

CFD aerodinamika

UVOD V OPENFOAM IN PARAVIEW

OpenFOAM

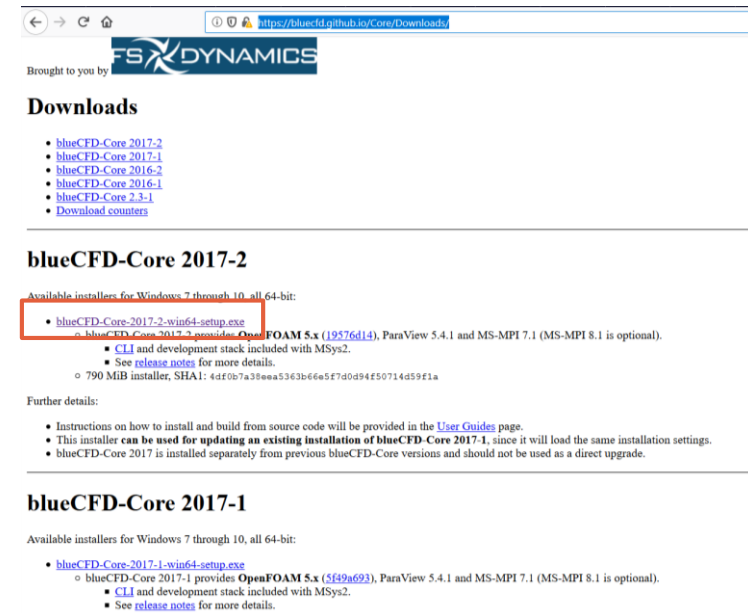
- Zbirka programov za CFD analize raznih tokov

- Za Windows:

- <https://bluecfcd.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://bluecfcd.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 a list of available installers for Windows 7 through 10, all 64-bit. The first item in this list is 'blueCFD-Core-2017-2-win64-setup.exe', which is also highlighted with a red box. Below this, there are links for 'blueCFD-Core 2017-1' and 'blueCFD-Core 2016-2'. The 'blueCFD-Core 2017-1' section is also visible, showing a list of available installers for Windows 7 through 10, all 64-bit. The first item in this list is 'blueCFD-Core-2017-1-win64-setup.exe', which is also highlighted with a red box. Below this, there are links for 'blueCFD-Core 2017-2' and 'blueCFD-Core 2016-2'.

Brought to you by **blueCFD**

Downloads

- [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

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).
 - [CLI](#) and development stack included with MSys2.
 - See [release notes](#) for more details.
 - 790 MiB installer, SHA1: 4df0b7a38ee5363b66e5f7d0d94f50714d59f1a

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).
 - [CLI](#) and development stack included with MSys2.
 - 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 ...

- Urejamo npr. z notepad++:
<https://notepad-plus-plus.org/downloads/v7.7.1/>

OpenFOAM

- ▶ 1. vaja:
 - ▶ S pomočjo raziskovalca v **F:\ProgramFiles\blueCFD-Core-2017\ofuser-of5** ustvari mapo "**prvaSimulacija**"
 - ▶ S pomočjo raziskovalca v mapo
F:\ProgramFiles\blueCFD-Core-2017\ofuser-of5\prvaSimulacija
prekopiraj mapo
F:\ProgramFiles\blueCFD-Core-2017\OpenFOAM5.x\tutorials\incompressible\simpleFoam\airFoil2D
(to je prednastavljeni testni primer, ki izračuna tok okoli krila, v mapi so shranjeni robin pogoji, materialni parametri ...)
 - ▶ Odpri **blueCFD-Core 2017 terminal**
 - ▶ V terminalu se s pomočjo ukazov **ls** (pokaži možne direktorije) in **cd** (change directory) premakni v mapo:
~\blueCFD-Core-2017\ofuser-of5\prvaSimulacija\airFoil2D
 - ▶ V terminalu poženi program za izračun nestisljivega, časovno ustaljenega turbulentnega toka: **simpleFoam**

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 $1\text{e-}6$ 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

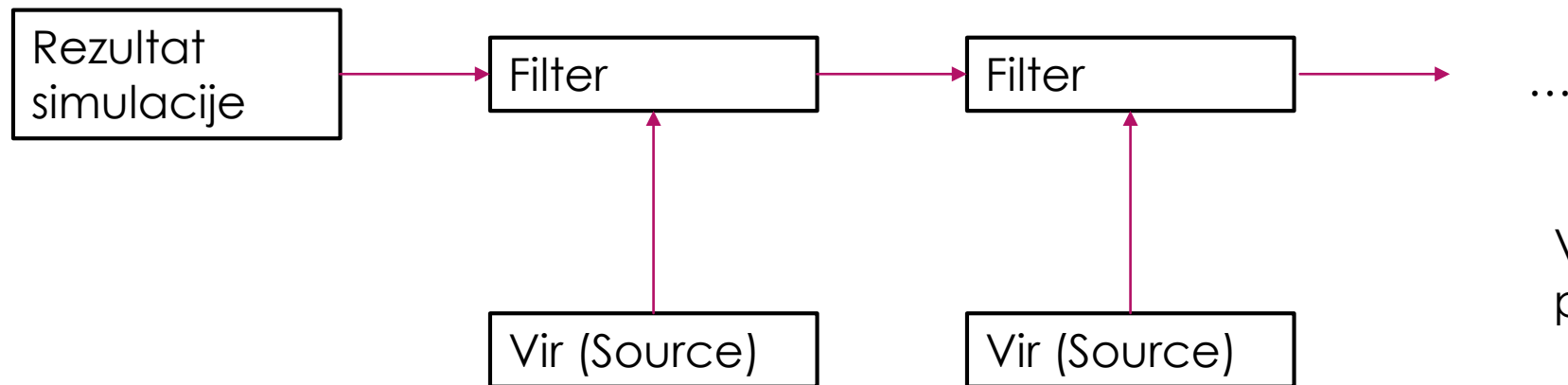
Uspešno izveden preračun

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



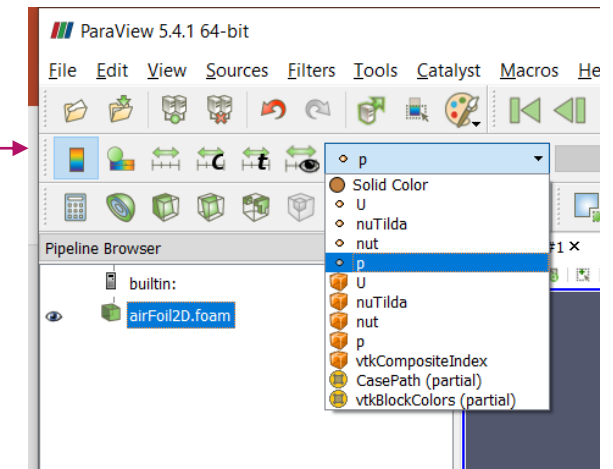
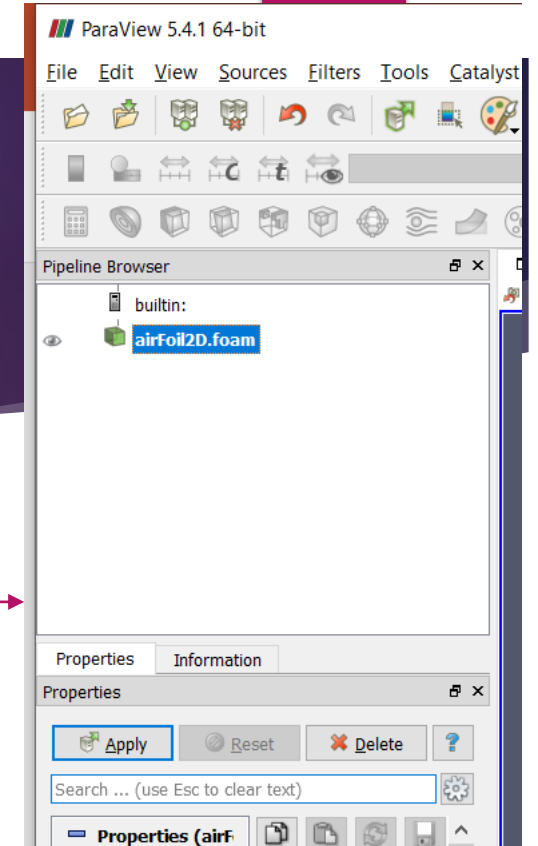
Vir: območje zajema podatkov

Filter:
bolj ali manj kompleksna operacija na podatkih

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

Paraview

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



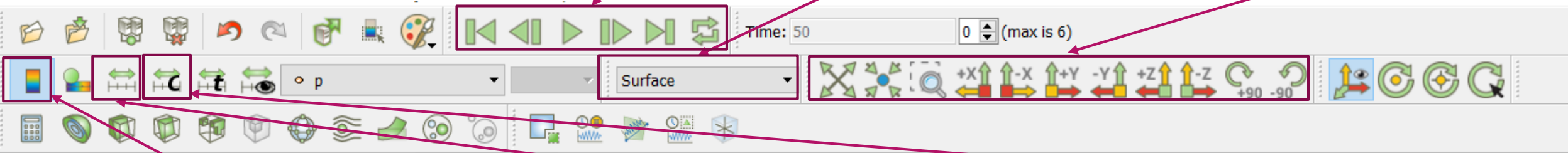
Paraview

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

Opazuj rezultate ob različnih časih/iteracijah

Izberi način prikaza ploskev

Orientiraj pogled



Prikaži/skrij barvno skalo

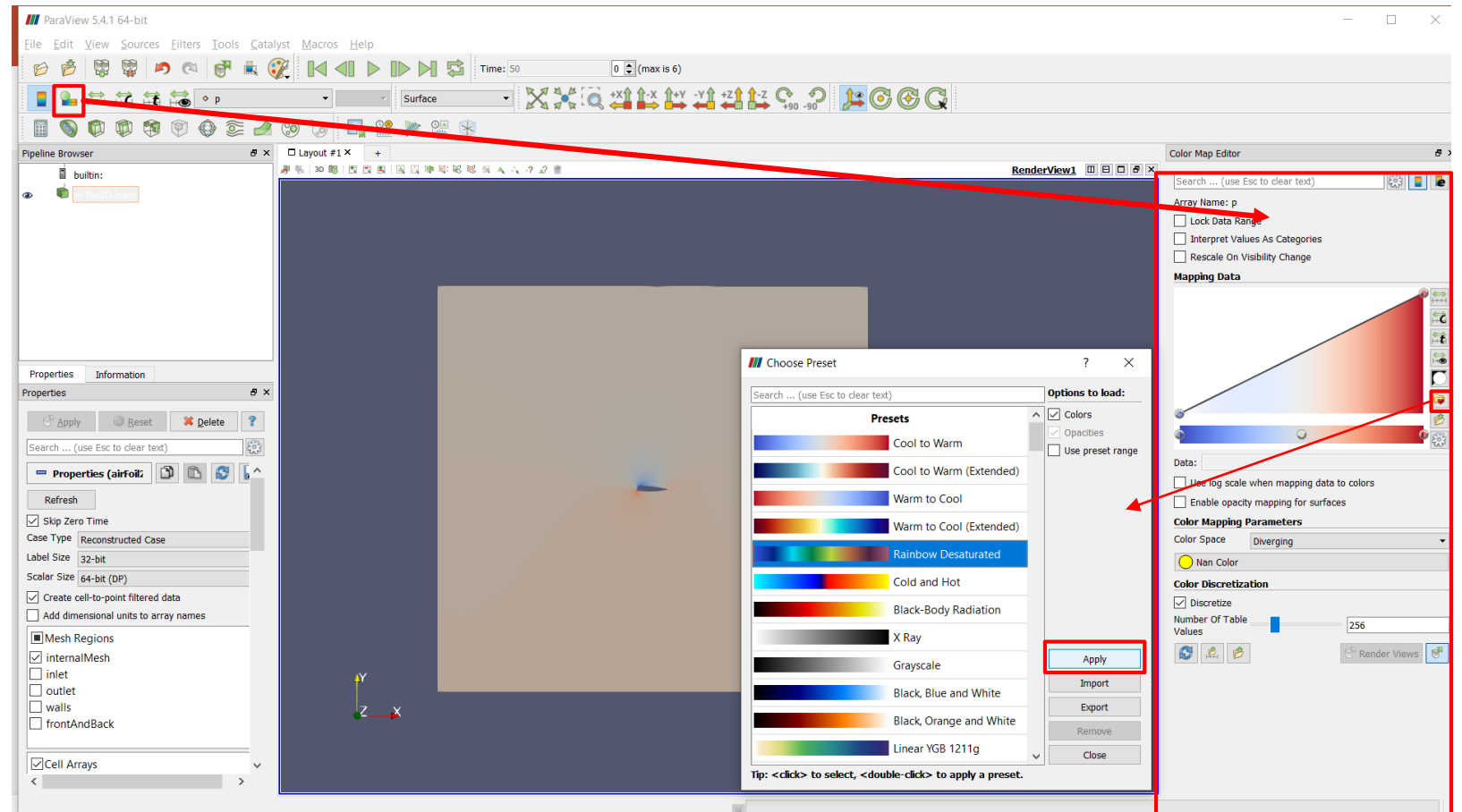
Prilagodi barvno skalo vidnemu

Ročno prilagodi barvno skalo

Paraview

Vaja 4:

Nastavi novo barvno lestvico:



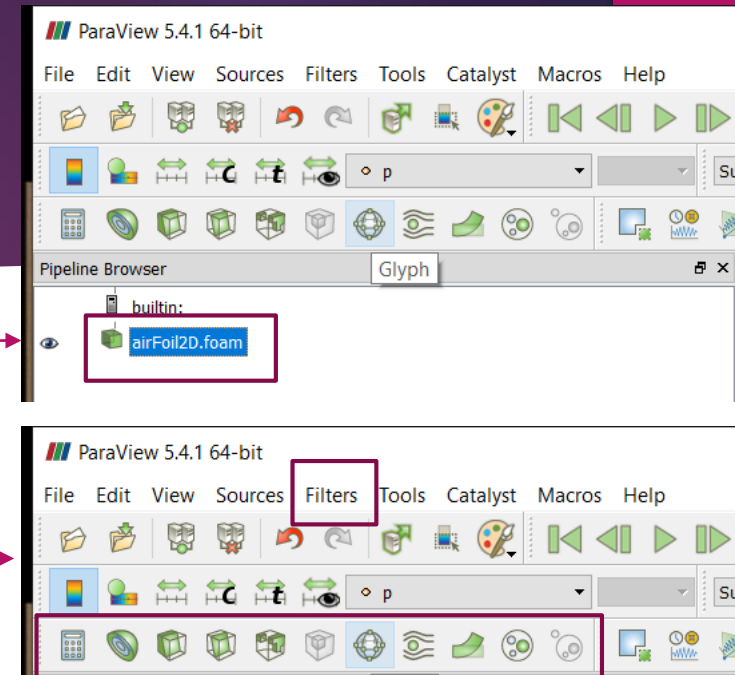
Paraview

► Filtri:

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

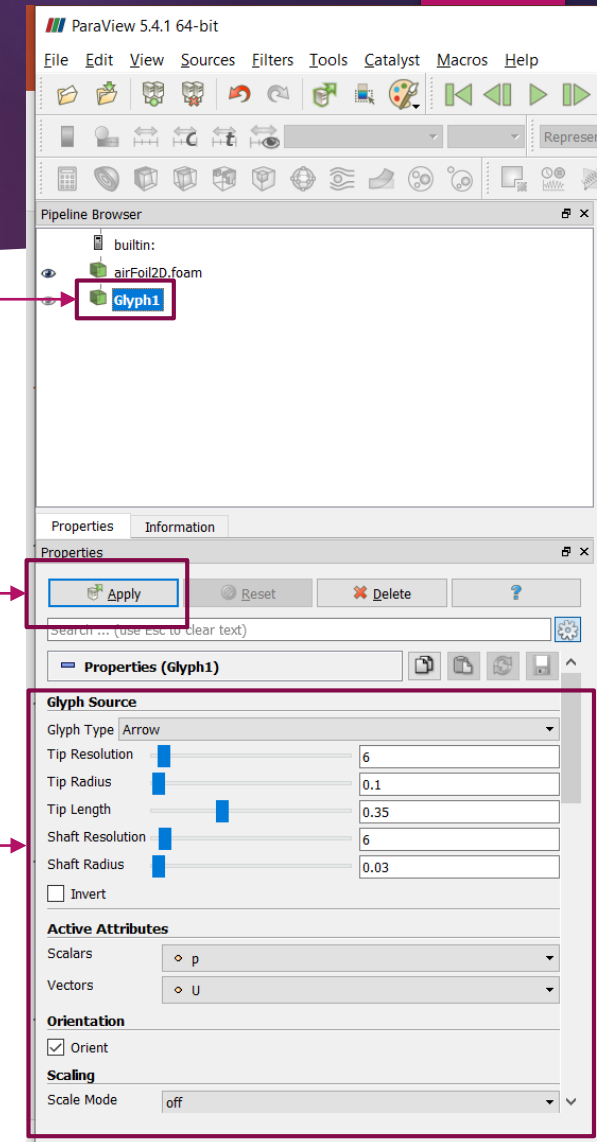
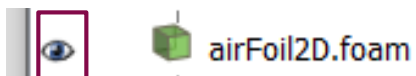
► Izberi željen filter

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



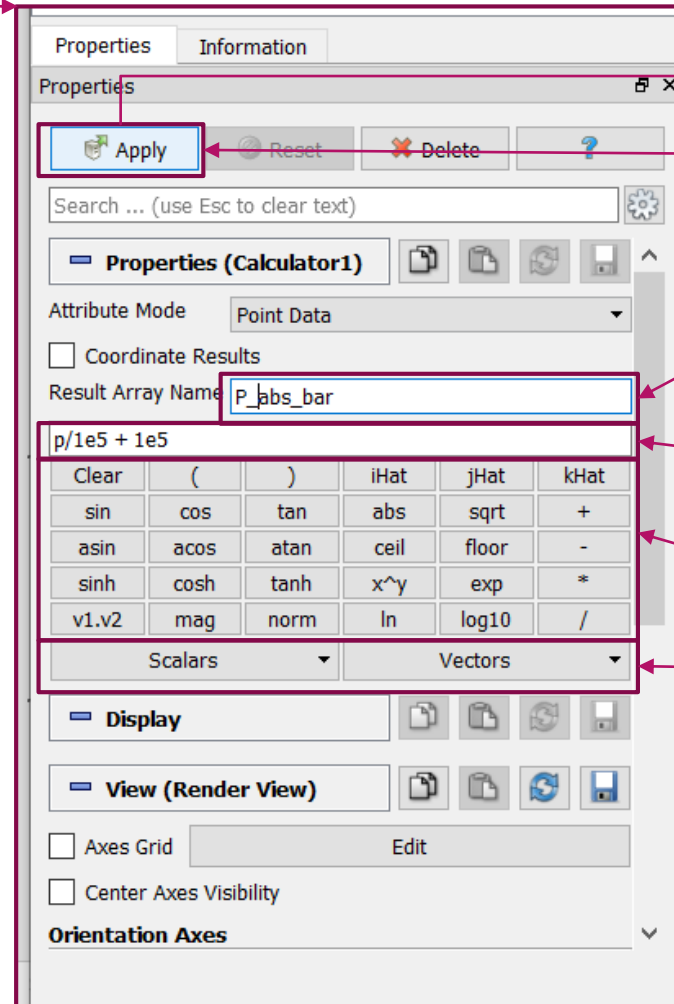
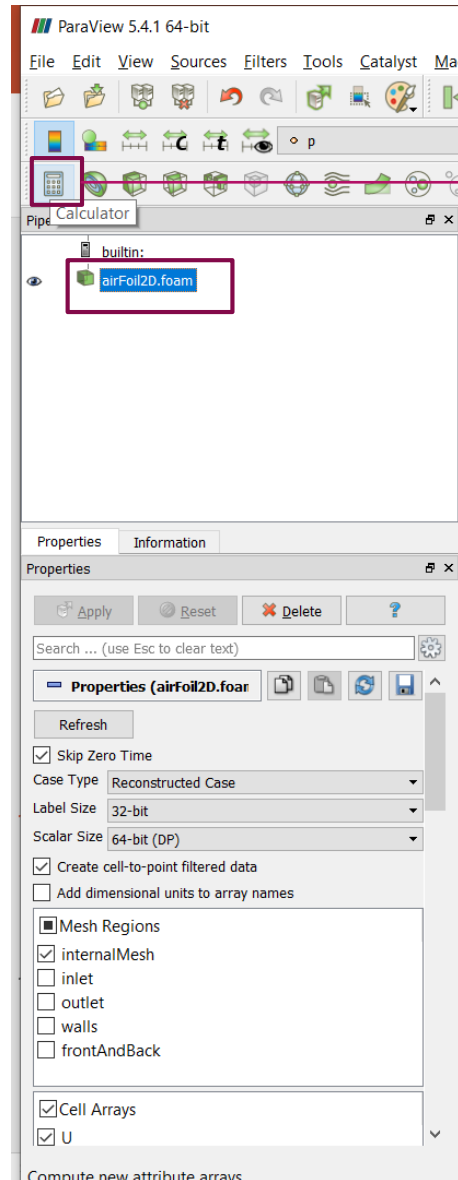
Paraview

- ▶ Filtri:
 - ▶ Izberi željen set podatkov iz "Pipeline Browser"-ja
 - ▶ 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".



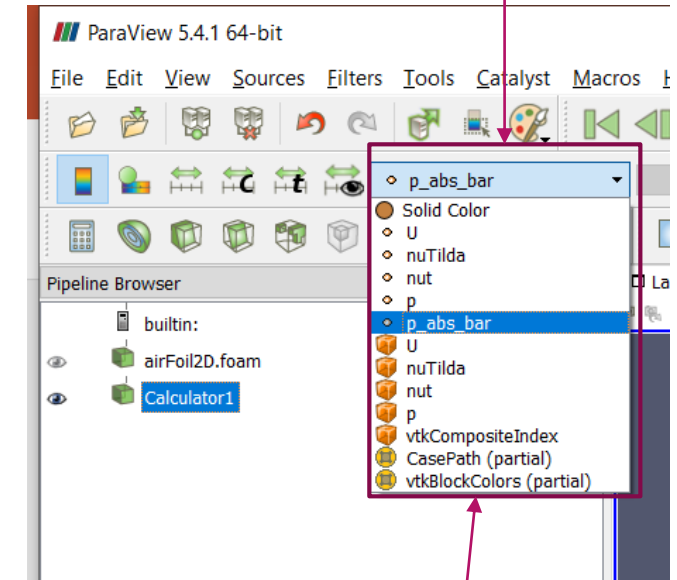
Izračunaj novo polje

Ime novega polja

Izraz

Možne operacije

Obstoječa polja

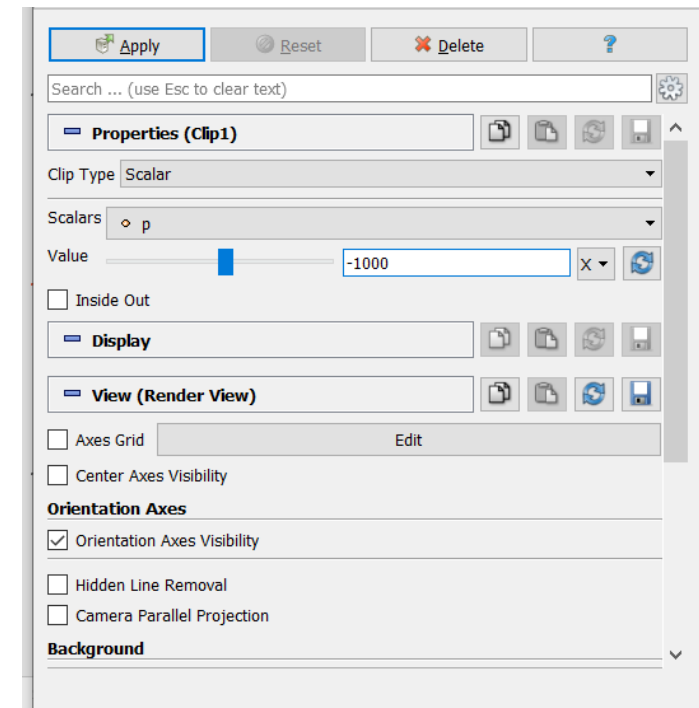
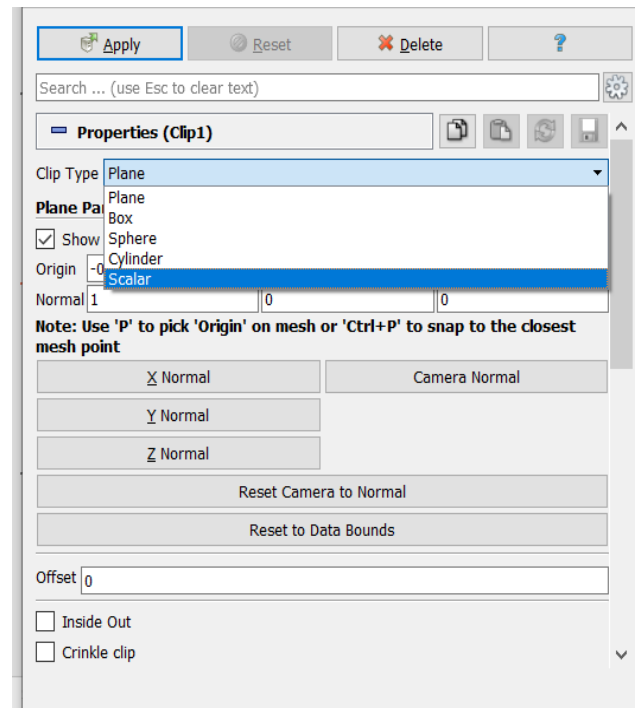


Novo polje
dostopno za prikaz
in nadaljno
obdelavo

Paraview

Vaja 6:

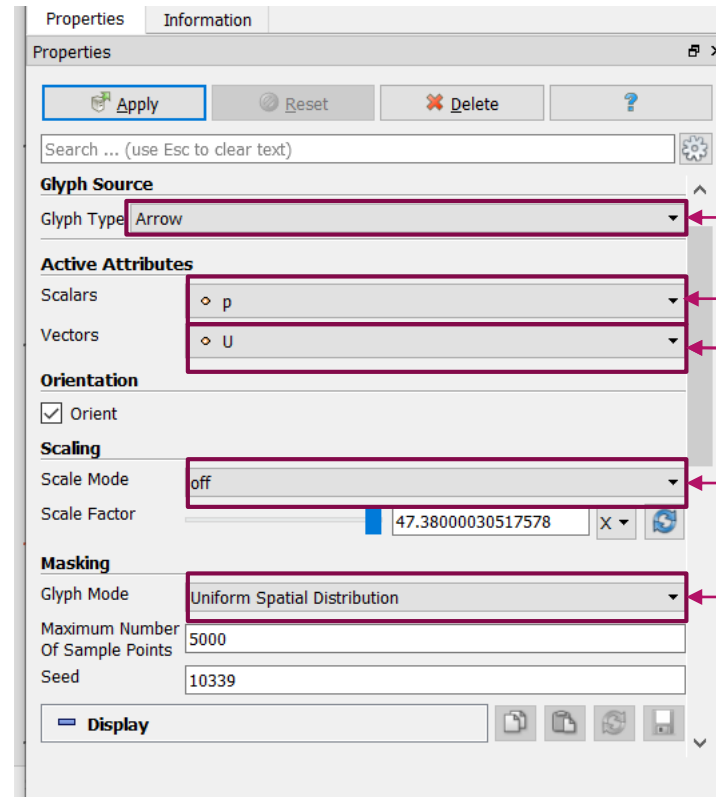
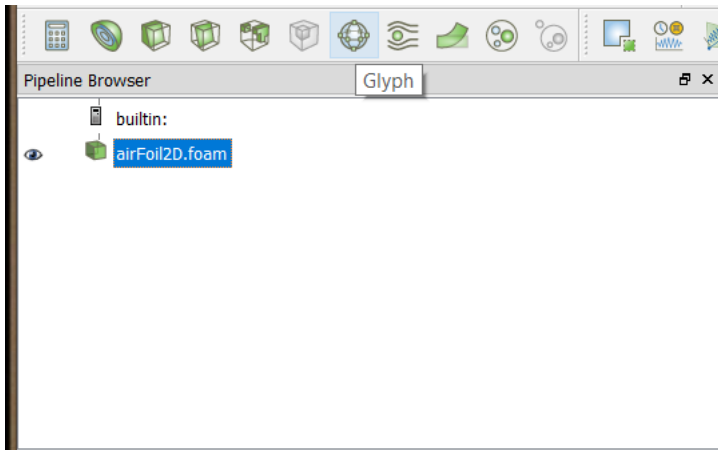
Prikaži območje, kjer tlak presega 1000 Pa s pomočjo filtra “Clip”



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)
 - ▶ Prilagodi hitrostni začetni pogoj in robne pogoje v **0/U** in shrani spremenjeno datoteko
 - ▶ 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;  
    }  
}
```