

# CFD aerodinamika

IZRAČUN AERODINAMIKE OKOLI DIRKALNIKA S POMOČJO  
OPENFOAM+SIMSCALE+PARAVIEW

# Potek dela



# Modeliranje

- ▶ “Lepa” geometrija:
  - ▶ Brez ostrih robov
  - ▶ Brez tankih rež
  - ▶ Brez konic, koničastih lukenj
  - ▶ Brez slabo ujemajočih se površin
- ▶ Oblikovanje aerodinamičnih naprav je posebna zgodba, ki je trenutno še ne bomo obdelali

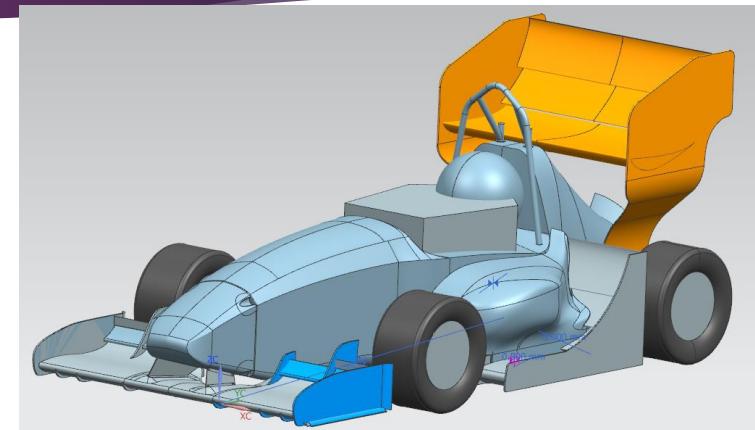
# Postavljanje v koordinate

- ▶ En sestav v katerem se dodaja, spreminja komponente
  - ▶ Sestav naj zaenkrat ne vsebuje detajlov reda velikosti okoli 1 cm (to je celotno podvozje, pokrovi, detajli v feltnah, vijaki za pritrditev aerodinamičnih naprav)
  - ▶ Sestav naj bo čez celotno sezono enako orientiran, da stl datoteke nespremenjenih kosov ostanejo enake
  - ▶ Spremenjeno/novo komponento umestimo v sestav

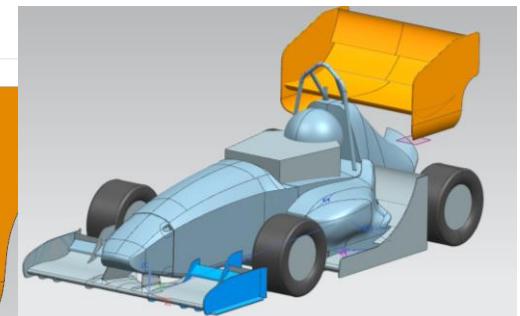
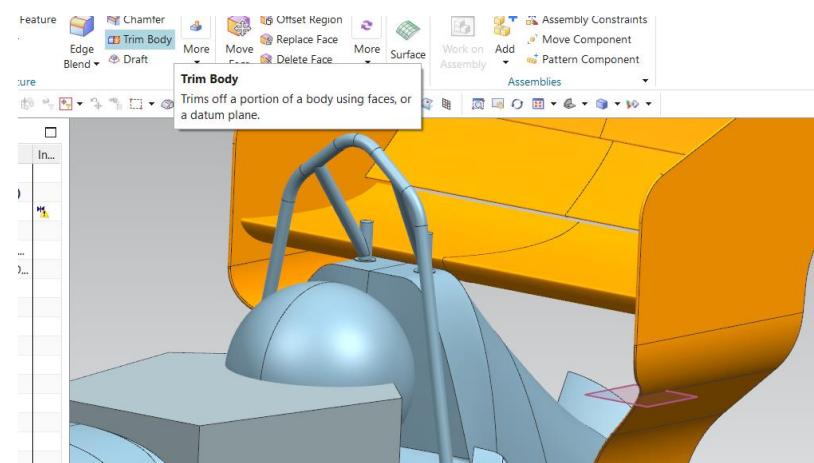
# Modeliranje in postavljanje v koordinate

## ► Naloga 1:

- V mapi OFkurz\simulacije\_naloge\naloga\_formulaCFD razpni completeAeroFin.zip in z NX-om odpri sestav COMPLETE\_AERO\_SESTAV.prt

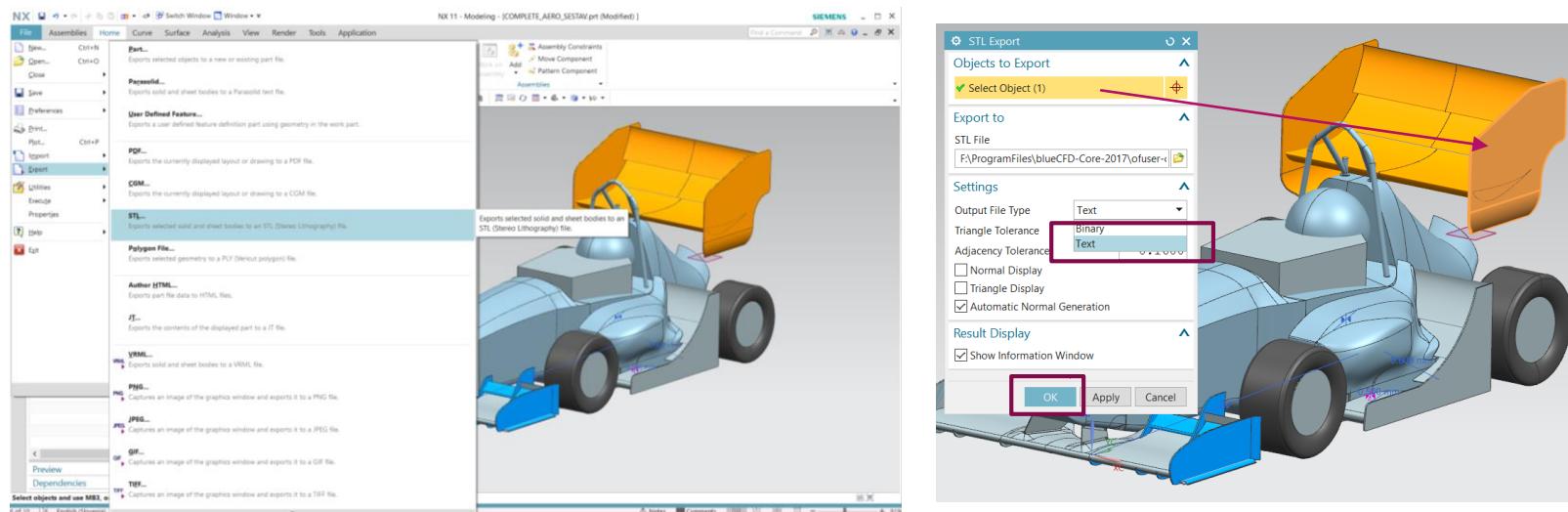


- Odreži spodnji del stranic, ki držijo zadnje krilo



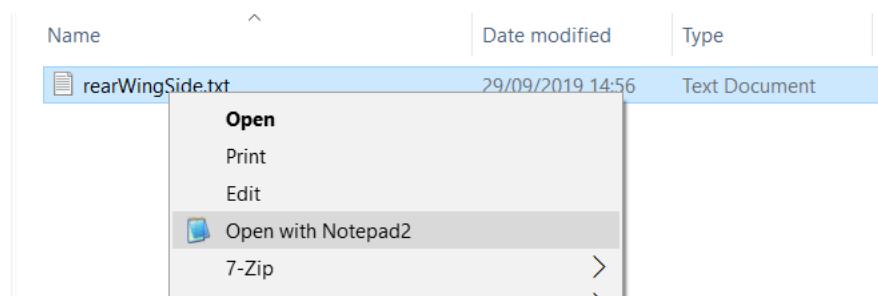
# Izvoz STL datoteke

- ▶ Datoteka mora biti v ASCII (text) formatu (da jo lahko beremo mi in OpenFOAM)
- ▶ Naloga 2: izvozi stl datoteko leve stranice zadnjega krila iz NX v OFkurz\simulacije\_naloge\naloga\_formulaCFD, pojmenuj jo rearWingSide



# Izvoz STL datoteke

- ▶ Vse površine, ki smo jih zapisali v STL datoteko lahko poimenujemo, kar omogoča nastavljanje robnih pogojev. To naredimo tako, da STL datoteko odpremo z Notepad2 in za vse besede solid zapišemo ime površine
- ▶ Naloga 3: poimenuj izvoženo površino!



```
* rearWingSide.txt - Notepad2
File Edit View Settings ?
File Open Save Save as Find Replace Options Encoding Encoding Help
1 solid rearWingSide
2 facet normal +1.2430552E-02 -6.6990526E-01 -7.4234252E-01
3 outer loop
4 vertex +4.2855547E+02 +2.0071695E+03 +6.1372540E+02
5 vertex +4.2848088E+02 +2.0071266E+03 +6.1376291E+02
6 vertex +4.2843805E+02 +2.0071674E+03 +6.1372540E+02
7 endloop
8 endfacet
9 facet normal -7.2165065E-01 -1.2442039E-01 +6.8098450E-01
10 outer loop
11 vertex +4.2649148E+02 +2.0091575E+03 +6.1821641E+02
12 vertex +4.2395856E+02 +2.0115665E+03 +6.1597237E+02
13 vertex +4.2366156E+02 +2.0108256E+03 +6.1552227E+02
14 endloop
15 endfacet
```

A screenshot of Notepad2 showing the content of the file 'rearWingSide.txt'. The file contains an STL-like code. The first line is 'solid rearWingSide'. This is followed by a series of 'facet' definitions, each starting with 'facet normal' and ending with 'endfacet'. The vertices for each facet are listed in a loop, with 'vertex' followed by three coordinates. The file ends with another 'endfacet' line. The entire code is numbered from 1 to 15.

# Izgradnja mreže s snappyHexMesh

- ▶ Vse datoteke združimo v eno datoteko:  
formula.stl in jo kopiramo v mapo  
constant/triSurface
- ▶ Izgradimo gabaritno mrežo z blockMesh
- ▶ Izgradimo priležno mrežo s snappyHexMesh

# Izgradnja mreže s snappyHexMesh

## ► Naloga 4:

1. S pomočjo raziskovalca zamenjaj obstoječi stl stranice zadnjega krila (\simulacije\_naloge\naloga\_formulaCFD\meshingCase\constant\stl\rearWingSide.txt) z novim stl-om površine stranice zadnjega krila:  
\simulacije\_naloge\naloga\_formulaCFD\rearWingSide.txt
2. Odpri blueCFD terminal in se premakni v mapo (blueCFD\ofuser5.0\OFKurz\simulacije\_naloge\naloga\_formulaCFD\meshingCase\)
3. V datoteki naloga\_formulaCFD\ system\decomposeParDict nastavi število razpoložljivih procesorskih jeder X:  
numberOfSubdomains X;
4. Daj pravice za izvajanje, pisanje in branje skripti buildMeshPar:  
\$ chmod 700 buildMeshPar
5. Poženi skripto buildMeshPar (traja približno 10 min):  
\$ ./buildMeshPar

# Naloži mrežo na simscale strežnik

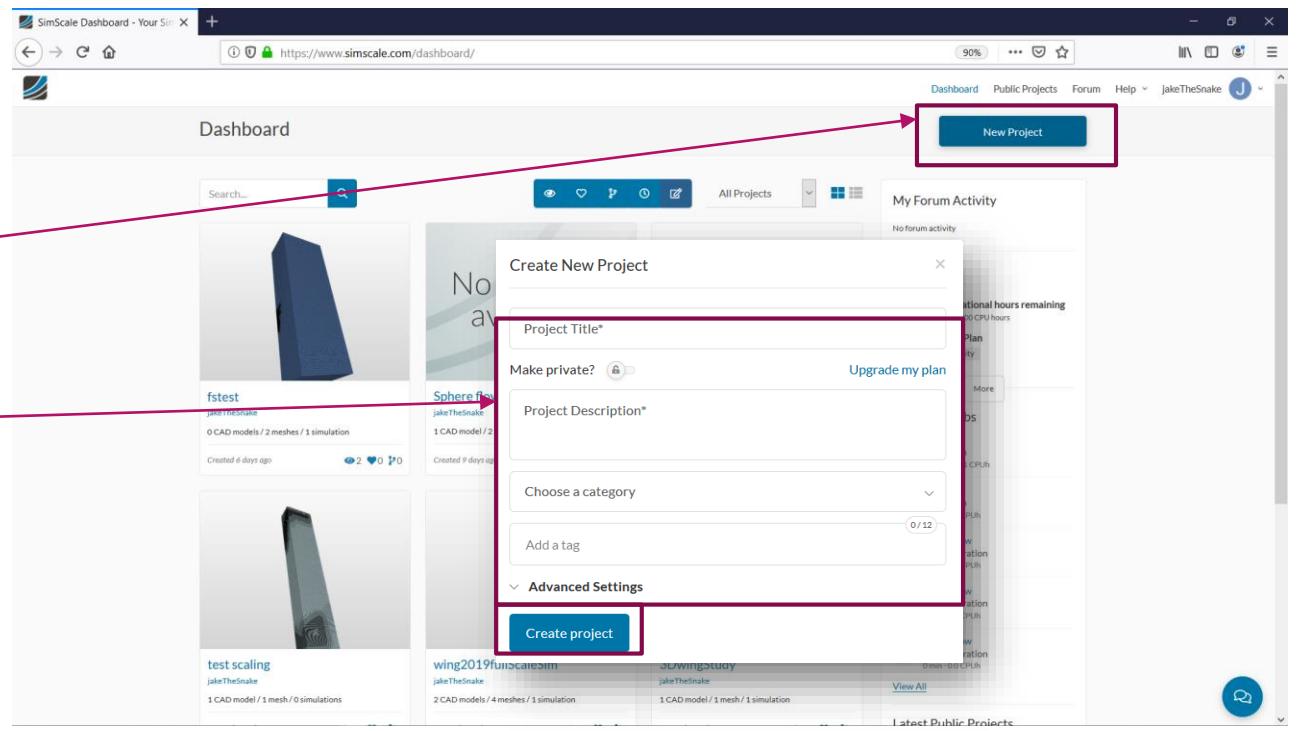
- ▶ Simscale je spletna stran, ki ponuja zastonjske ure računskega časa, v ozadju pa poganja OpenFOAM. Mi bomo izkoristili njihov vmesnik za nastavitev in pogon simulacije.
- ▶ Naloga 5:
  1. Ustvari si račun na Simcale: <https://www.simscale.com/>



# Naloži mrežo na simscale strežnik

## ► Naloga 6: ustvari nov Simscale project:

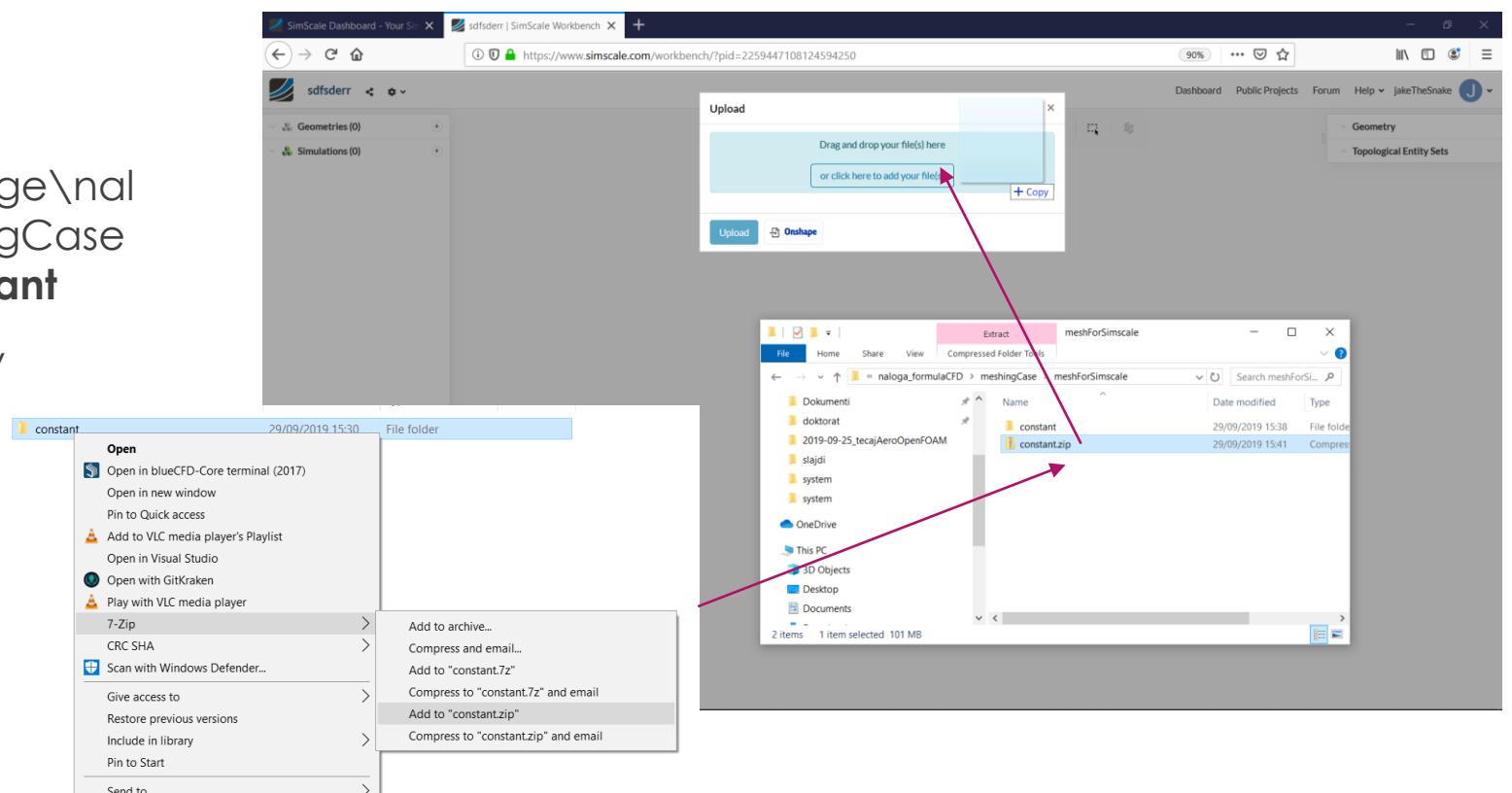
1. Prijavi se v svoj račun
2. Ustvari nov project:  
“New Project”
3. Izpolni osnovne podatke o projektu in ustvari projekt
4. Projekt je javno dostopen!



# Naloži mrežo na simscale strežnik

## ► Naloga 7: Naloži mrežo

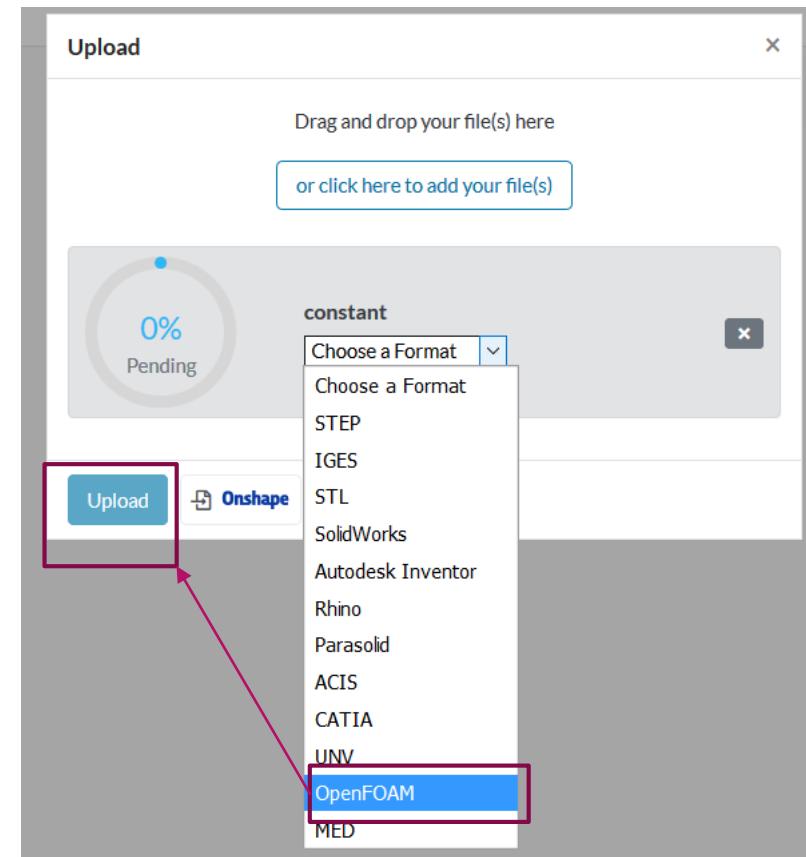
1. Zazipaj mapo:  
...OFKurz\simulacije\_naloge\naloga\_formulaCFD\meshingCase  
**\meshForSimscale\constant**
2. Zazipano mapo potegni v okvirček "Upload"



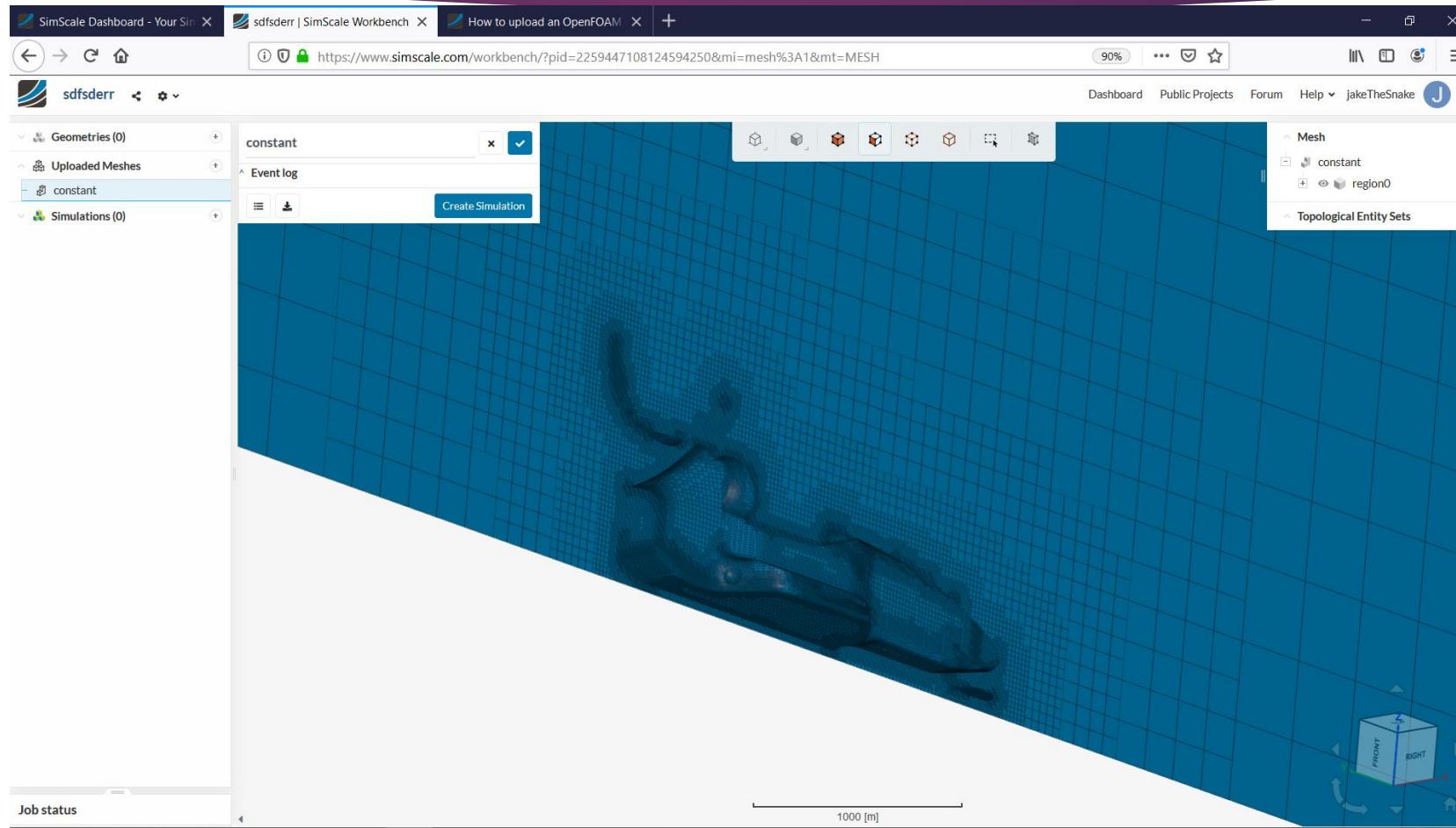
# Naloži mrežo na simscale strežnik

## ► Naloga 7: Naloži mrežo

1. Zazipaj mapo:  
...OFKurz\simulacije\_naloge\naloga\_formulaCFD\meshingCase\meshForSimscale\constant
2. Zazipano mapo potegni v okvirček "Upload"
3. Izberi format OpenFOAM in naloži mrežo s pritiskom na "Upload" (nalaganje se na 99% za nekaj časa ustavi, kar mirno)



# Naloži mrežo na simscale strežnik



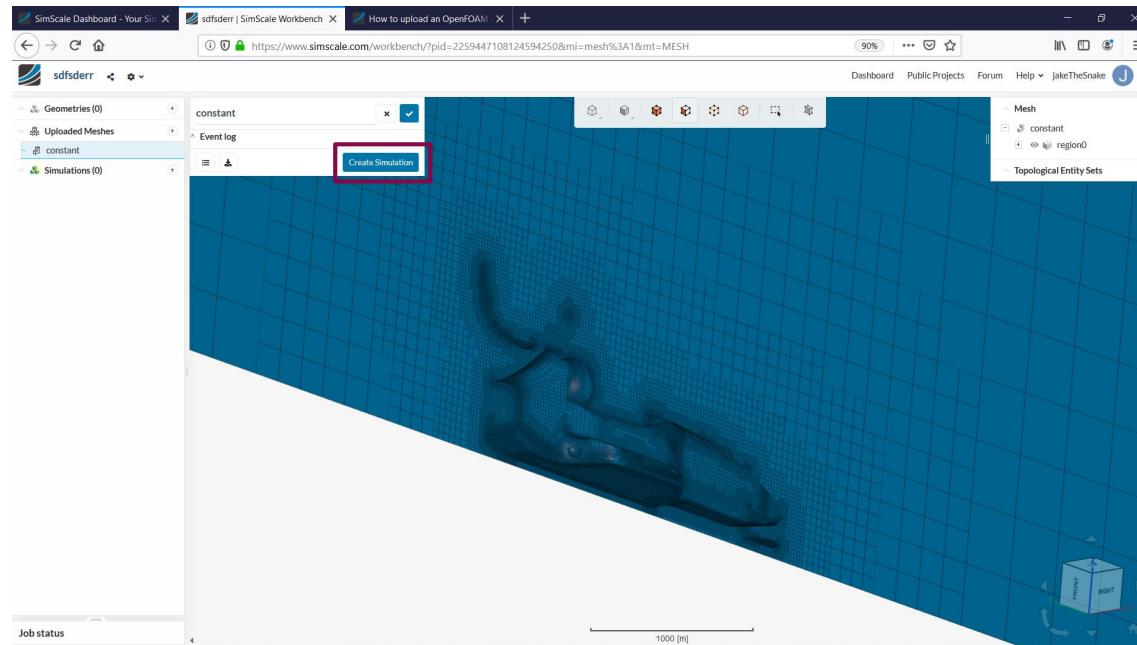
# Nastavi simulacijo

► Nastaviti moramo:

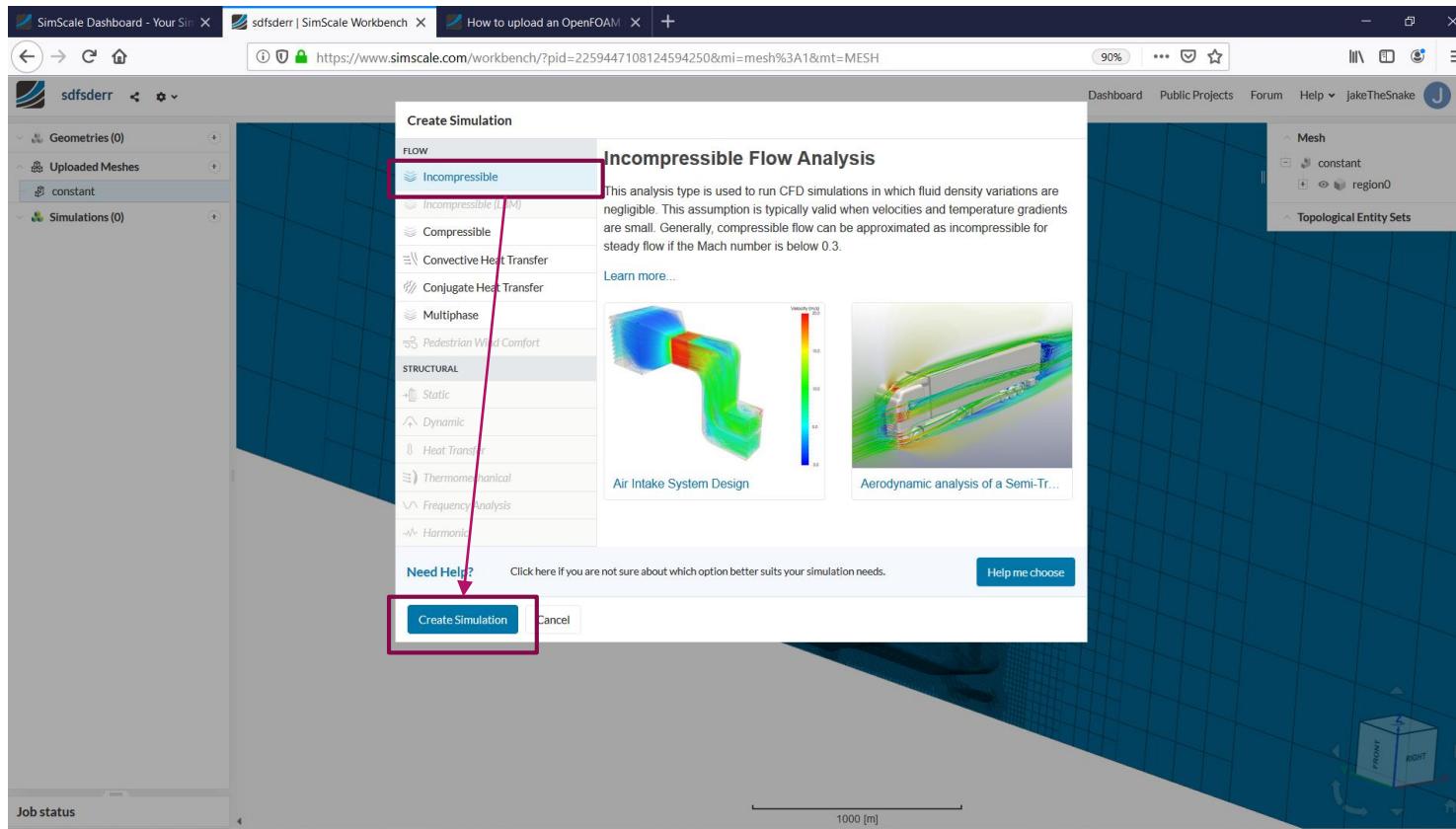
1. Fizikalni model
2. Stacionarni/tranzientni tip simulacije
3. Materialne lastnosti (zrak)
4. Robne pogoje
  1. Inlet
  2. Outlet
  3. Symmetry
  4. Atmosphere
  5. Ground
  6. Formula
  7. Kolesa

# Nastavi simulacijo

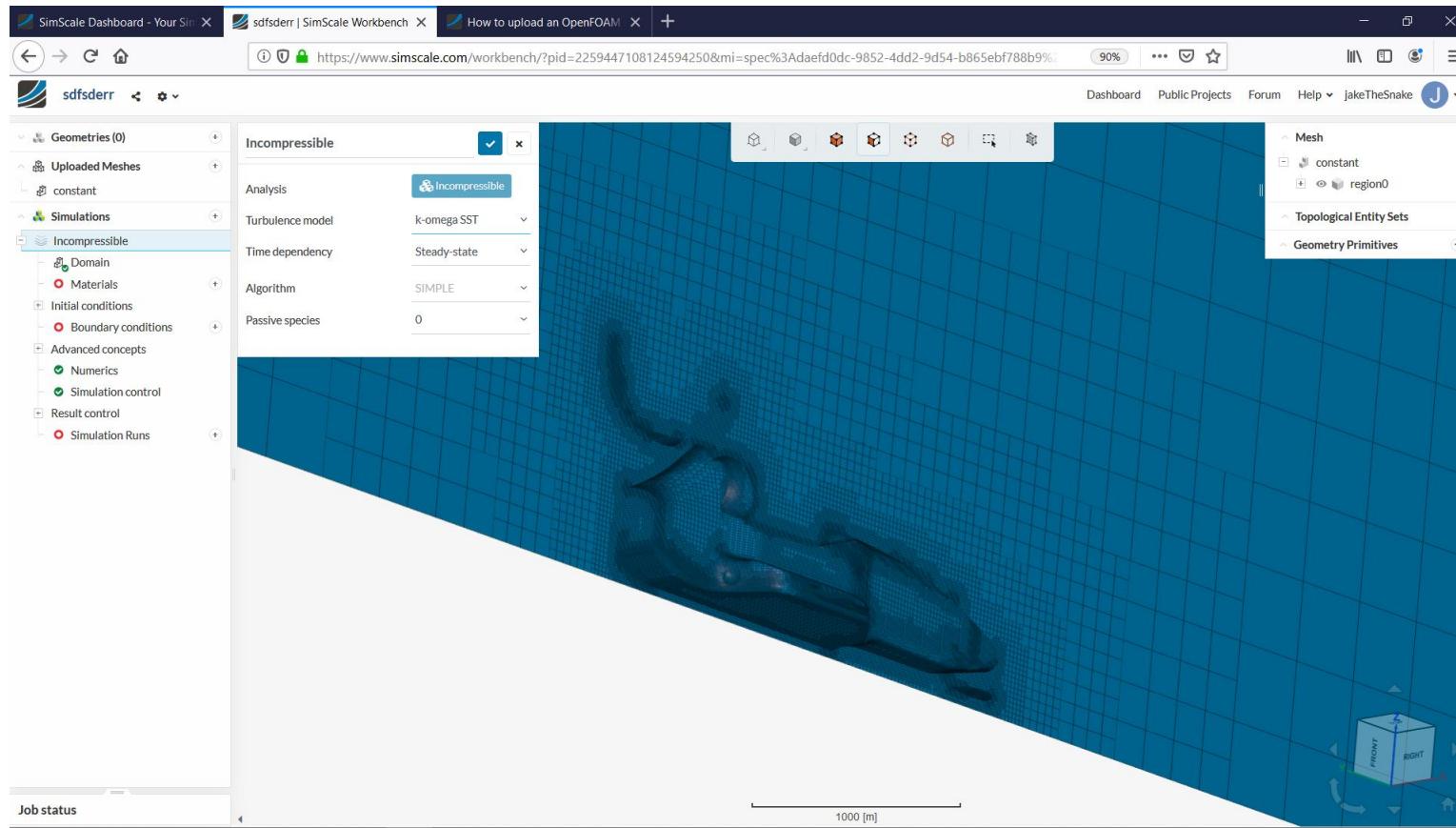
- ▶ Naloga 8: Nastavi simulacijo!
  - ▶ Sledi slikovnemu gradivu



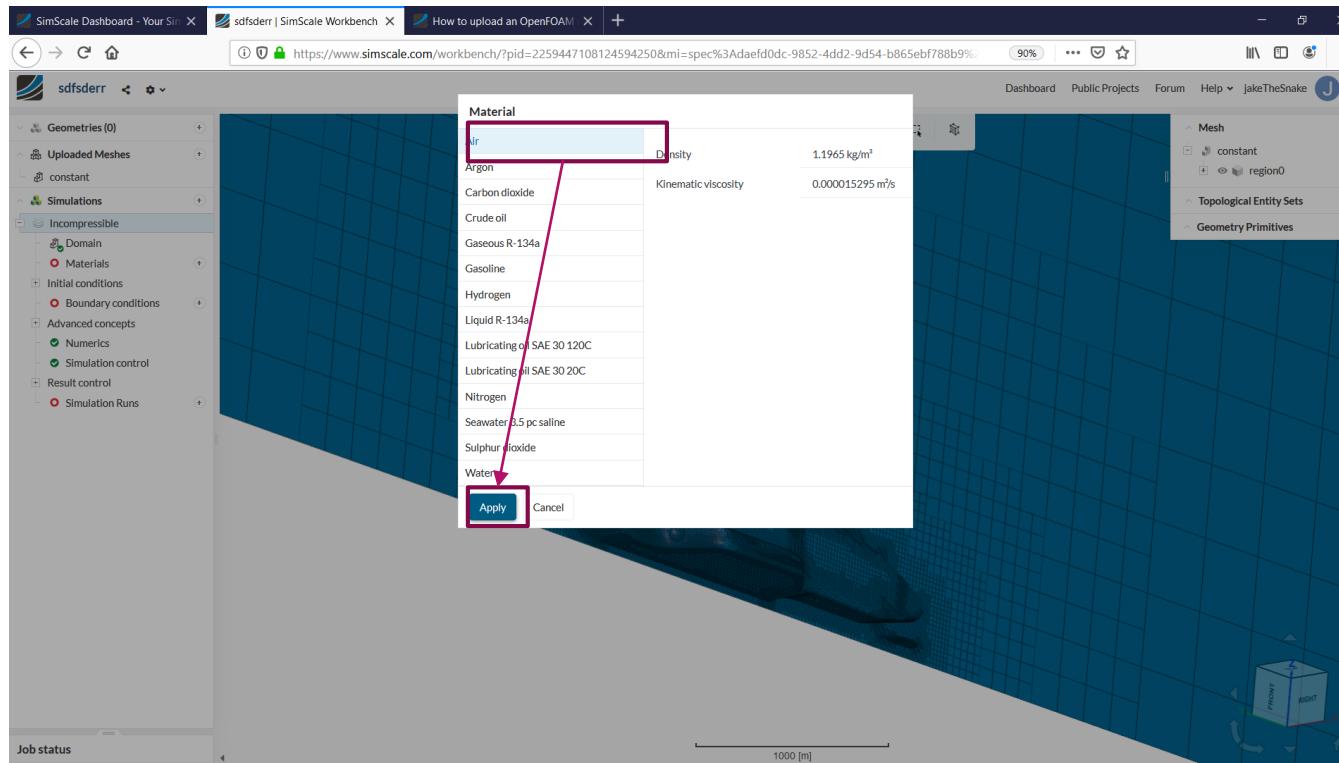
# Nastavi simulacijo



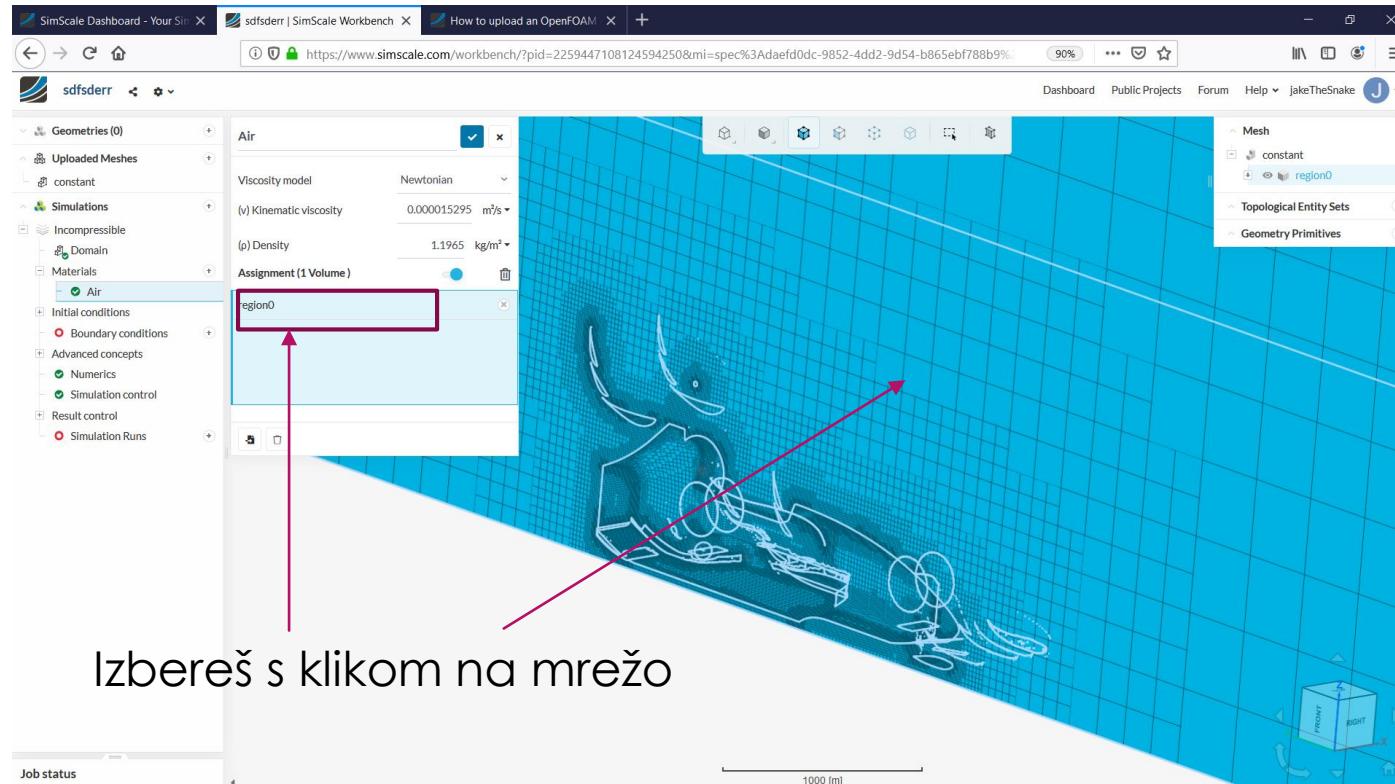
# Turbulenčni model, ustaljenost



# Materialne lastnosti



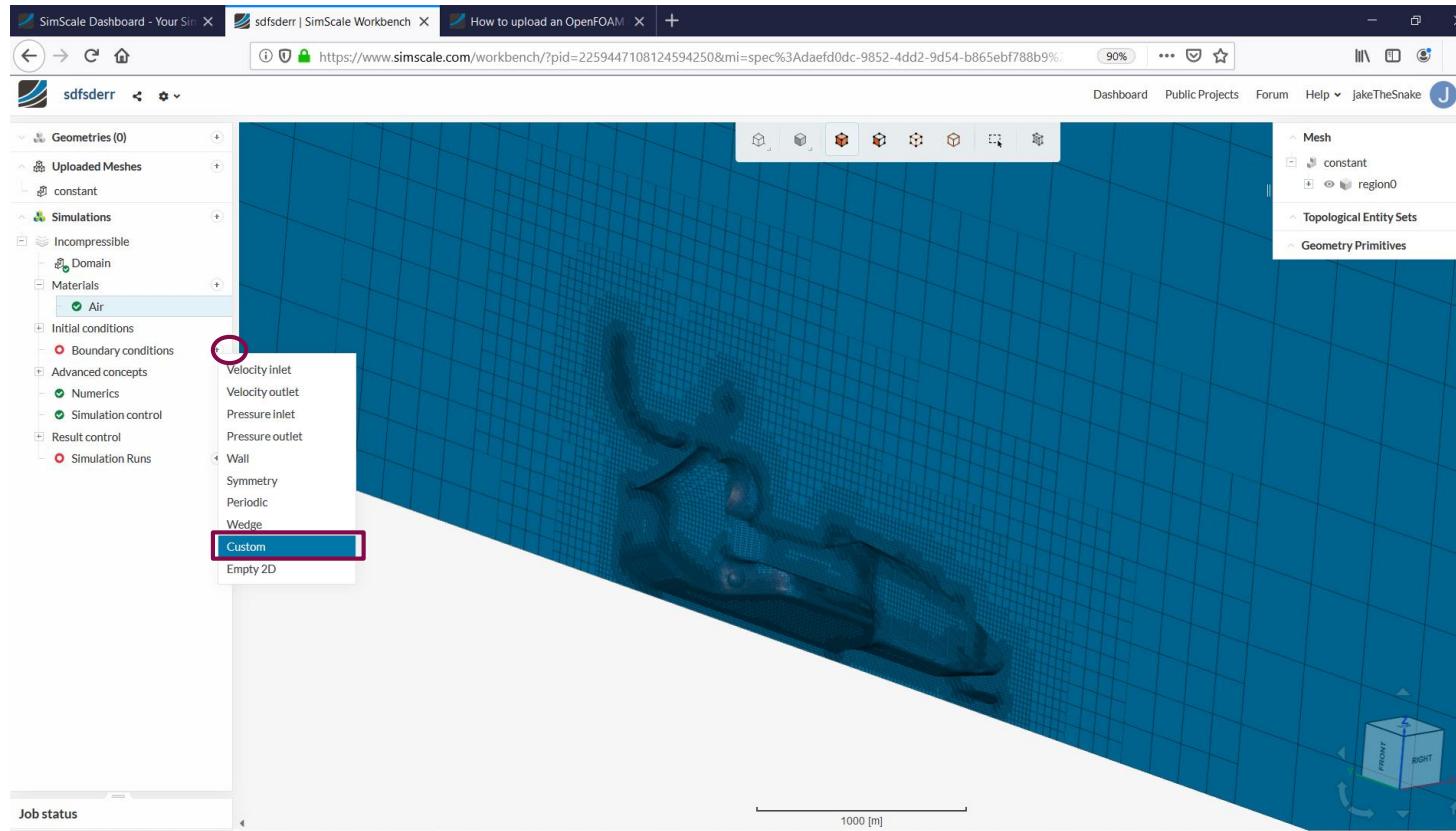
# Materialne lastnosti



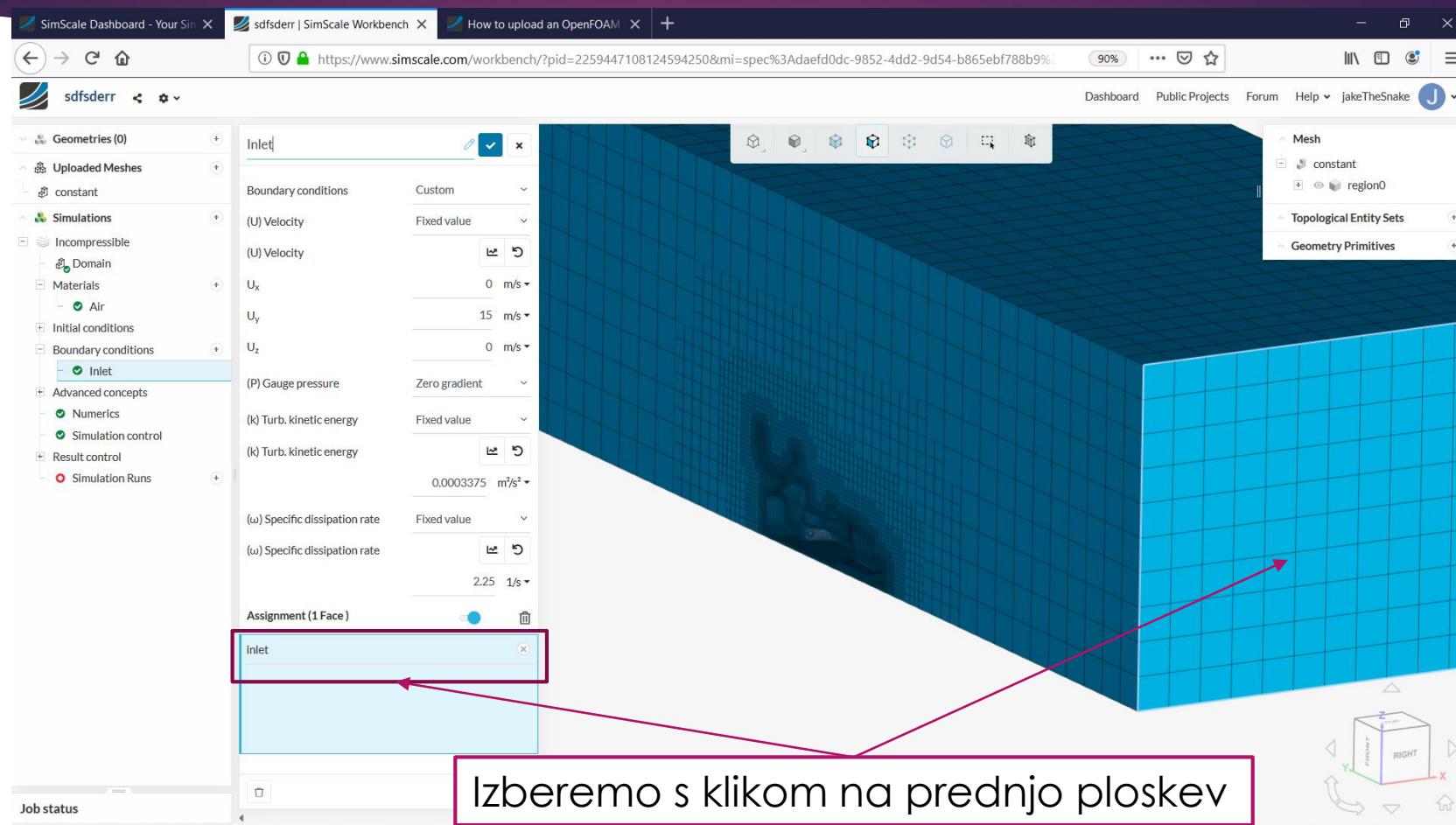
# Robni pogoji

- ▶ Fixed value
- ▶ Fixed gradient
- ▶ Mešani, kompleksni ...
- ▶ Robne pogoje nastavimo tako, da na meji predpišemo takšno vrednost hitrosti, tlaka, turb. Kin. energije in disipacije, kakršna v resnici nastopi:
  - ▶ Zidovi: hitrosti = hitrosti zidu, tlak = zeroGradient, turbulence: stenske funkcije
  - ▶ Simetrija: simetrija
  - ▶ Inlet: hitrosti = konstantna hitrost vožnje, tlak = zeroGradient, turbulence: značilne vrednosti za vetrovnik
  - ▶ Outlet: tlak = tlak okolice, hitrosti = zeroGradient, turbulenca = zeroGradient

# Robni pogoji - inlet



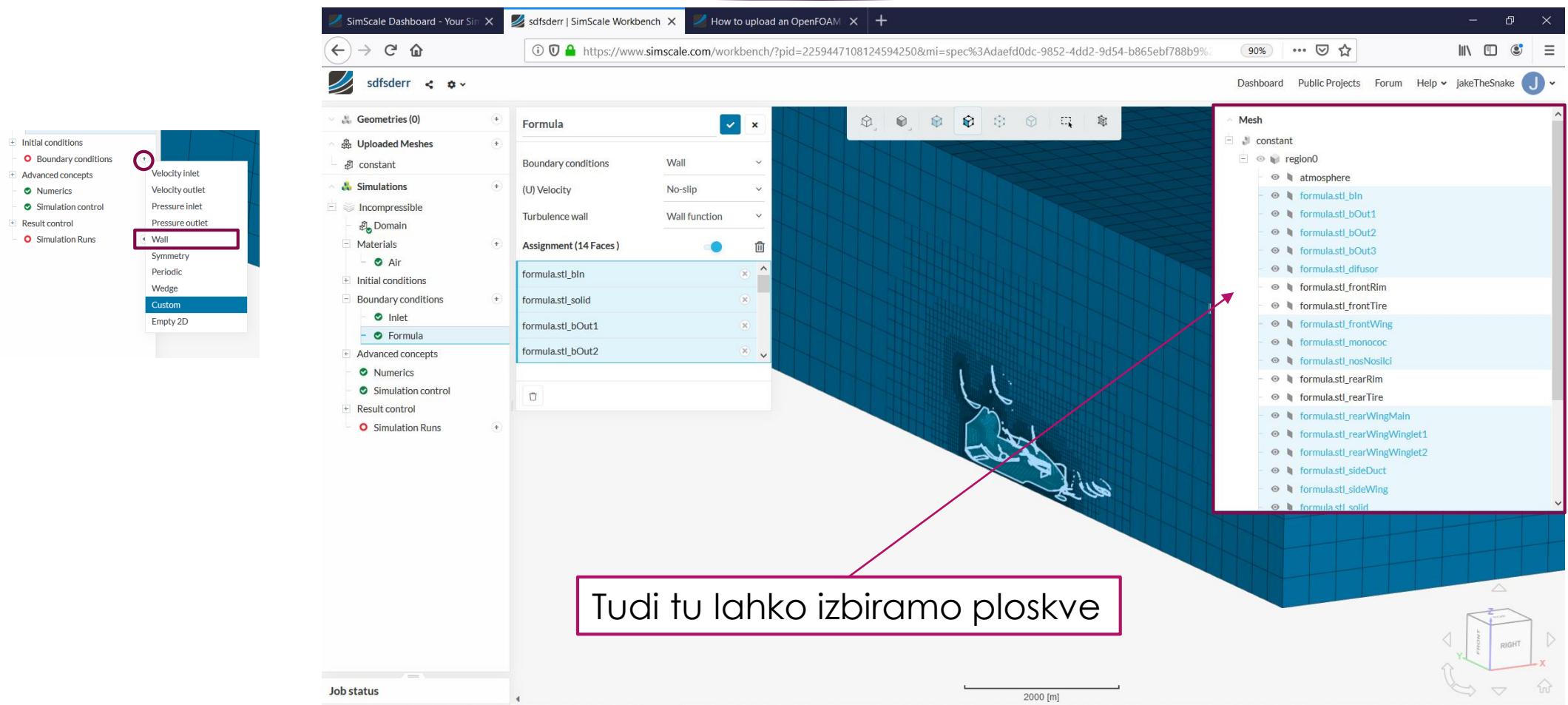
# Robni pogoji - inlet



# Robni pogoji - inlet

- ▶ Izračun kinetične turbulenčne energije na vstopu v domeno:
  - ▶ Keyword: turbulence kinetic energy of free stream
- ▶ Iskane spremenljivke k in omega izračunamo iz ocenjene intenzitete turbulence (za mirujoč zrak skozi katerega vozi formula je to okoli 0,05%), in razmerja turbulentne in dinamične viskoznosti, ki za mirujoč zrak znaša od 1 do 10.
- ▶ Dobra referenca npr.: <http://jullio.pe.kr/fluent6.1/help/html/ug/node178.htm>

# Robni pogoji – formula, zid



# Robni pogoji – formula, zid

- ▶ Hitrost ob zidu je enaka hitrosti zidu
- ▶ Tlak se v smeri normale zidu ob zidu ne spreminja: zeroGradient
- ▶ Turbulenco ob zidu lahko modeliramo na 2 načina:
  - ▶ S pomočjo stenskih funkcij (wallFunction robni pogoj)
    - ▶ **V tem primeru mora biti  $y^+$  oddaljenost centra prve celice ob steni 30-300**
    - ▶ Izdelana mreža, ki jo uporabljamo v tem primeru je primerna za to opcijo
  - ▶ Do laminarne podplasti
    - ▶ **V tem primeru mora biti  $y^+$  oddaljenost centra prve celice ob steni ~1**
    - ▶ Za to opcijo bi morali celice ob steni dirkalnika močno zgostiti z uporabo plastenja ("layering")

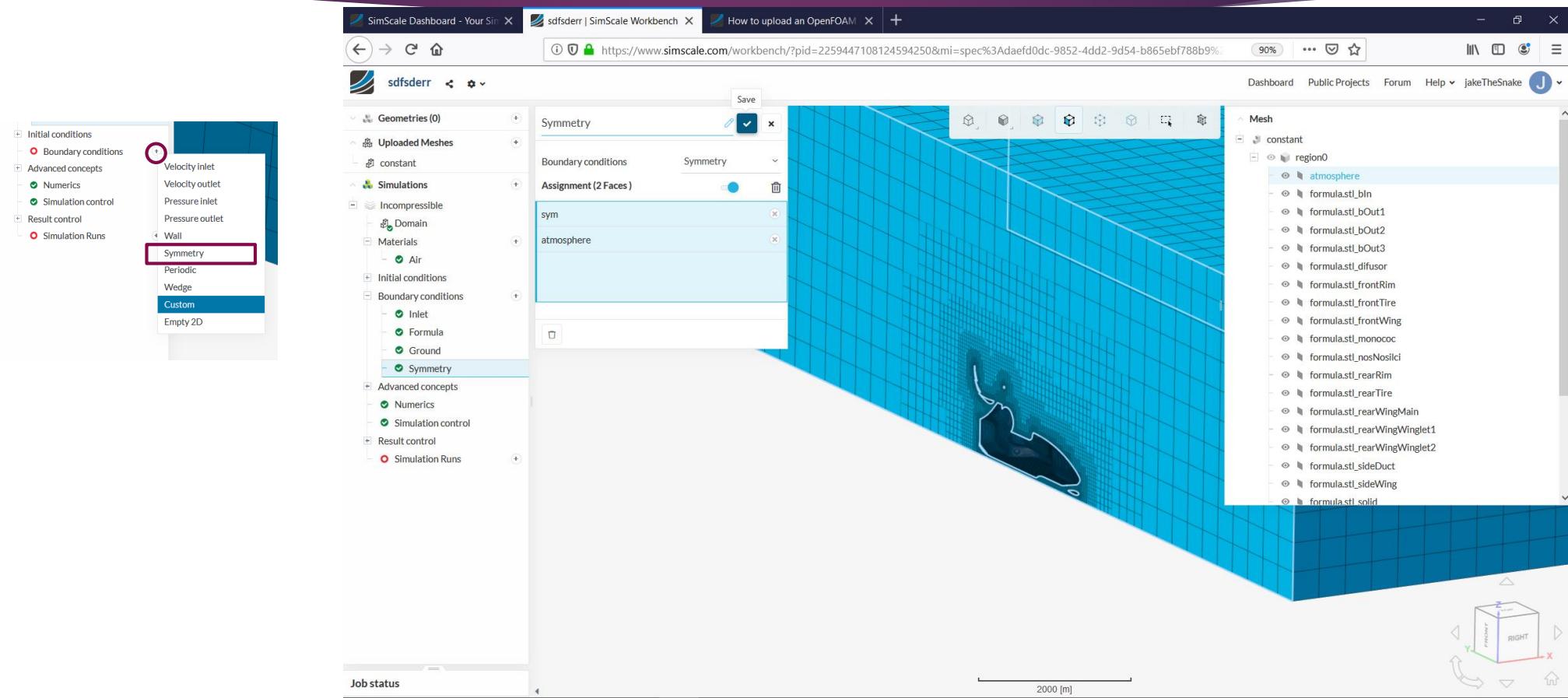
Če tega ne upoštevamo simulacija ni pravilna in se ponavadi tudi sesuje!

# Robni pogoj - tla

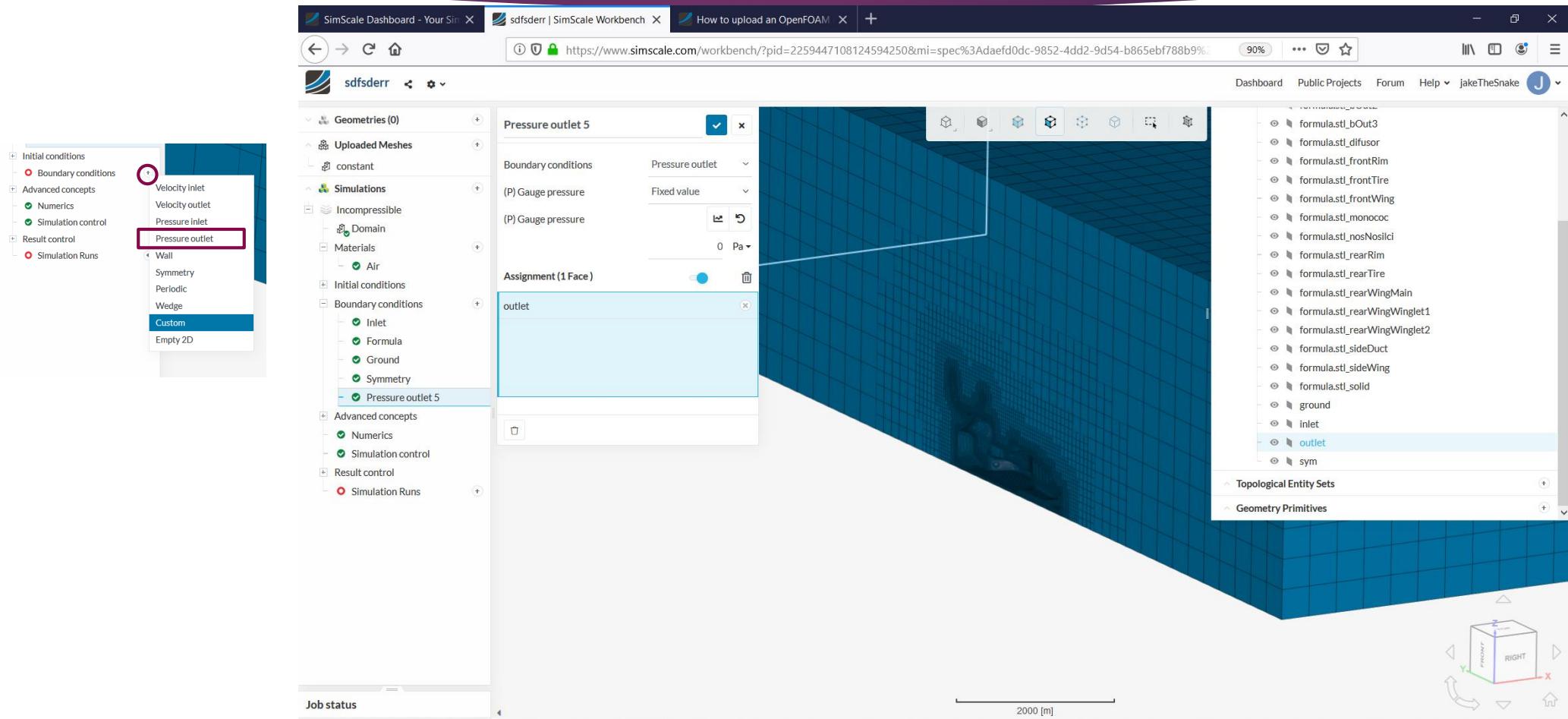
The screenshot shows the SimScale Workbench interface. On the left, the sidebar has a section for 'Boundary conditions' highlighted with a red circle. Under 'Wall', 'Moving wall' is selected and highlighted with a red box. In the main workspace, a 3D model of a racing car is shown with a grid. A specific face of the car's body is highlighted with a red box and labeled 'ground'. The 'Boundary conditions' panel shows 'Moving wall' assigned to this face. The 'Wall function' dropdown is set to 'Formula'. The 'Ux' field is set to 0 m/s, 'Uy' to 15 m/s, and 'Uz' to 0 m/s. The right side of the interface shows a tree view of the simulation setup, including 'Geometries', 'Uploaded Meshes', 'Simulations', 'Advanced concepts', and 'Result control' sections.

Ker je koordinatni sistem fiksiran na dirkalnik se tla glede na ta koordinatni Sistem gibljejo s hitrostjo vožnje.

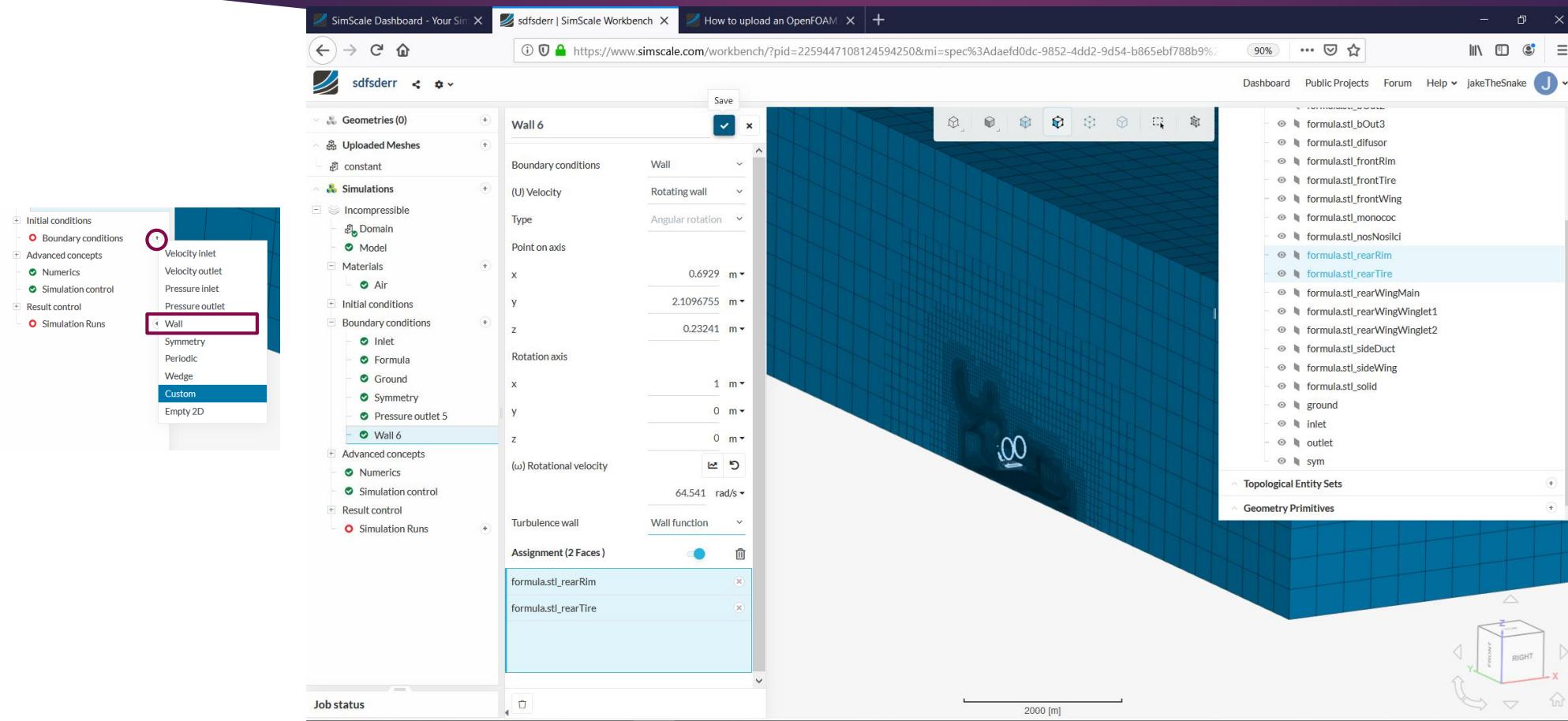
# Robni pogoj – simetrijska ravnina



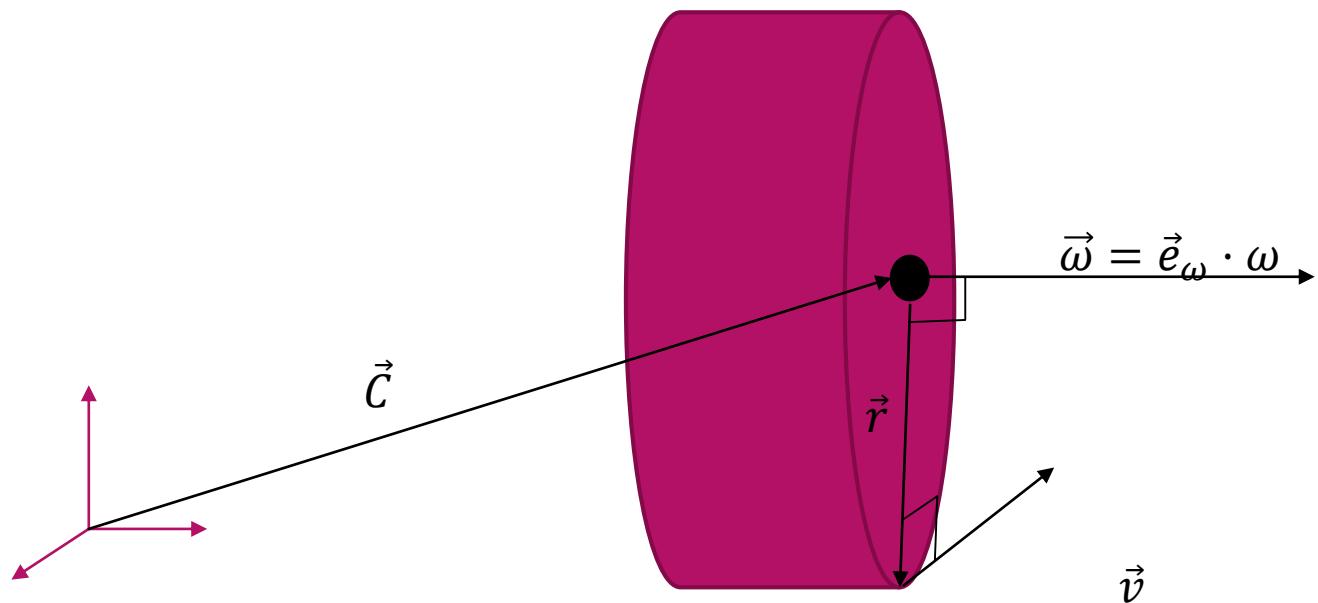
# Robni pogoj - outlet



# Robni pogoj – zadnje kolo



# Robni pogoj – zadnje kolo



$\vec{C}$  point on axis

$\vec{e}_\omega$  rotation axis

$\omega$  rotational velocity

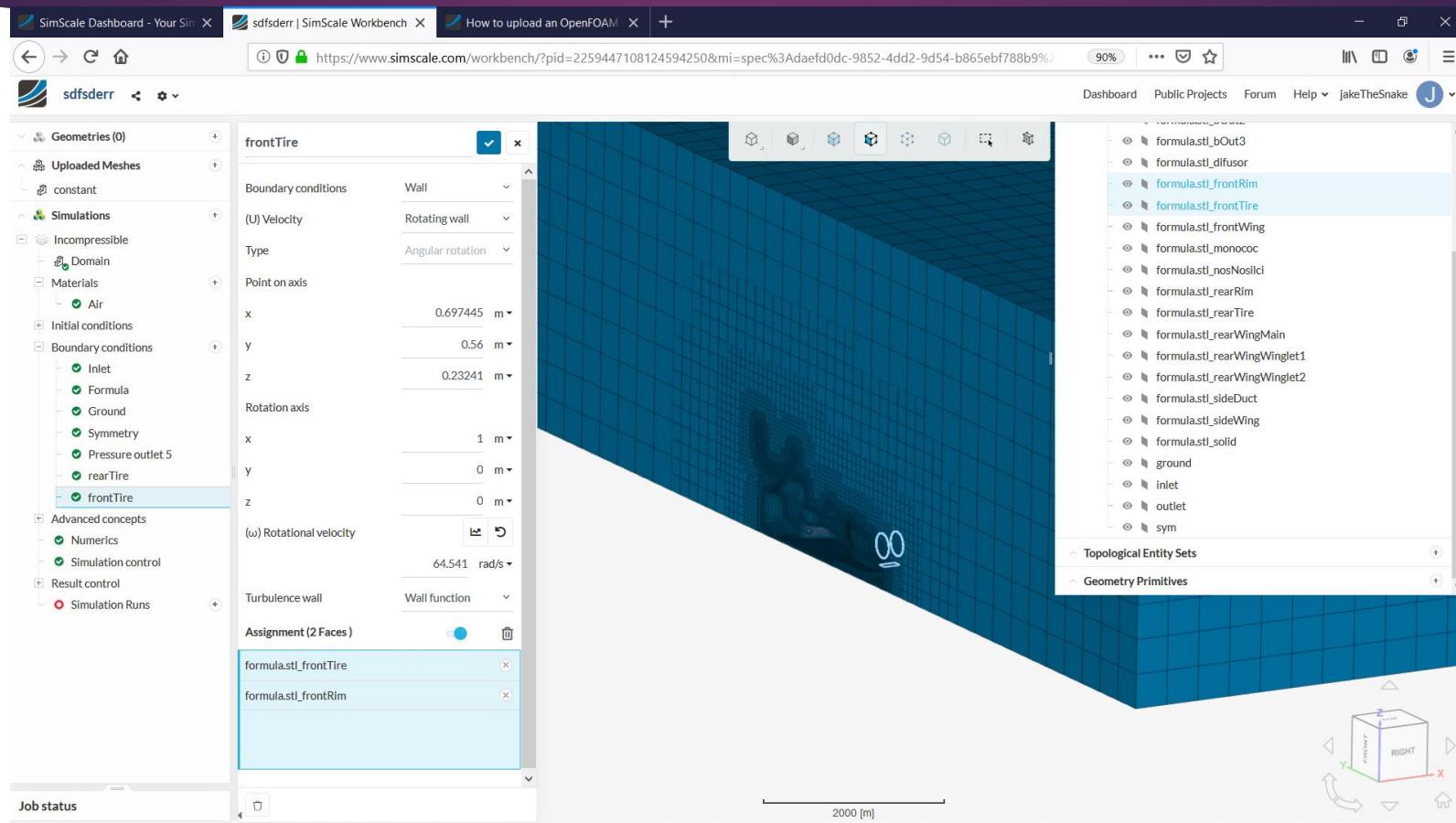
$\vec{r}$  tire half diameter

$\vec{v}$  travel velocity

$$\text{Velja: } |\vec{v}| = \omega \cdot |\vec{r}|$$

$$\text{In posledično: } \omega = |\vec{v}| / |\vec{r}|$$

# Robni pogoj – sprednje kolo



# Nastavitev simulacije

The screenshot shows the SimScale Workbench interface with a project titled "sdfsderr". The main area displays a 3D mesh of a car chassis and rear wheel assembly. On the left, the "Simulation control" panel is open, showing the following configuration:

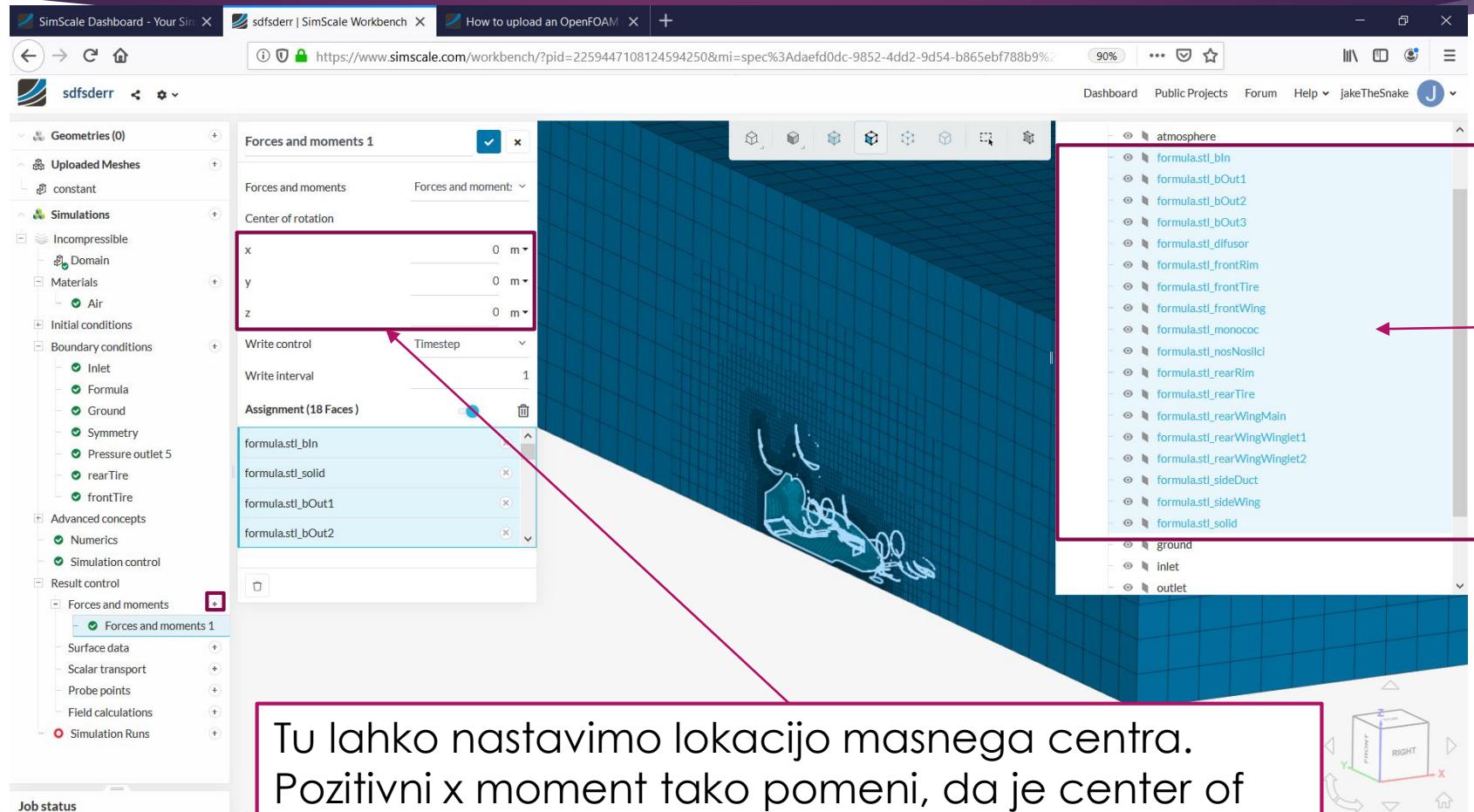
- Start time: 0 s
- End time: 1000 s
- Delta t: 1 s
- Write control: Timestep
- Write interval: 1000
- Number of processors: 16 (highlighted with a red box)
- Maximum runtime: 20000 s
- Potential foam initialization: Enabled
- Decompose algorithm: Scotch

The left sidebar contains a tree view of simulation components:

- Domain
  - Materials: Air, Air
  - Initial conditions
  - Boundary conditions: Inlet, Formula, Ground, Symmetry, Pressure outlet 5, rearTire, frontTire
  - Advanced concepts: Numerics, Simulation control (selected)
  - Result control: Forces and moments, Surface data, Scalar transport, Probe points, Field calculations: yPlus, Simulation Runs
- Job status: Incompressible - Run 1 (Cancelled), Incompressible - Run 3 (Cancelled), Incompressible - Run 2 (Cancelled)

The top right corner shows a small 3D coordinate system with axes labeled FRONT, RIGHT, and UP, and a scale bar indicating 2 [m].

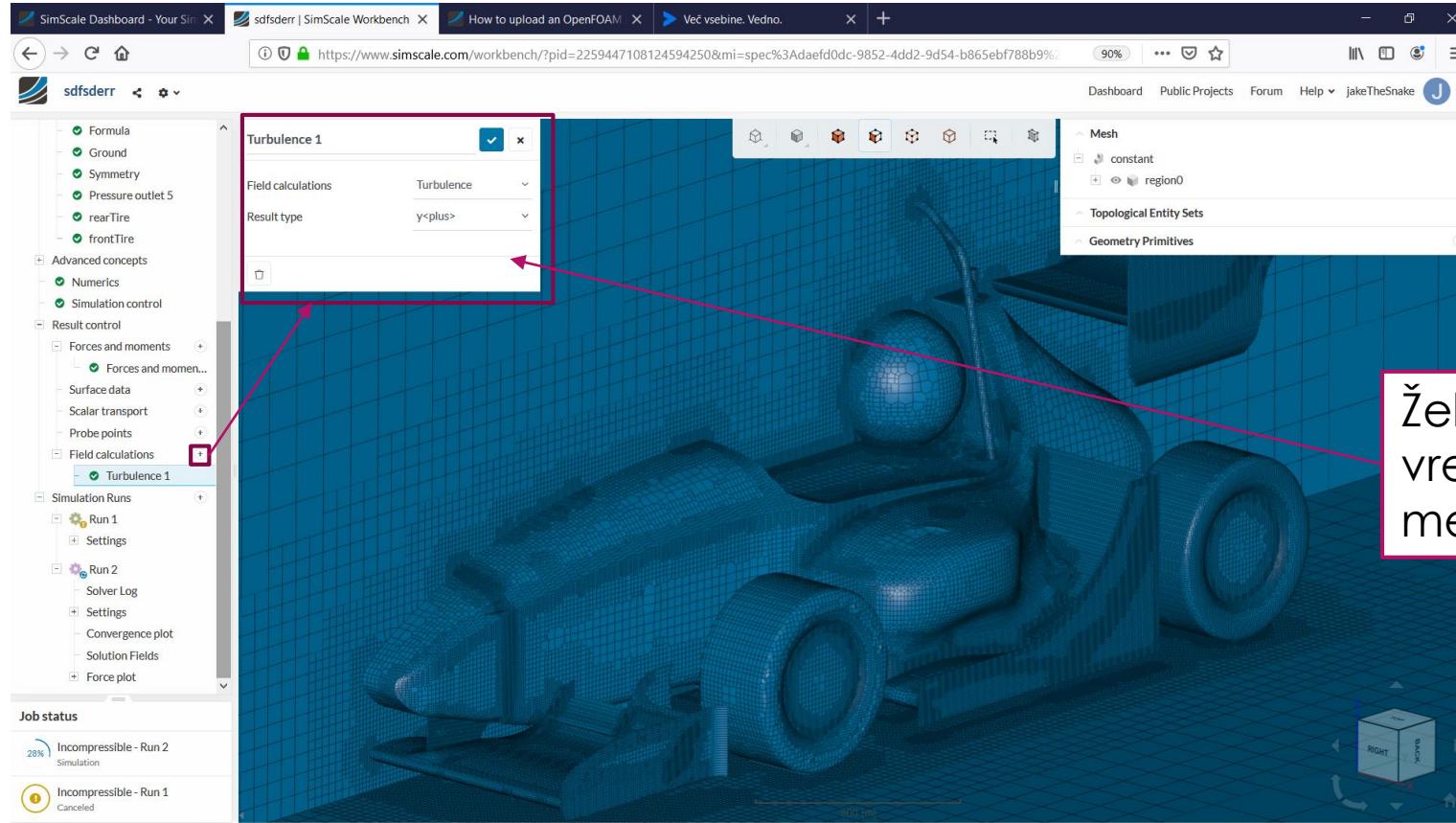
# Nastavitev spremljanja aerodinamičnih sil



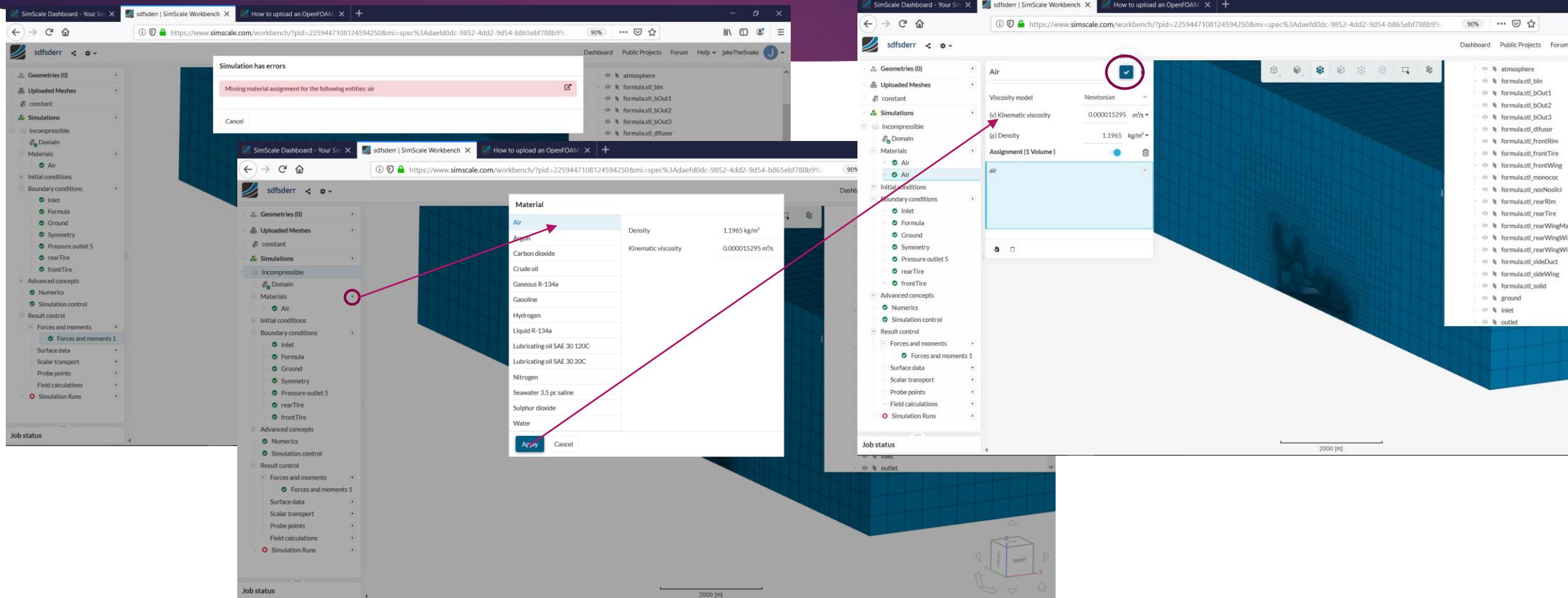
Tu lahko nastavimo lokacijo masnega centra.  
Pozitivni x moment tako pomeni, da je center of  
Pressure pred težiščem, negativni, pa da je za njim

Zanima nas  
sila na vse  
površine  
dirkalnika

# Nastavitev spremljanja $y+$ vrednosti

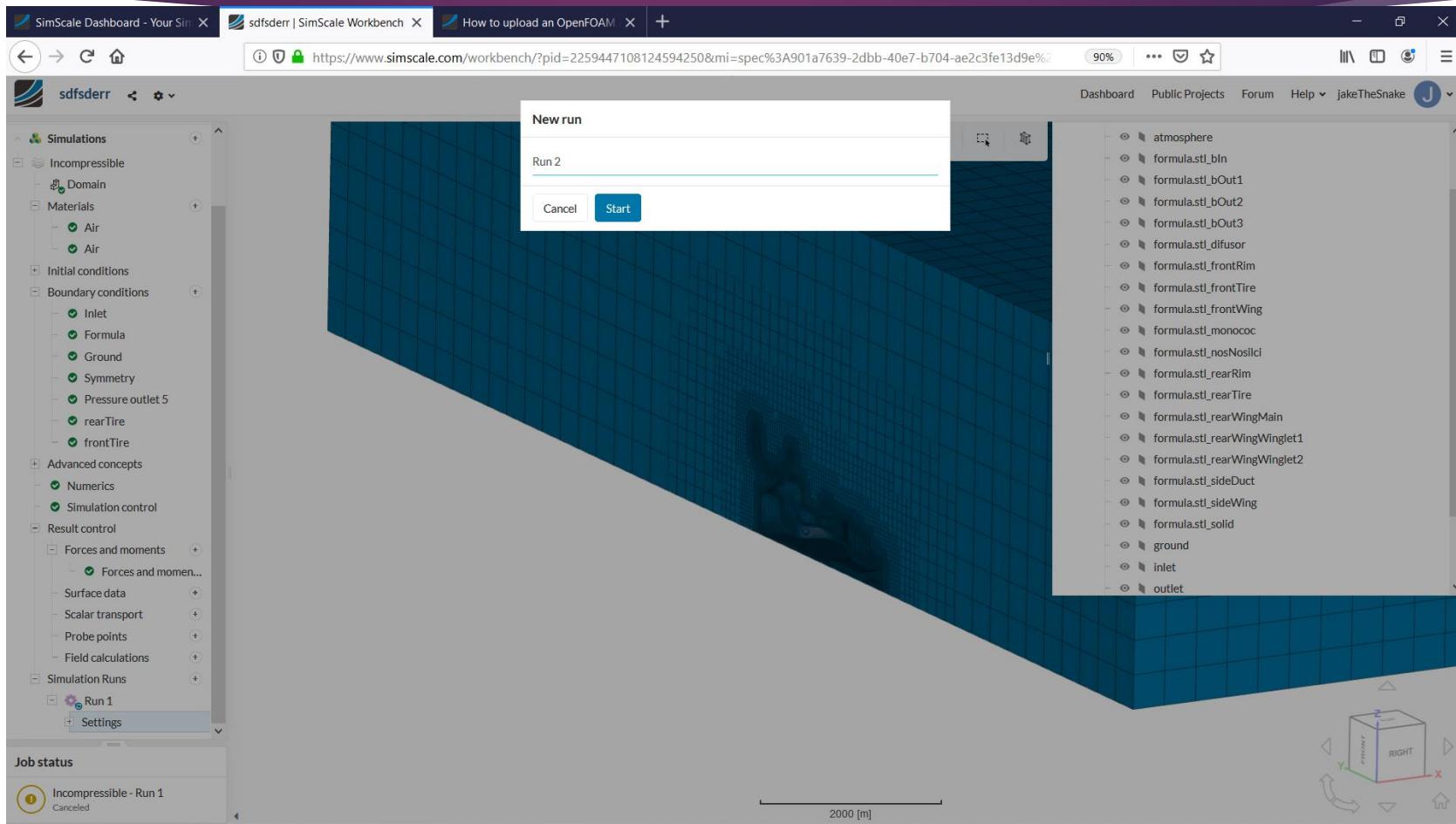


# Nastavitev manjkajoče “air” cone

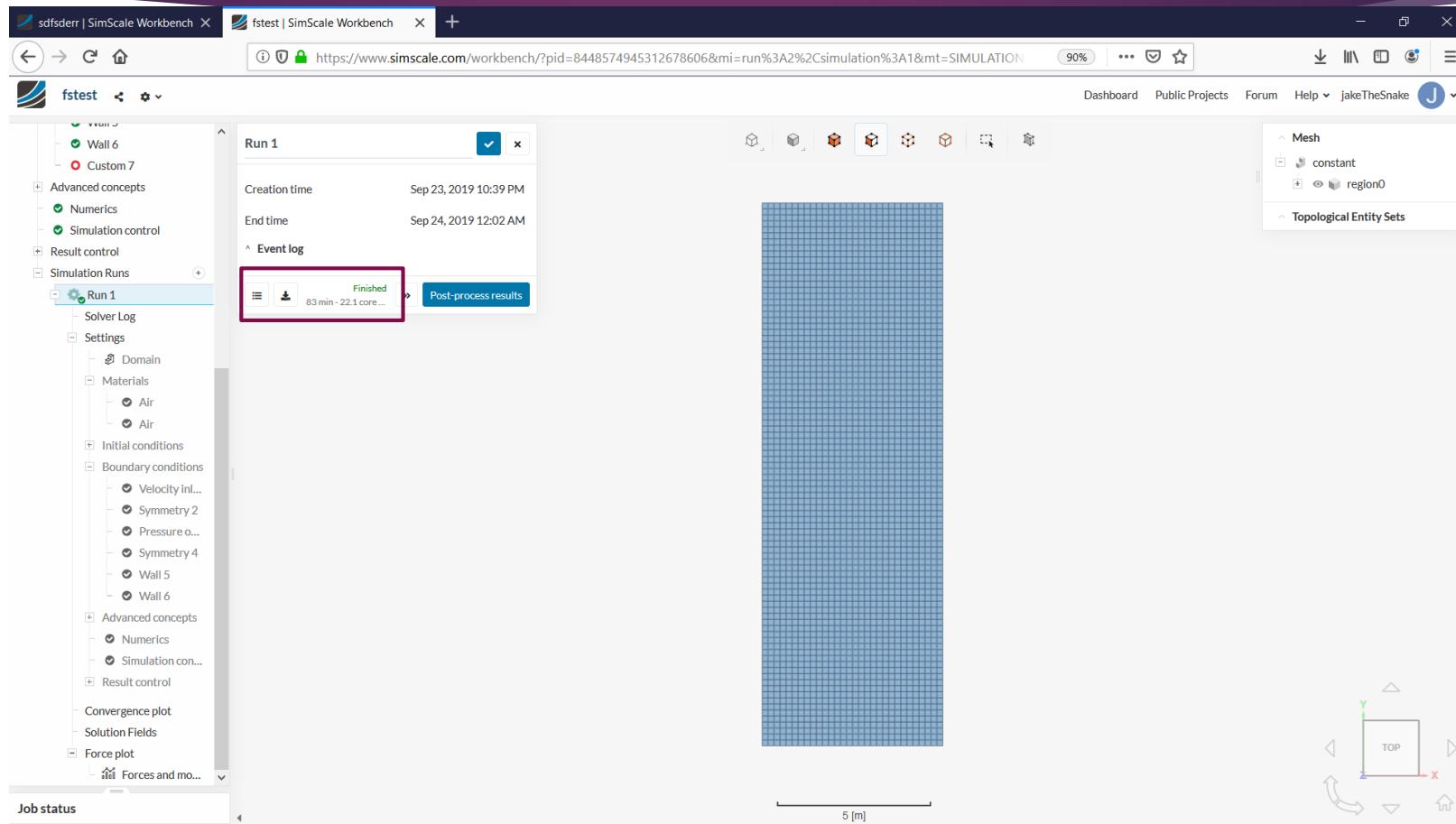


To je bug, vendar rešljiv  
Po zgornjem postopku.

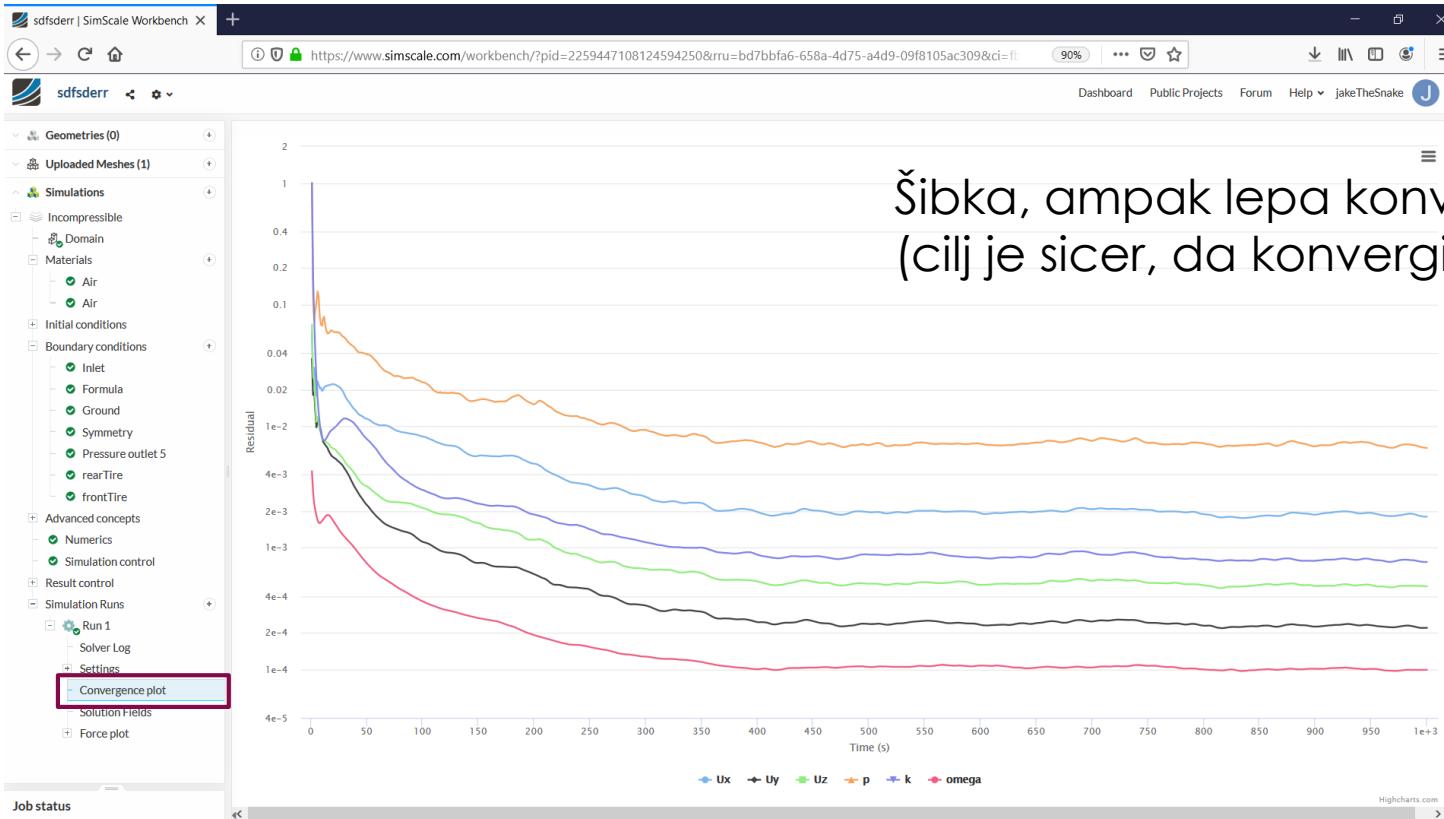
# Zagon simulacije



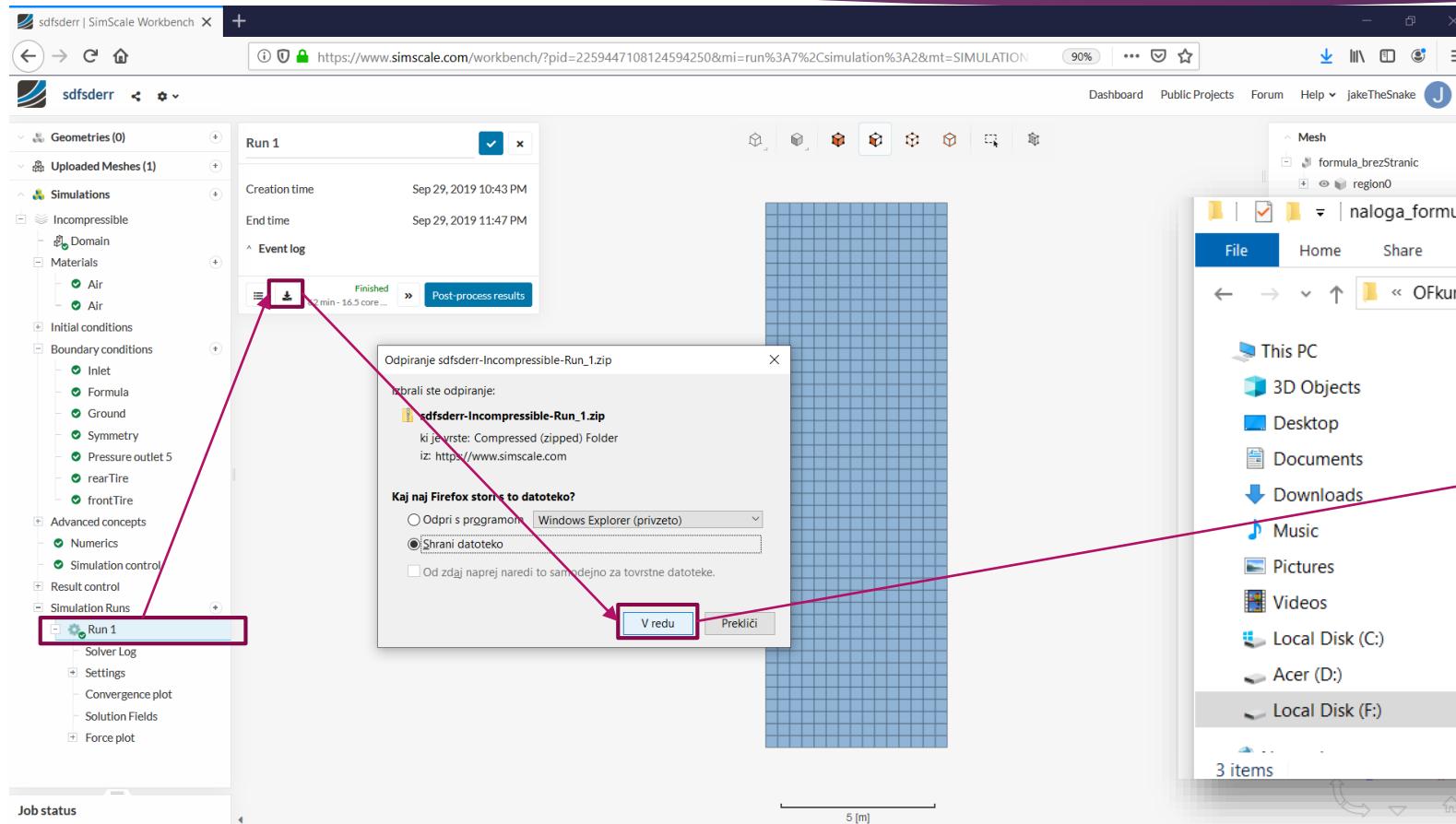
# Zagon simulacije



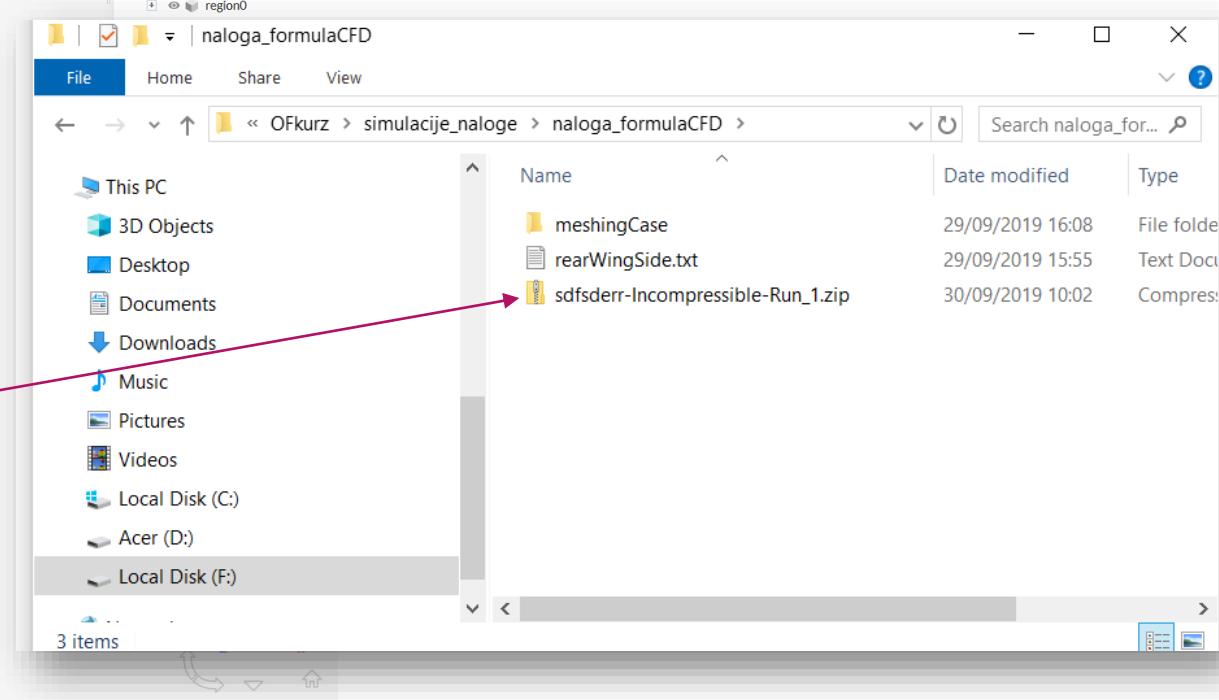
# Po končanem preračunu



# Potegni rezultate na svoj računalnik

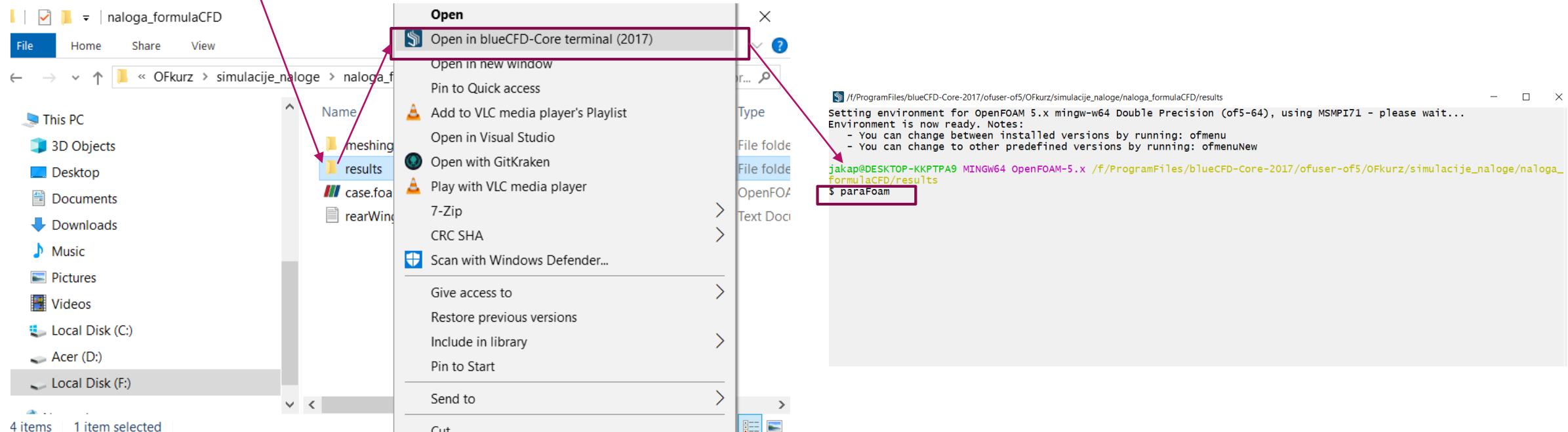


Premakni v mapo  
naloge\_formulaCFD,  
Razširi.

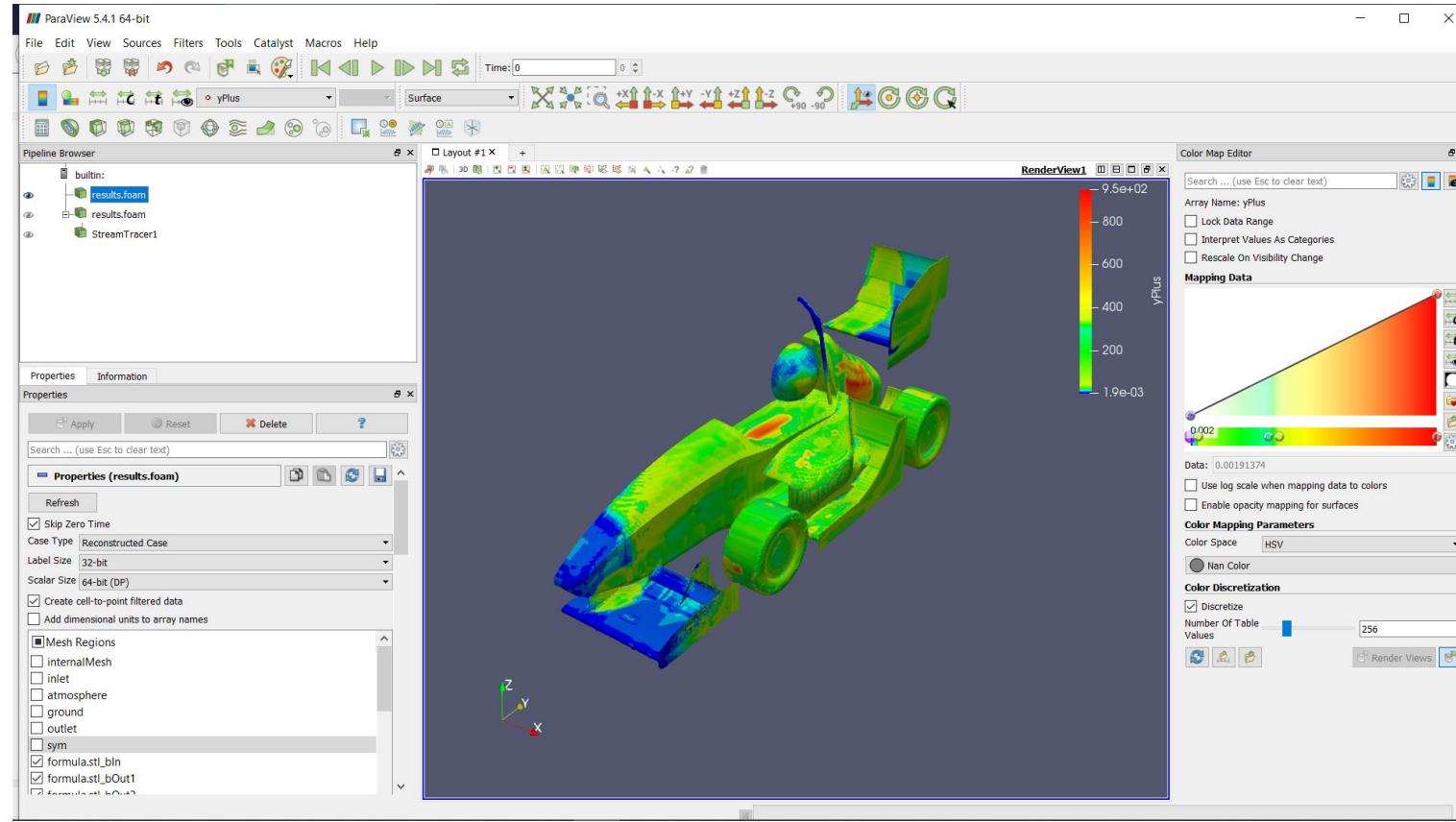


# Pregrelj rezultate s paraview

## Ekstrahirani rezultat



# Preveri $y+$



- ▶  $y+$  ustreza na večini površine dirkalnika
- ▶  $y+$  je premajhen na nosu in prednjem krilu:
  - ▶ Tam bi bilo treba celice povečati
- ▶  $y+$  je prevelik na določenih delih monocoja:
  - ▶ Tam bi bilo treba celice zmanjšati

# Preveri tlačno in tokovni polje

