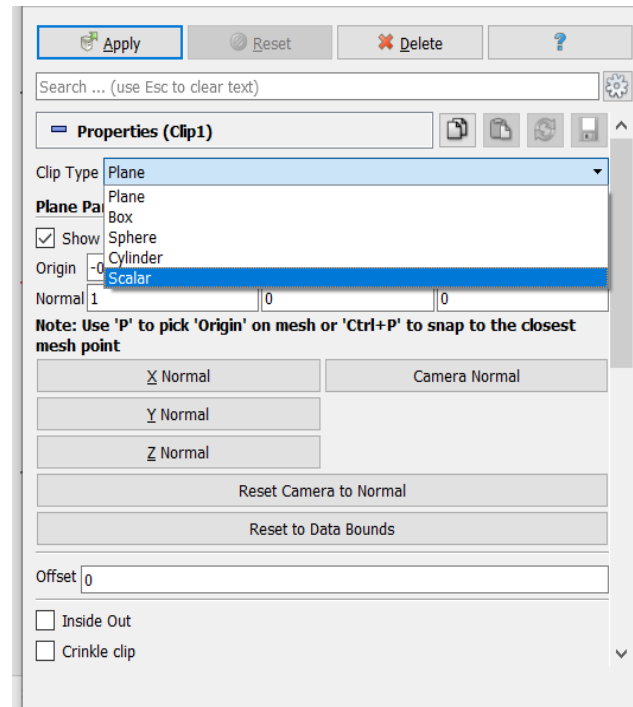


Novo polje  
dostopno za prikaz  
in nadaljno  
obdelavo

# Paraview

Vaja 6:

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



Izbrano polje

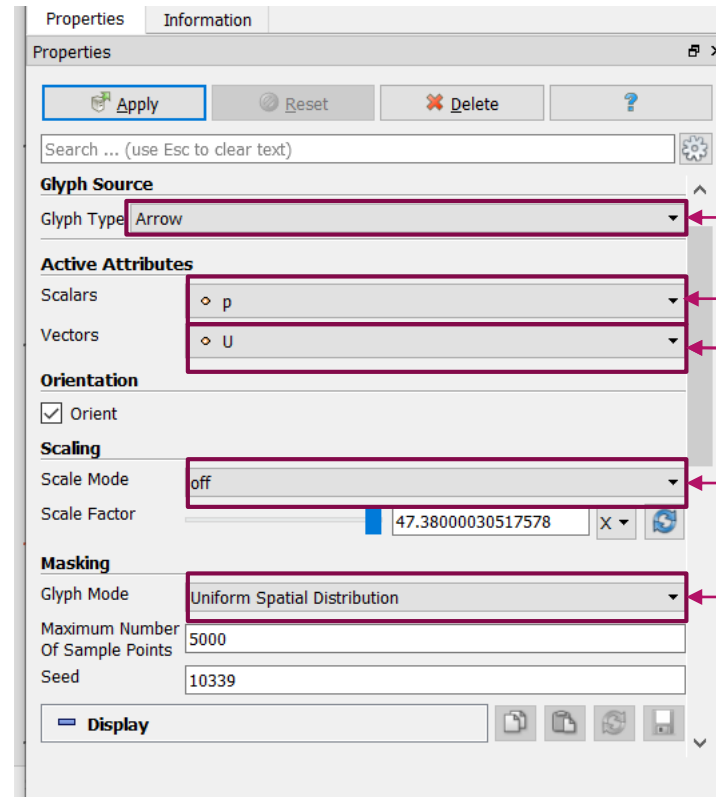
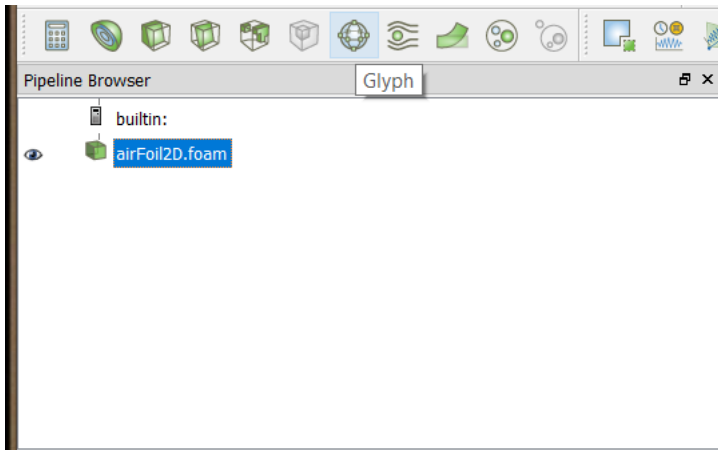
Mejna vrednost

Obrne "Clip"

# Paraview

Vaja 6:

S pomočjo filtra “Glyph” izriši vektorje hitrosti.



Tip “Glypha” (**puščica**, krogla ...)

Lahko definira velikost puščic

Definira smer puščic

Definira velikost puščic (off .. Konst., glede na skalar, glede na vektor)

Koliko puščic se nariše (vse, prostorsko enakomerno razredčeno, vsaka tretja ...)

# Paraview

- ▶ 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

# OpenFOAM

- ▶ 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];
internalField    uniform (50 3.62 0);

boundaryField
{
    inlet
    {
        type      freestream;
        freestreamValue uniform (50 3.62 0);
    }

    outlet
    {
        type      freestream;
        freestreamValue uniform (50 3.62 0);
    }

    walls
    {
        type      noSlip;
    }

    frontAndBack
    {
        type      empty;
    }
}
```