

Disclaimer

“This offering is not approved or endorsed by OpenCFD Limited, the producer of the OpenFOAM software and owner of the OPENFOAM® and OpenCFD® trade marks.”

Introductory OpenFOAM® Course

From 14th to 18th July, 2014

University of Genoa, DICCA

Dipartimento di Ingegneria Civile, Chimica e Ambientale



Your Lecturer

Joel GUERRERO

joel.guerrero@unige.it



UNIVERSITÀ DEGLI STUDI
DI GENOVA

guerrero@wolfdynamics.com



wolfdynamics

Today's lecture

1. Crash introduction to turbulence modeling

Crash introduction to turbulence modeling

- The majority of engineering flows are turbulent, hence the necessity of modeling turbulence.
- Modeling turbulence is not an easy task. There is no universal turbulence model.
- We are going to mainly deal with wall bounded turbulence. But have in mind that shear free turbulence can also happen.
- Simulating turbulent flows in OpenFOAM® requires selecting a turbulence model, providing initial conditions and boundary conditions, and selecting a near-wall modeling.
- You also need to choose the discretization scheme of the turbulent equations.

Crash introduction to turbulence modeling

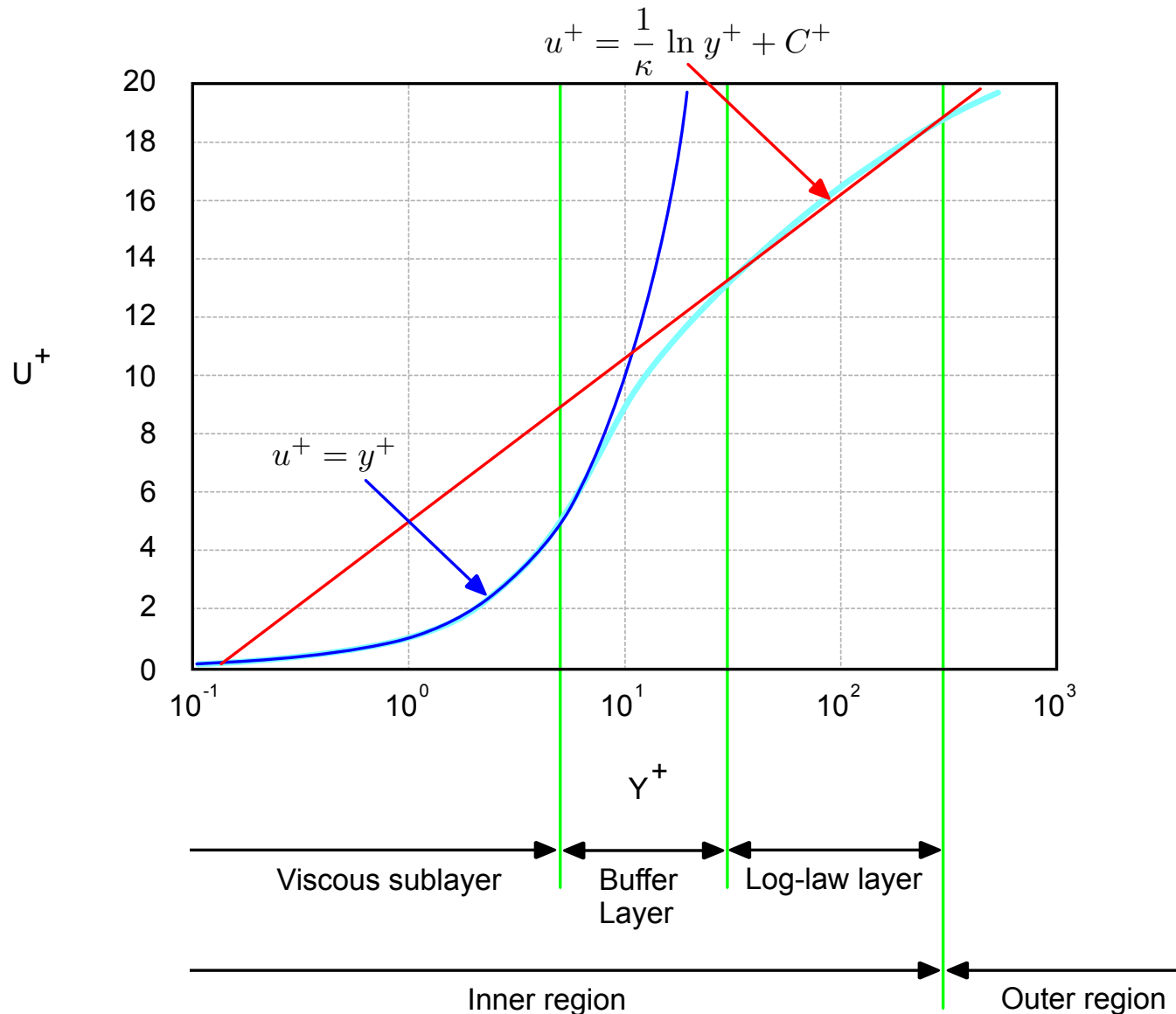
“Essentially, all models are wrong,
but some are useful”

G.E.P Box

- All turbulence models contain some sort of empiricism.
- Equations cannot be derived from fundamental principles.
- Some calibration to observed solutions and intelligent guessing is contained in the turbulence models.
- A lot of uncertainty is involved.

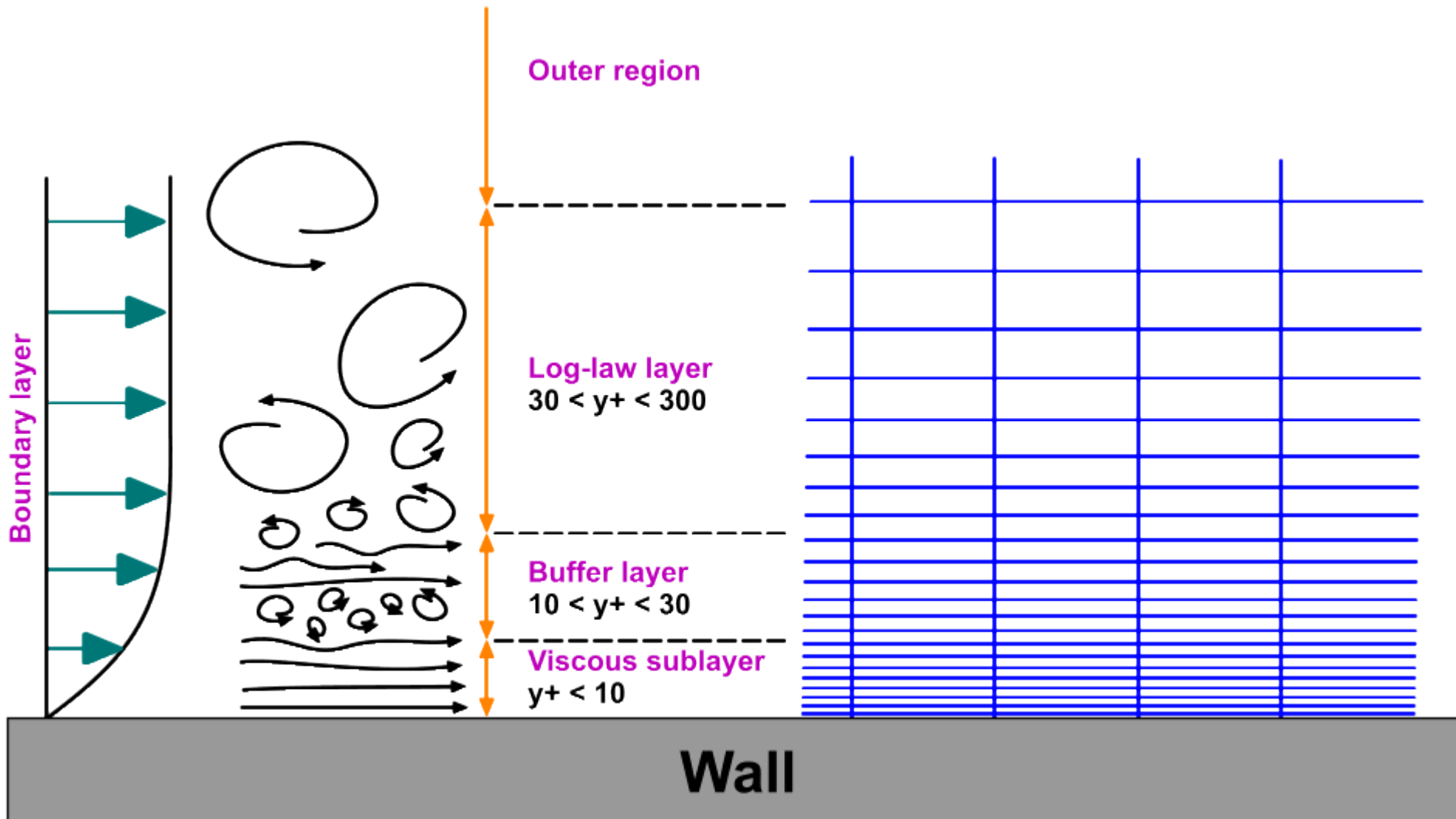
Crash introduction to turbulence modeling

Turbulence near the wall - Law of the wall



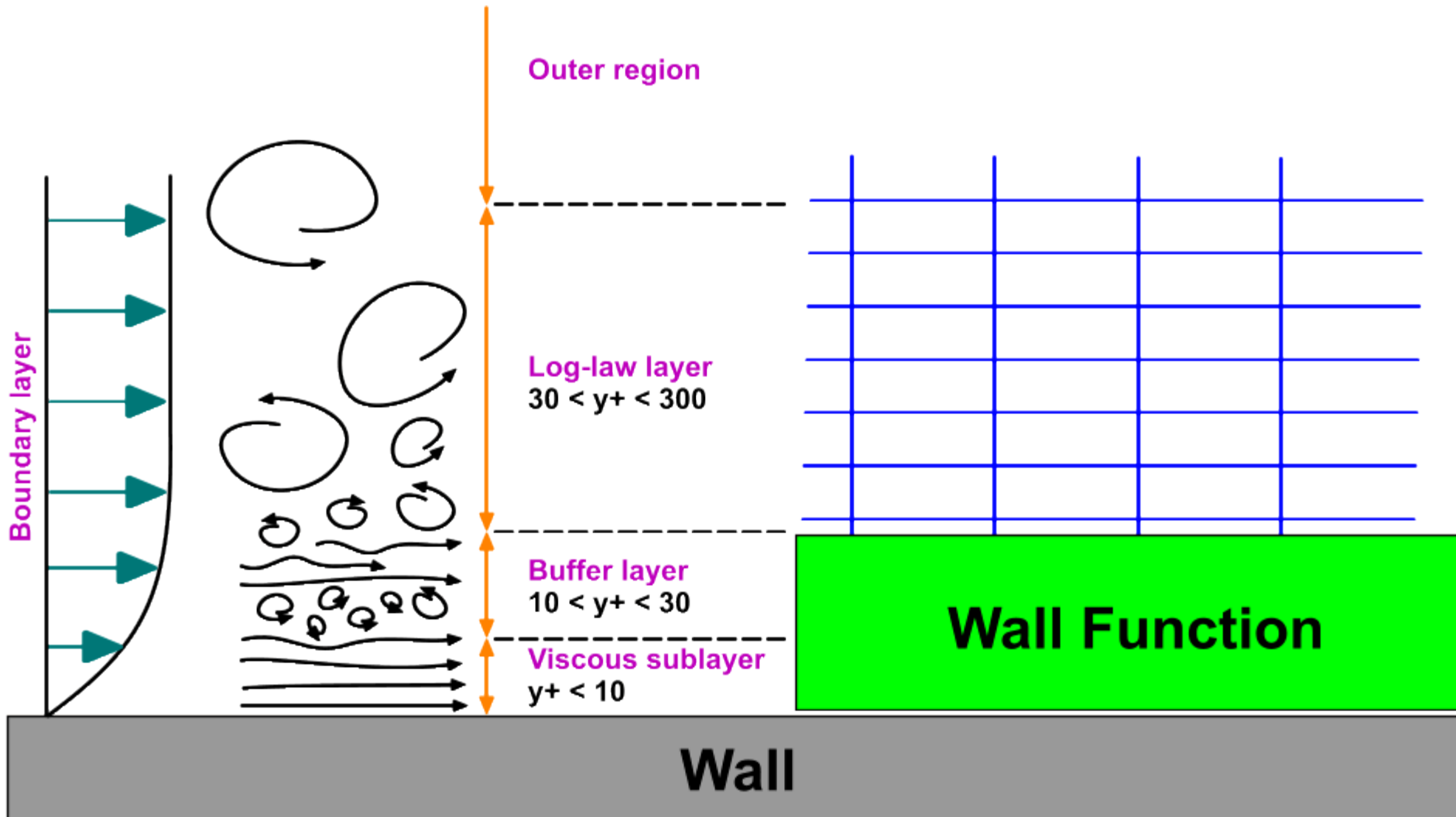
Crash introduction to turbulence modeling

Turbulence near the wall



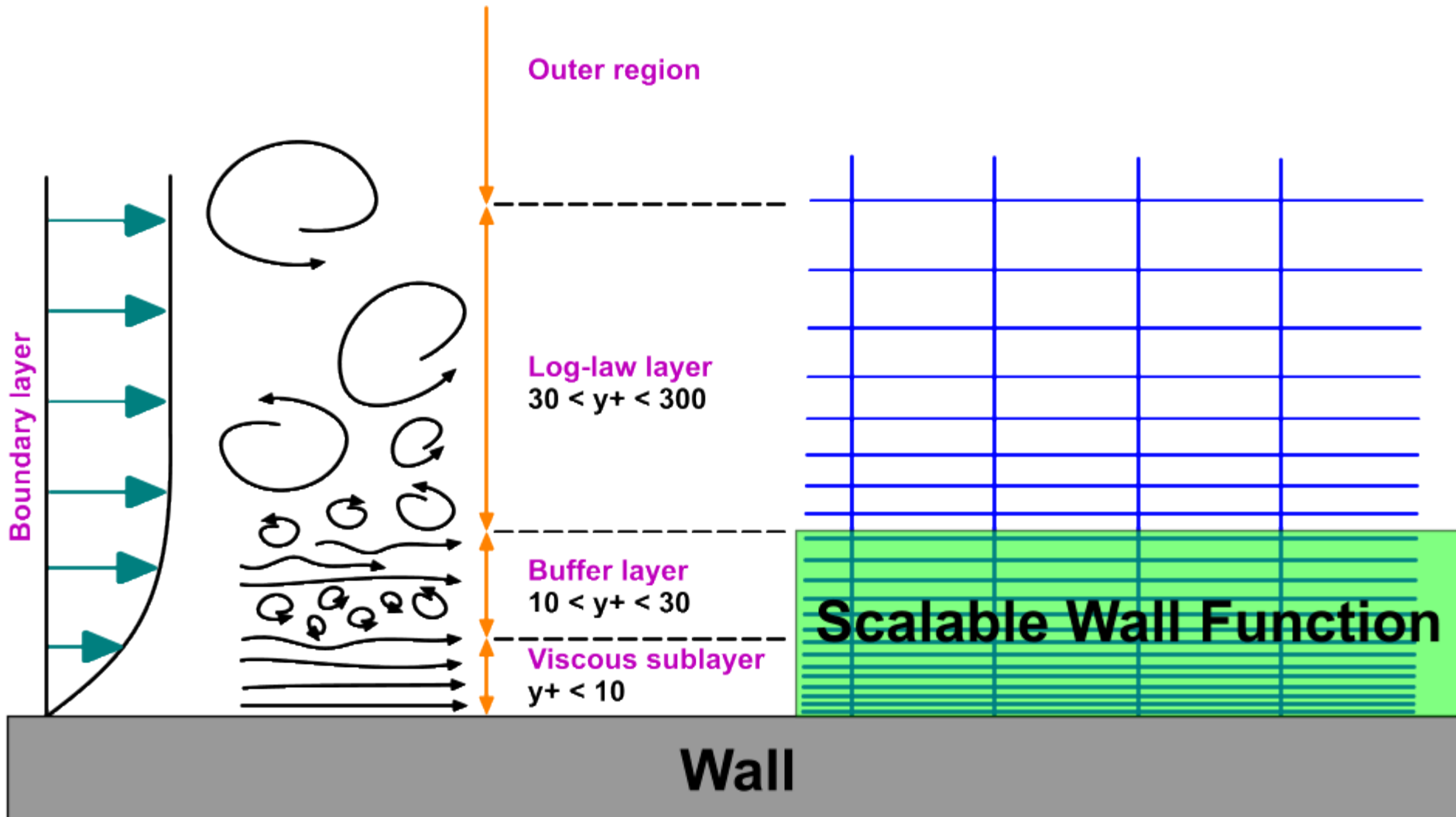
Crash introduction to turbulence modeling

Turbulence near the wall



Crash introduction to turbulence modeling

Turbulence near the wall



Crash introduction to turbulence modeling

Turbulence modeling

WIDELY USED IN INDUSTRY
A LOT RESEARCH DONE IN
THE ACADEMIA

A LOT RESEARCH DONE IN
ACADEMIA
LESS USED IN INDUSTRY

SOME INDUSTRIAL
APPLICATIONS ON LES
DNS MAINLY RESEARCH

RANS
URANS
SAS
DES
LES
DNS



Increasing computational costs

Crash introduction to turbulence modeling

RANS/URANS Turbulence models available in OpenFOAM®

- `LRR`
- `LamBremhorstKE`
- `LaunderGibsonRSTM`
- `LaunderSharmaKE`
- `LienCubicKE`
- `LienCubicKELowRe`
- `LienLeschzinerLowRe`
- `NonlinearKEShih`
- `RNGkEpsilon`
- `SpalartAllmaras`
- `kEpsilon`
- `kOmega`
- `kOmegaSST`
- `kkLOmega`
- `laminar`
- `qZeta`
- `realizableKE`
- `v2f`

Crash introduction to turbulence modeling

DES/LES Turbulence models available in OpenFOAM®

- `DeardorffDiffStress`
- `LRRDiffStress`
- `Smagorinsky`
- `SpalartAllmaras`
- `SpalartAllmarasDDES`
- `SpalartAllmarasIDDES`
- `dynLagrangian`
- `dynOneEqEddy`
- `homogeneousDynOneEqEddy`
- `homogeneousDynSmagorinsky`
- `kOmegaSSTAS`
- `laminar`
- `mixedSmagorinsky`
- `oneEqEddy`
- `spectEddyVisc`

Crash introduction to turbulence modeling

Short description of some of the turbulence models available in OpenFOAM®

Model	Short description
Spalart-Allmaras	Suitable for external aerodynamics, turbomachinery and high speed flows. Good for mildly complex external/internal flows and boundary layer flows under pressure gradient. Performs poorly for free shear flows and flows with strong separation.
Standard k-epsilon	Robust. Widely used despite the known limitations of the model. Performs poorly for complex flows involving severe pressure gradient, separation, strong streamline curvature. Suitable for initial iterations, initial screening of alternative designs, and parametric studies.
Realizable k-epsilon	Suitable for complex shear flows involving rapid strain, moderate swirl, vortices, and locally transitional flows (e.g. boundary layer separation, massive separation, and vortex shedding behind bluff bodies, stall in wide-angle diffusers, room ventilation).
Standard k-omega	Superior performance for wall-bounded boundary layer, free shear, and low Reynolds number flows compared to models from the k-epsilon family. Suitable for complex boundary layer flows under adverse pressure gradient and separation (external aerodynamics and turbomachinery).
SST k-omega	Offers similar benefits as standard k-omega. Not overly sensitive to inlet boundary conditions like the standard k-omega. Provides more accurate prediction of flow separation than other RANS models.

Crash introduction to turbulence modeling

Turbulence models in OpenFOAM®

- There is a cornucopia, plethora, fullness, abundance of turbulence models implemented in OpenFOAM®, from RANS to DES/LES.
- You can also implement yours!!!.
- If you have absolutely no idea of what model to use, I highly recommend you the $\kappa - \omega$ family models or the realizable $\kappa - \epsilon$
- When a turbulent flow enters a domain, turbulent boundary conditions and initial conditions must be specified.
- Also, if you are dealing with wall bounded turbulence you will need to choose the near-wall treatment. You can choose to solve the viscous sublayer or use wall functions.
- Remember, for the near-wall treatment you need to give the appropriate boundary conditions to the walls.

Crash introduction to turbulence modeling

Turbulence models in OpenFOAM®

- The initial conditions and boundary conditions change from model to model and depends on the physics involve. Remember, there is no universal turbulence model.
- To set the initial conditions and boundary conditions, my best advise is to find and read the original reference of the turbulence model to use.
- In OpenFOAM®, you can find the turbulence models in the directory `$FOAM_SRC/turbulenceModels`.
- For instance, if you are interested in the incompressible version of the $k - \omega$ turbulence model, you can go to the directory `$FOAM_SRC/turbulenceModels/incompressible/RAS/kOmegaSST` and take a look at the source code.

Crash introduction to turbulence modeling

Turbulence models in OpenFOAM®

- For instance, if you are interested in the incompressible version of the SST $\kappa - \omega$ turbulence model, you can go to the directory `$FOAM_SRC/turbulenceModels/incompressible/RAS/kOmegaSST` and take a look at the source code.
- If you open the file `$FOAM_SRC/turbulenceModels/incompressible/RAS/kOmegaSST/kOmegaSST.H`, in the header you will find the references used to implement the model in OpenFOAM® namely:

Menter, F., Esch, T., “Elements of Industrial Heat Transfer Prediction”, 16th Brazilian Congress of Mechanical Engineering (COBEM), Nov. 2001.

Hellsten, A. “Some Improvements in Menter’s k-omega-SST turbulence model”, 29th AIAA Fluid Dynamics Conference, AIAA-98-2554, June 1998.

- Remember, you have the source code.

Crash introduction to turbulence modeling

$\kappa - \omega$ Turbulence model free-stream boundary conditions

- The boundary conditions change from model to model. Hereafter, I am going to talk about the $\kappa - \omega$.
- The initial value for the turbulent kinetic energy κ can be found as follows

$$\kappa = \frac{3}{2}(UI)^2$$

- The initial value for the specific kinetic energy ω can be found as follows

$$\omega = \frac{\rho\kappa}{\mu} \frac{\mu_t}{\mu}^{-1}$$

- Where $\frac{\mu_t}{\mu}$ is the viscosity ratio and $I = \frac{u'}{\bar{u}}$ is the turbulence intensity.

Crash introduction to turbulence modeling

$\kappa - \omega$ Turbulence model free-stream boundary conditions

- If you are totally lost, you can use these reference values. They work most of the times, but it is a good idea to have some experimental data or initial estimate.

	Low	Medium	High
I	1.0 %	5.0 %	10.0 %
μ/μ_t	1	10	100

- By the way, use these guidelines for external aerodynamics only.

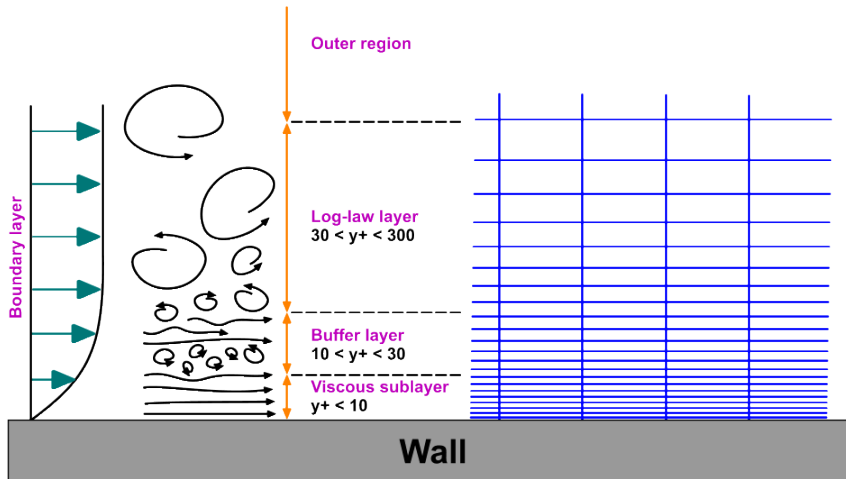
Crash introduction to turbulence modeling

Turbulence models wall functions

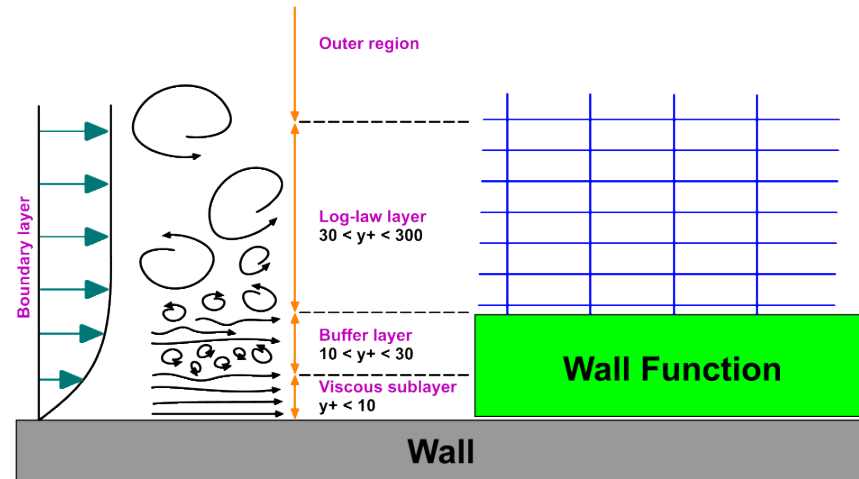
- To deal with wall bounded turbulence, several wall functions are available in OpenFOAM®, *e.g.*, `alphatJayatillekeWallFunction`, `epsilonWallFunction`, `nutUSpaldingWallFunction`, `omegaWallFunction`, `v2WallFunction`.
- If the first node normal to the wall is in the viscous sublayer region, you do not use wall functions. This approach is computationally expensive but gives the best results.
- Instead, if the first node normal to the wall is in the log-law layer, you need to use wall functions.
- I rather prefer **not to give any guidelines** about how to choose the right values for the wall functions, because that is model and problem specific. My best advise is to read the source code and find the references used to implement the model.
- If you want to find where are the directories containing the wall functions:
 - `find $FOAM_SRC/turbulenceModels -iname wallFunctions`

Crash introduction to turbulence modeling

Turbulence models wall functions



No wall-functions



Wall-functions

- Generally speaking, wall functions is the approach to use if you are more interested in the mixing in the outer region, rather than the forces on the wall.
- By the way, wall functions should never be used if $y^+ < 30$. What is y^+ ?, we are going to talk about this later on.
- If accurate prediction of forces or heat transfer on the wall are key to your simulation (aerodynamic drag, turbomachinery blade performance, heat transfer) you might not want to use wall functions.
- The recommended turbulence model for most cases is SST $\kappa - \omega$

Crash introduction to turbulence modeling

$\kappa - \omega$ Turbulence model wall functions

- I could not resist it. Follow these guidelines if you are struggling. I highly recommend you to read the source code and find the references used to implement the model.
- As for the free-stream boundary conditions, you need to give some value to the wall functions.
- For the $\kappa - \omega$ turbulence models, the following values are good choices. For the wall function `kqRWallFunction`,

$$\kappa_{wall} = 0$$

and for the wall function `omegaWallFunction`,

$$\omega_{wall} = 10 \frac{6\nu}{\beta y^2}$$

where $\beta = 0.075$ and y is the distance to the first cell center normal to the wall.

Crash introduction to turbulence modeling

y^+ Wall distance units

- To compute the wall distance units y^+ , use the following equation

$$y^+ = \frac{\rho \times U_\tau \times y}{\mu}$$

- Where y is the distance to the first cell center normal to the wall, and U_τ is the friction velocity and is equal to

$$U_\tau = \sqrt{\frac{\tau_w}{\rho}}$$

where τ_w is the wall shear stresses.

- We never know a priori the y^+ value.
- What we usually do is to run the simulation for a few time-steps or iterations, and then we get an estimate of the y^+ value.

Crash introduction to turbulence modeling

y^+ Wall distance units

- We never know a priori the y^+ value.
- What we usually do is to run the simulation for a few time-steps or iterations, and then we get an estimate of the y^+ value.
- After determining where we are in the boundary layer (viscous sublayer, buffer layer or log-law layer), we take the mesh as a good one or we modify it if is deemed necessary.
- It is an iterative process and it can be very time consuming.
- Usually, you will find the bad quality elements in the areas where you stretch the mesh to resolve the boundary layer and close to sharp angles.
- To get an initial estimate of the distance from the wall to the first cell center y , you can proceed as follows,

Crash introduction to turbulence modeling

y normal wall distance estimation

- To estimate the distance from the wall to the first cell center y , you can proceed as follows,

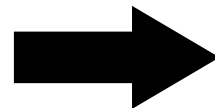
1. $Re = \frac{\rho \times U \times L}{\mu}$

2. $C_f = 0.058 \times Re^{-0.2}$
Skin friction coefficient of a flat plate

3. $\tau_w = \frac{1}{2} \times C_f \times \rho \times U_\infty^2$

4. $U_\tau = \sqrt{\frac{\tau_w}{\rho}}$

5. $y^+ = \frac{\rho \times U_\tau \times y}{\mu}$



$$y = \frac{\mu \times y^+}{\rho \times U_\tau}$$

Crash introduction to turbulence modeling

$\kappa - \omega$ Turbulence model overview

- It is called $\kappa - \omega$ because it solve two additional equations for the turbulence model, namely, the turbulent kinetic energy κ and the specific kinetic energy ω .

$$\rho \frac{\partial \kappa}{\partial t} + \rho \nabla \cdot (\bar{\mathbf{u}} \kappa) = \tau^R : \nabla \bar{\mathbf{u}} - \beta^* \rho \kappa \omega + \nabla \cdot [(\mu + \sigma^* \mu_T) \nabla \kappa]$$

$$\rho \frac{\partial \omega}{\partial t} + \rho \nabla \cdot (\bar{\mathbf{u}} \omega) = \alpha \frac{\omega}{\kappa} \tau^R : \nabla \bar{\mathbf{u}} - \beta \rho \omega^2 + \nabla \cdot [(\mu + \sigma \mu_T) \nabla \omega]$$

Crash introduction to turbulence modeling

$\kappa - \omega$ Turbulence model overview

- At the end of the day, we want to determine the turbulent eddy viscosity

$$\mu_T = \frac{\rho \kappa}{\omega}$$

- The turbulent eddy viscosity is used to compute the Reynolds stress tensor,

$$\tau^R = -\rho \overline{\mathbf{u}'\mathbf{u}'} = 2\mu_T \bar{\mathbf{D}}^R - \frac{2}{3}\rho\kappa\mathbf{I} = \mu_T \left[\nabla \bar{\mathbf{u}} + (\nabla \bar{\mathbf{u}})^T \right] - \frac{2}{3}\rho\kappa\mathbf{I},$$

- The Reynolds stress tensor is derived from the Boussinesq Approximation.

Crash introduction to turbulence modeling

Incompressible RANS equations

- The following equations are the incompressible Reynolds-Averaged Navier-Stokes equations (RANS). These are the equations we want to solve.

$$\nabla \cdot (\bar{\mathbf{u}}) = 0,$$

$$\frac{\partial \bar{\mathbf{u}}}{\partial t} + \nabla \cdot (\bar{\mathbf{u}}\bar{\mathbf{u}}) = \frac{-\nabla \bar{p}}{\rho} + \nu \nabla^2 \bar{\mathbf{u}} - \frac{1}{\rho} \nabla \cdot \tau^R.$$

where τ^R is the Reynolds stress tensor and is given by,

$$\tau^R = -\rho \overline{(\mathbf{u}'\mathbf{u}')} = 2\mu_T \bar{\mathbf{D}}^R - \frac{2}{3}\rho\kappa\mathbf{I} = \mu_T \left[\nabla \bar{\mathbf{u}} + (\nabla \bar{\mathbf{u}})^T \right] - \frac{2}{3}\rho\kappa\mathbf{I},$$

Crash introduction to turbulence modeling

Turbulence modeling guidelines

- Compute Reynolds number and determine whether the flow is turbulent.
- Estimate y before generating the mesh.
- Run the simulation for a few time steps and get a better prediction of y^+ and correct your initial prediction of y .
- If you do not know what model to use the $\kappa - \omega$ family models is a good choice.
- If you are interesting in modeling the smallest eddies, DES or LES is the right choice.
- If you do not have any restriction in the near wall treatment method, use wall functions.
- Use the default model constants unless you know what are you doing or you are confident that you have better values.
- Set reasonable boundary and initial conditions for the turbulence model variables.

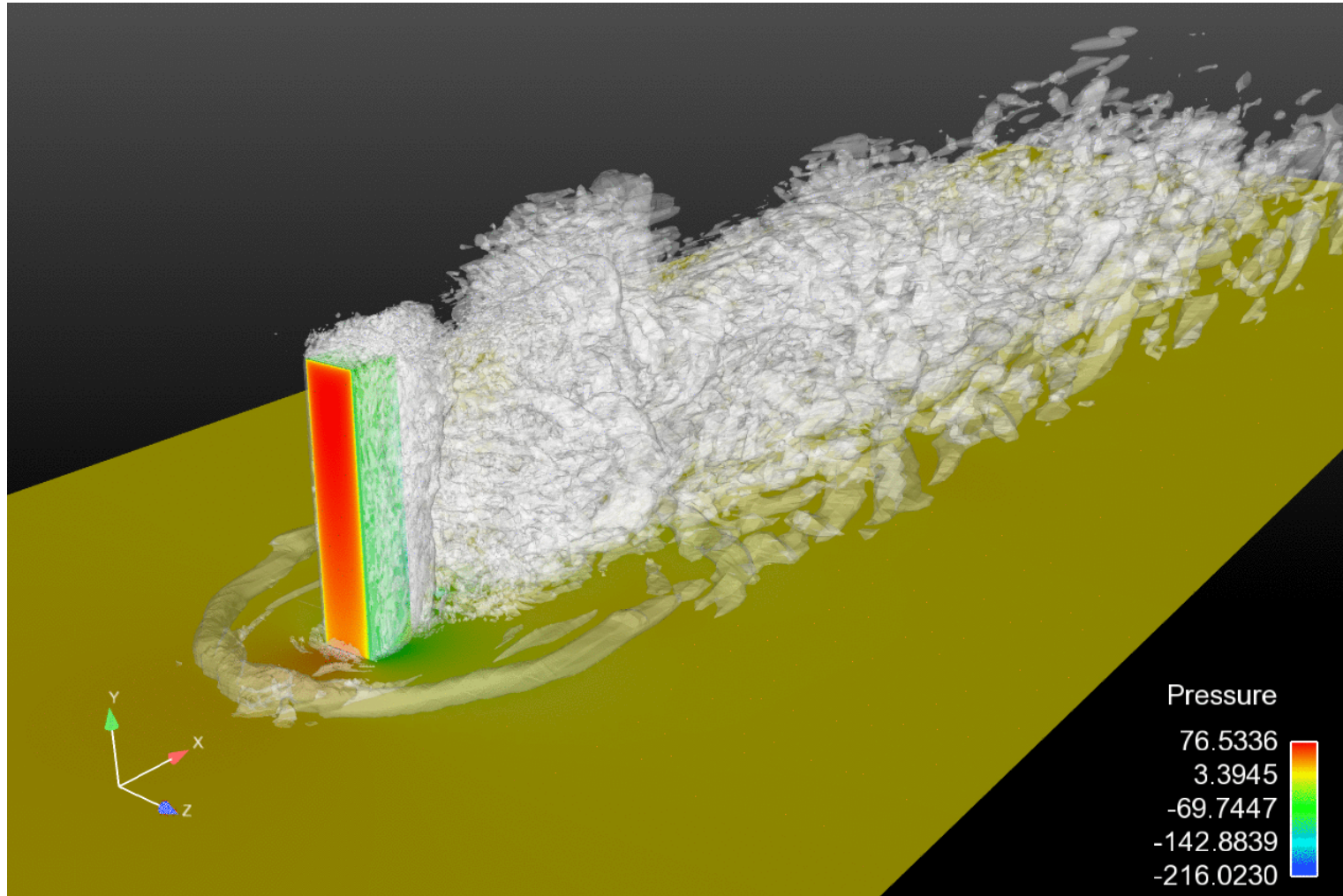
Crash introduction to turbulence modeling

OpenFOAM® Tips – Turbulence Tips

- The utility `refineWallLayer`, can be used to refine the cells next to patches. This can save you a lot time, as you do not have to go back to the meshing stage.
- The realizable $k - \epsilon$ or SST $k - \omega$ models are good choices for general applications.
- Be aware of the limitations of the turbulence model chosen, find and read the original references used to implement the model in OpenFOAM®.
- Choose your near-wall modeling strategy ahead of time and check y^+ and y values to make sure the near-wall mesh is suitable.
- Use the utility `yPlusRAS` to check the y^+ value.

Crash introduction to turbulence modeling

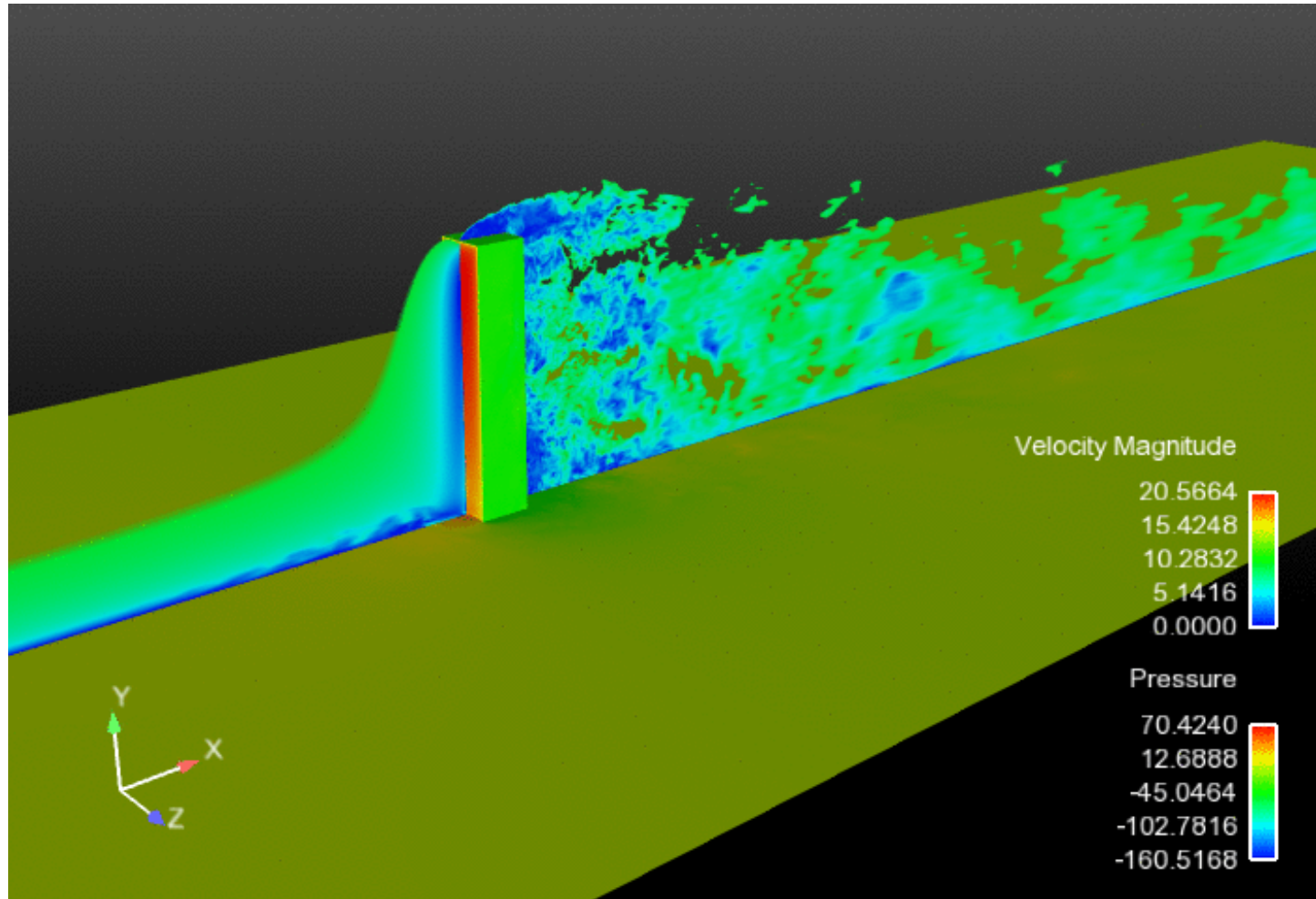
LES simulation



Iso surfaces of Q criterion. Walls coloured by instantaneous pressure.

Crash introduction to turbulence modeling

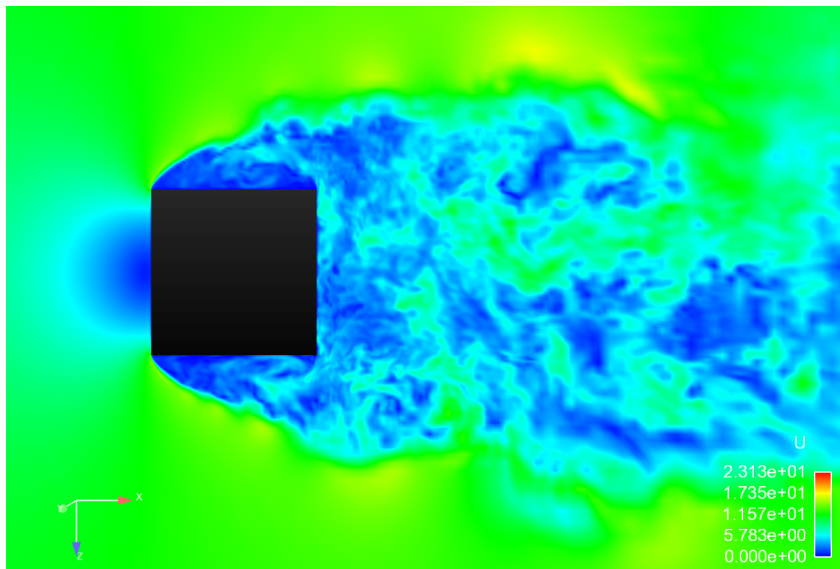
LES simulation



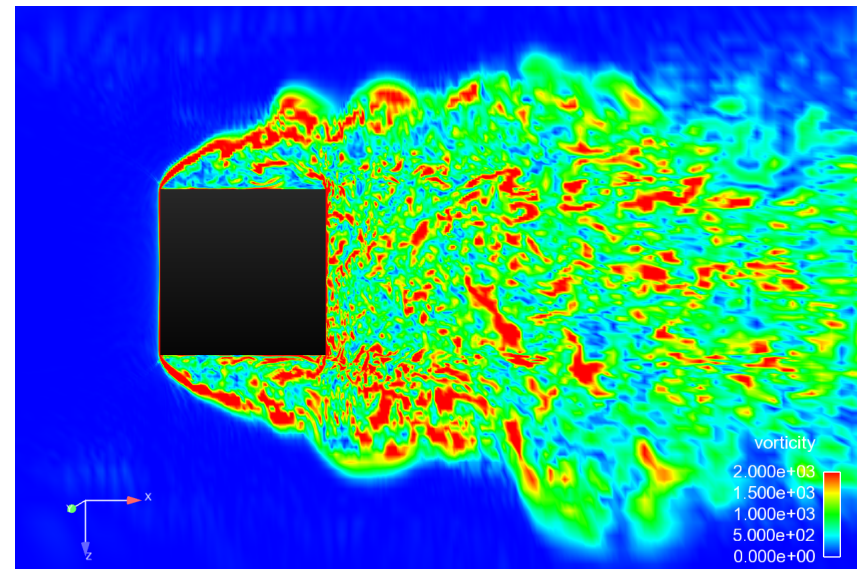
Cut plane with instantaneous velocity magnitude. Walls coloured by instantaneous pressure.

Crash introduction to turbulence modeling

LES simulation



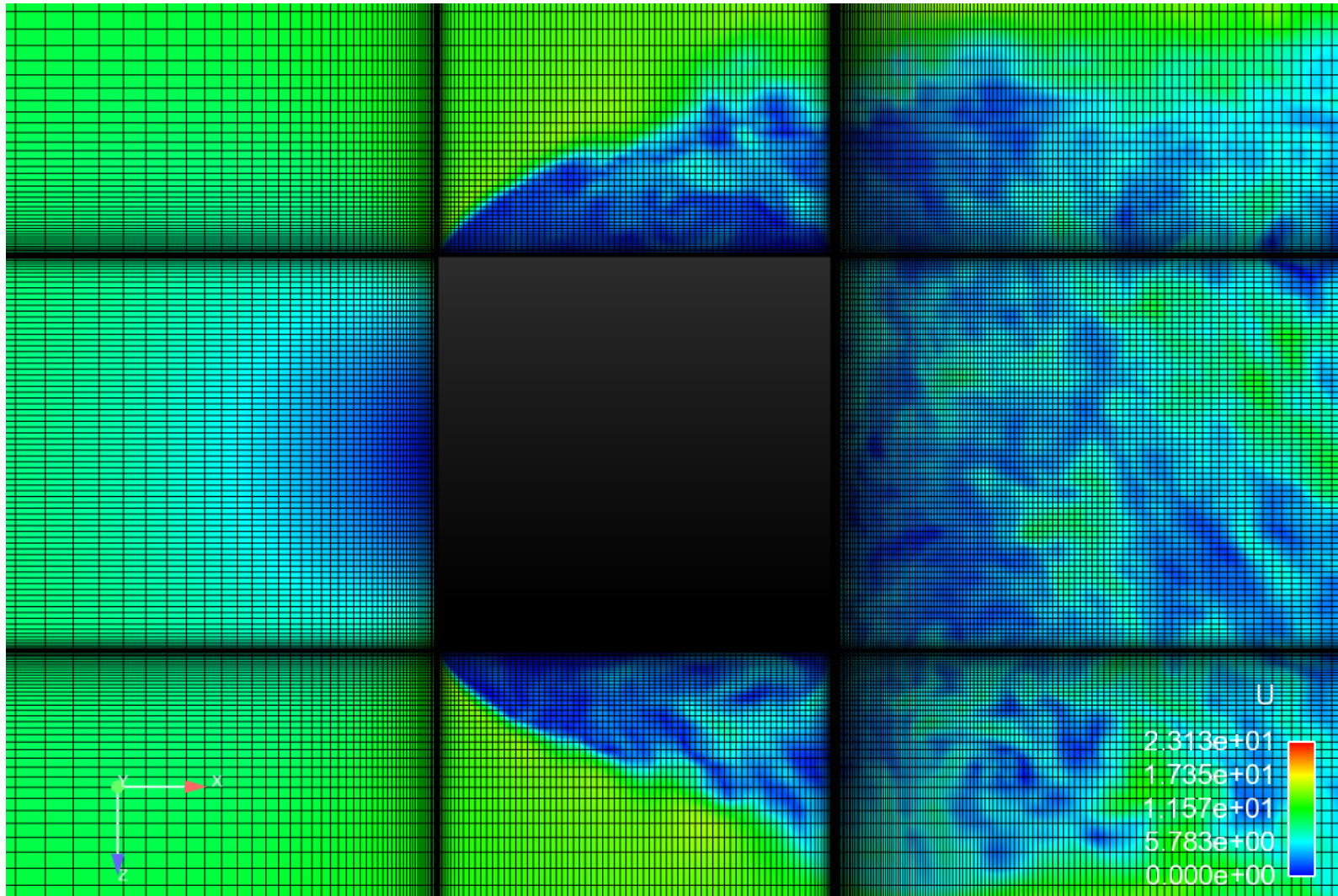
Instantaneous velocity magnitude.



Instantaneous vorticity magnitude.

Crash introduction to turbulence modeling

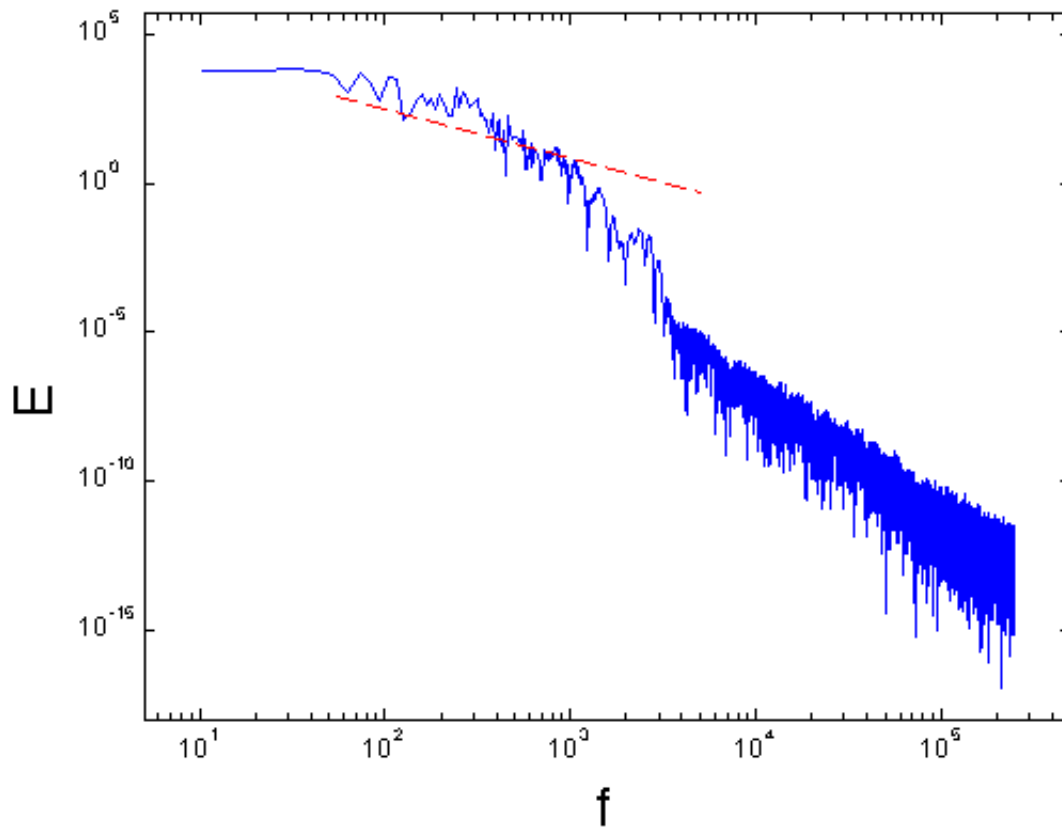
LES simulation



Mesh and instantaneous velocity magnitude.

Crash introduction to turbulence modeling

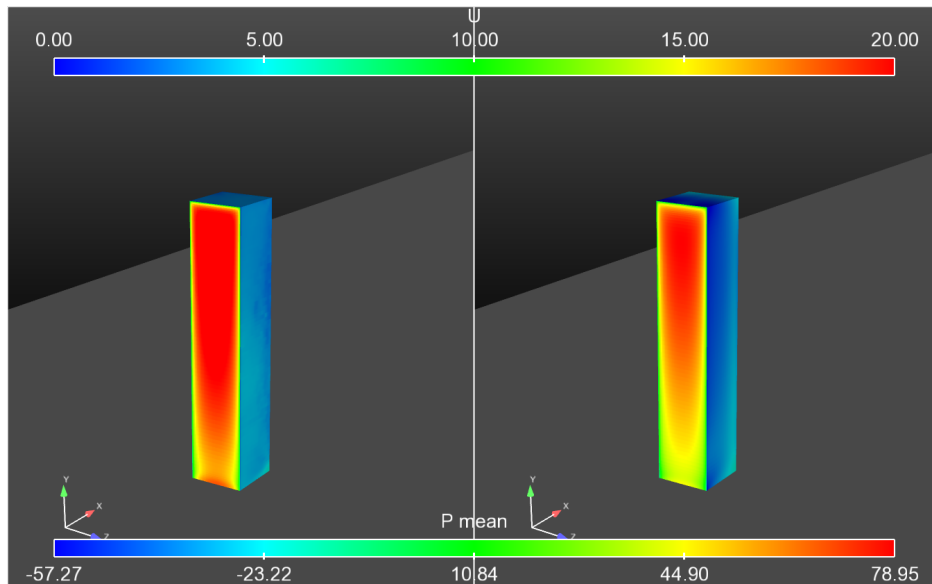
LES simulation



Energy spectrum and $-5/3$ slope.

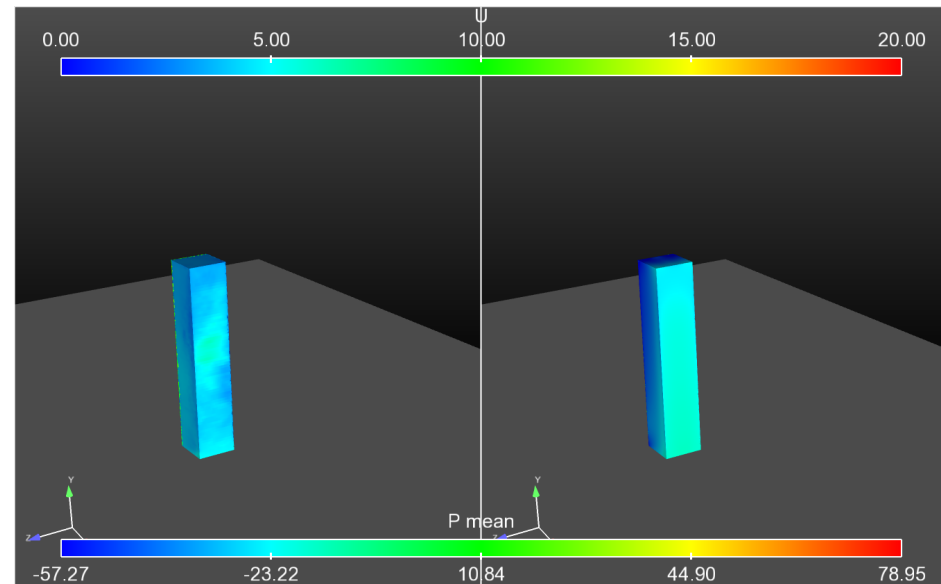
Crash introduction to turbulence modeling

LES simulation Vs. RANS simulation



LES

RANS



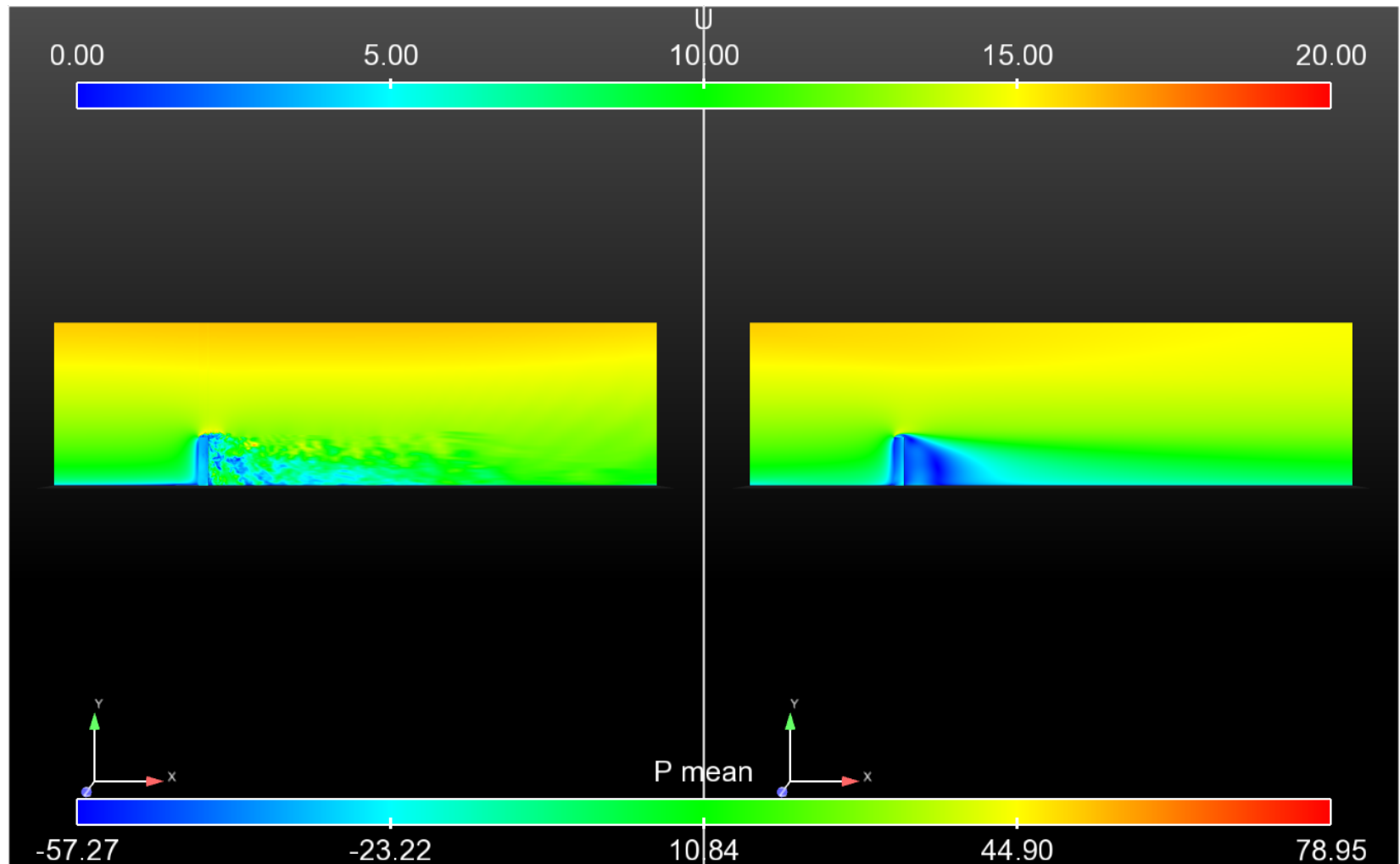
LES

RANS

- LES simulations are coloured by instantaneous values.
- RANS simulations are coloured by mean values.

Crash introduction to turbulence modeling

LES simulation Vs. RANS simulation

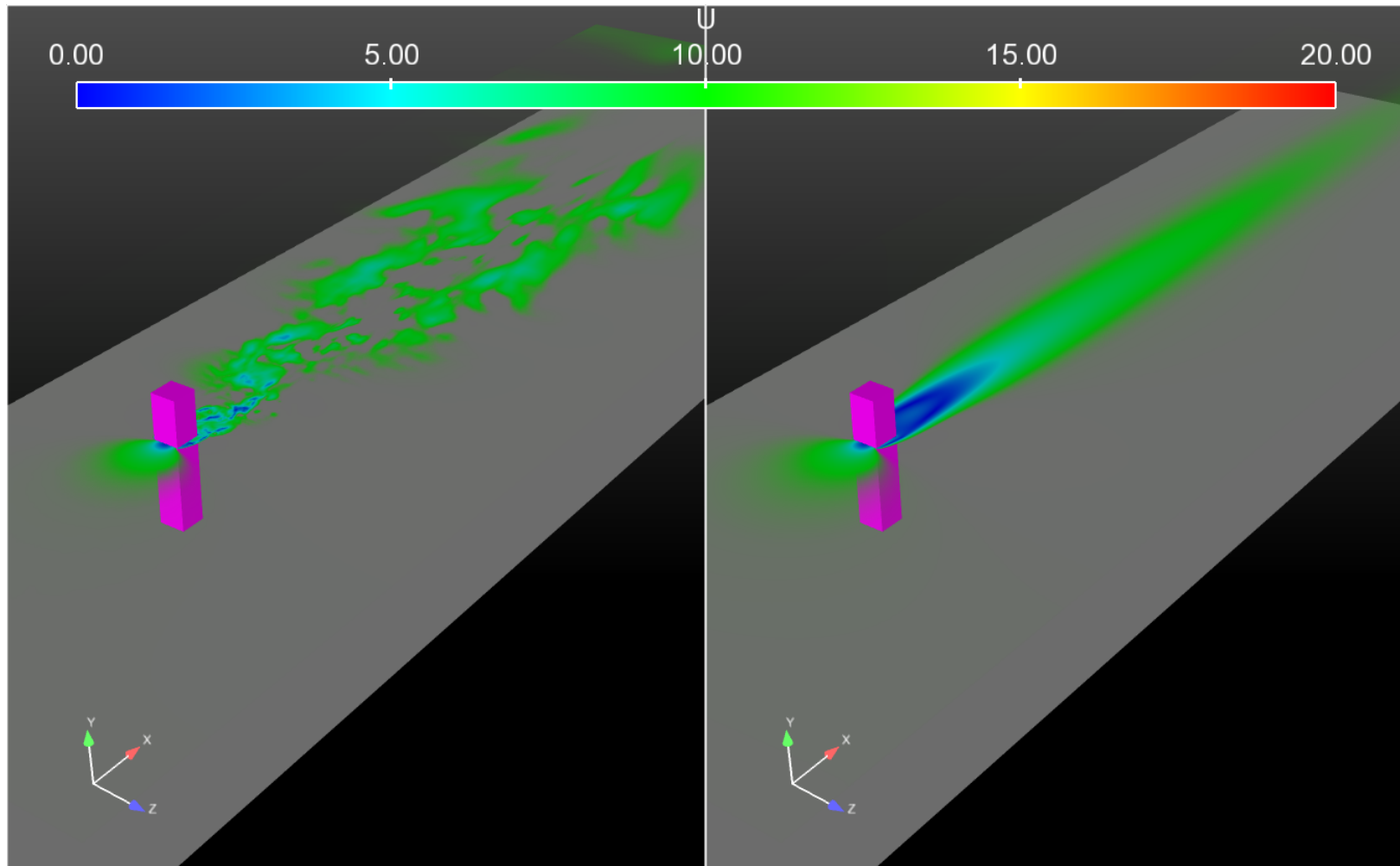


LES (instantaneous velocity)

RANS

Crash introduction to turbulence modeling

LES simulation Vs. RANS simulation

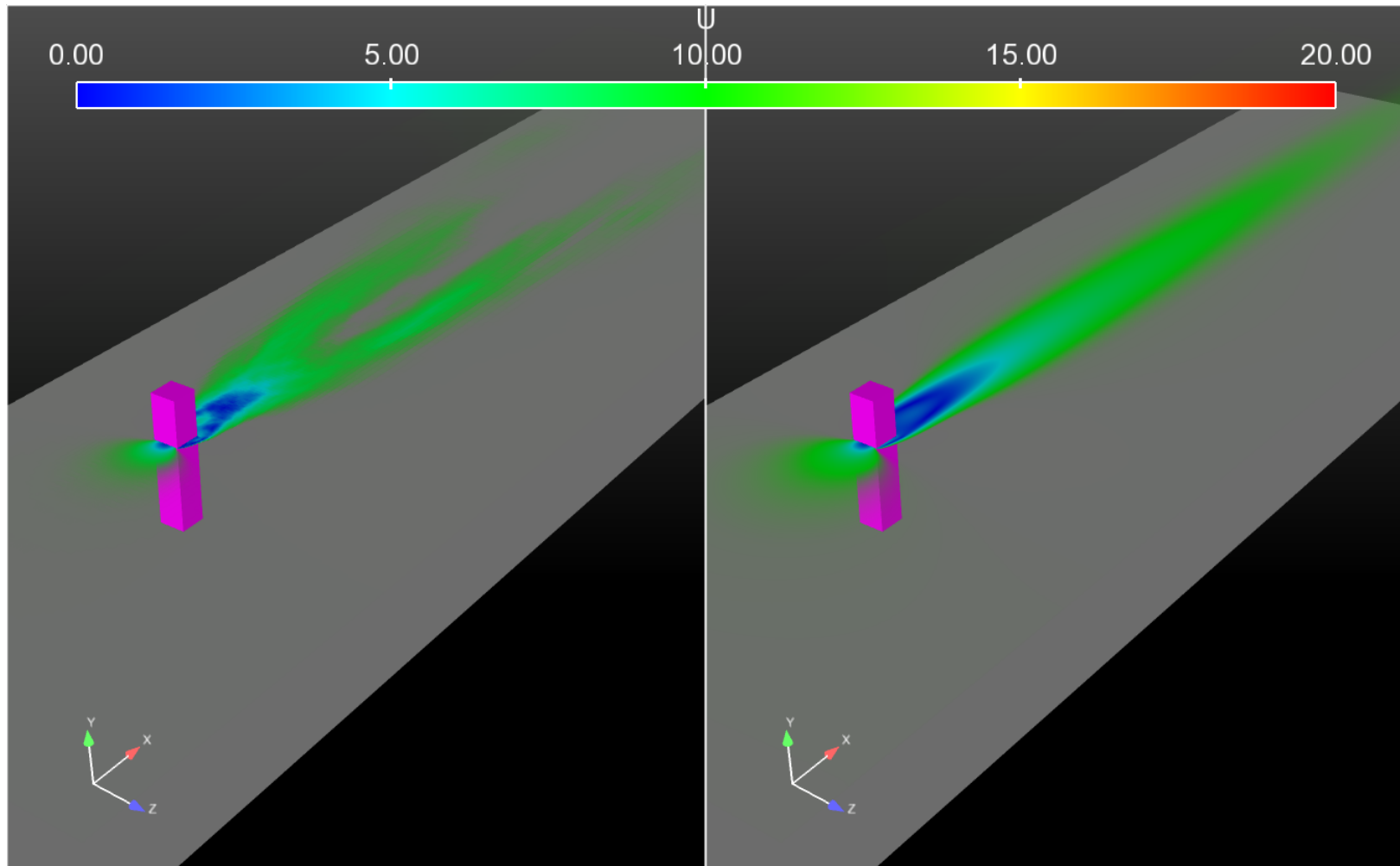


LES (instantaneous velocity)

RANS

Crash introduction to turbulence modeling

LES simulation Vs. RANS simulation

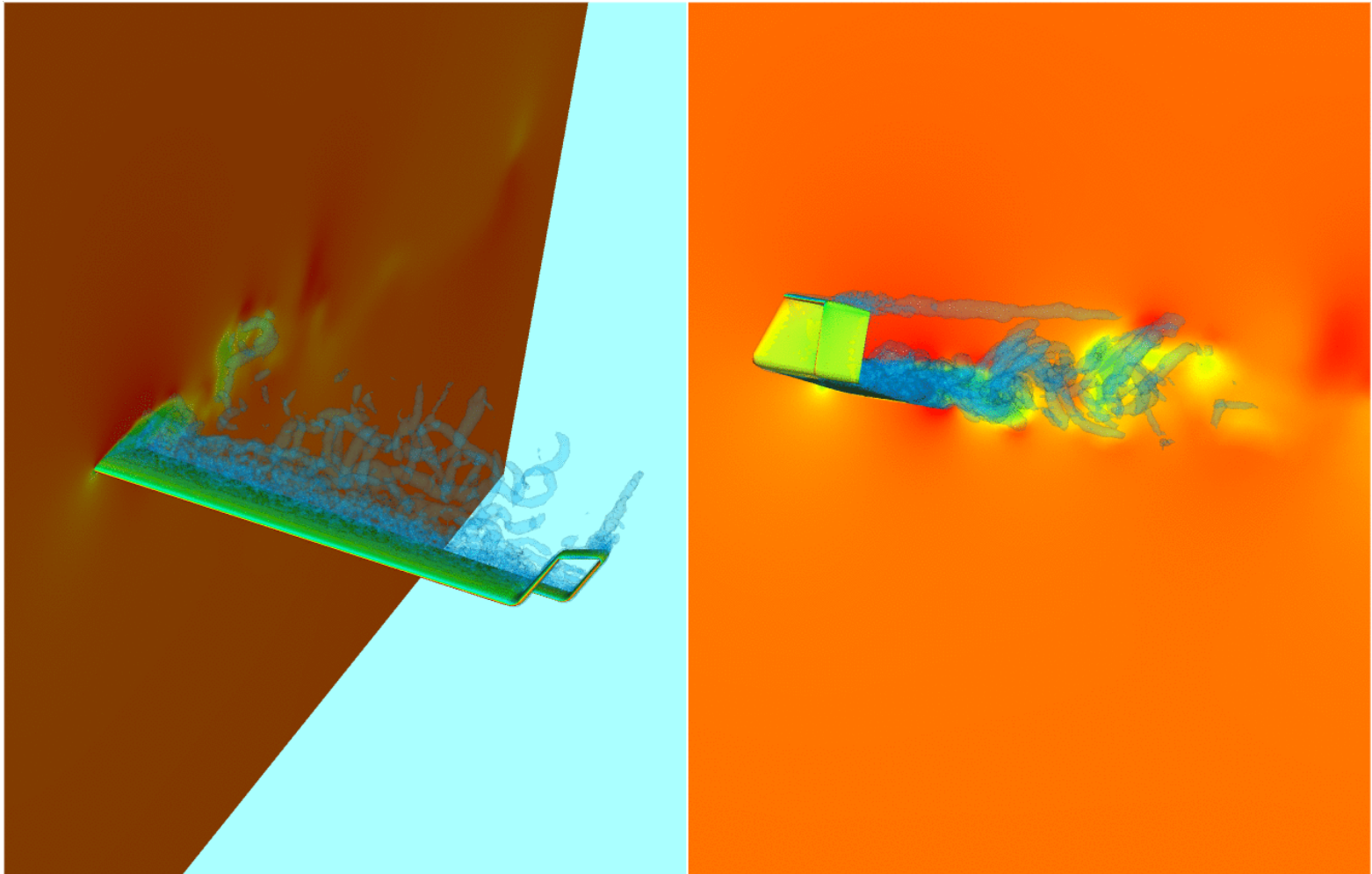


LES (Mean velocity)

RANS

Crash introduction to turbulence modeling

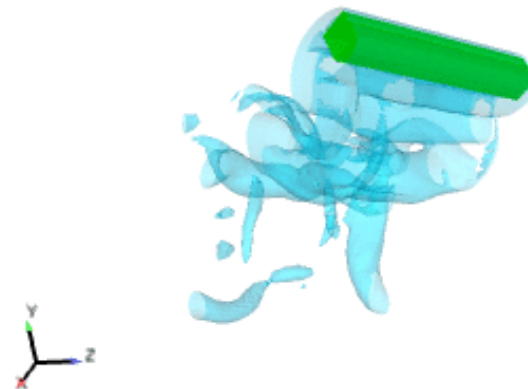
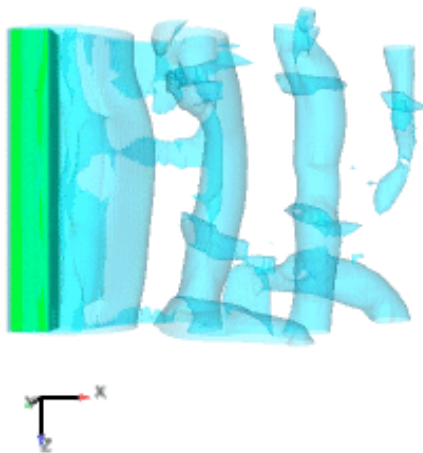
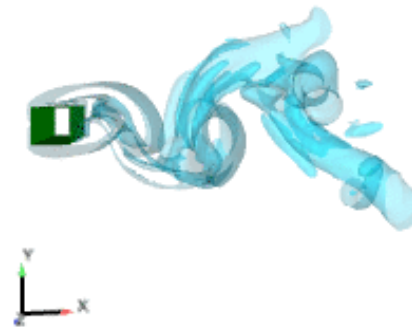
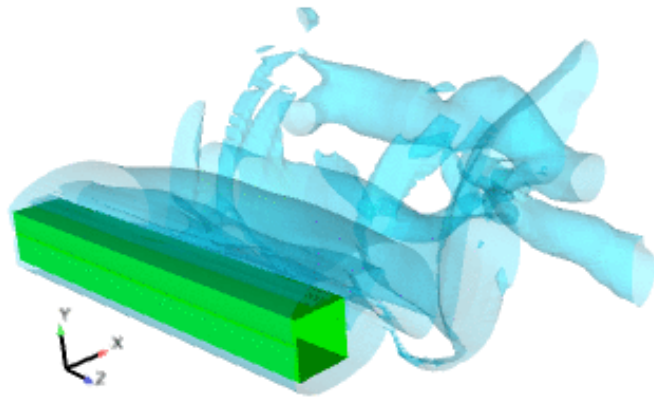
DES simulation



Iso surfaces of Q criterion.

Crash introduction to turbulence modeling

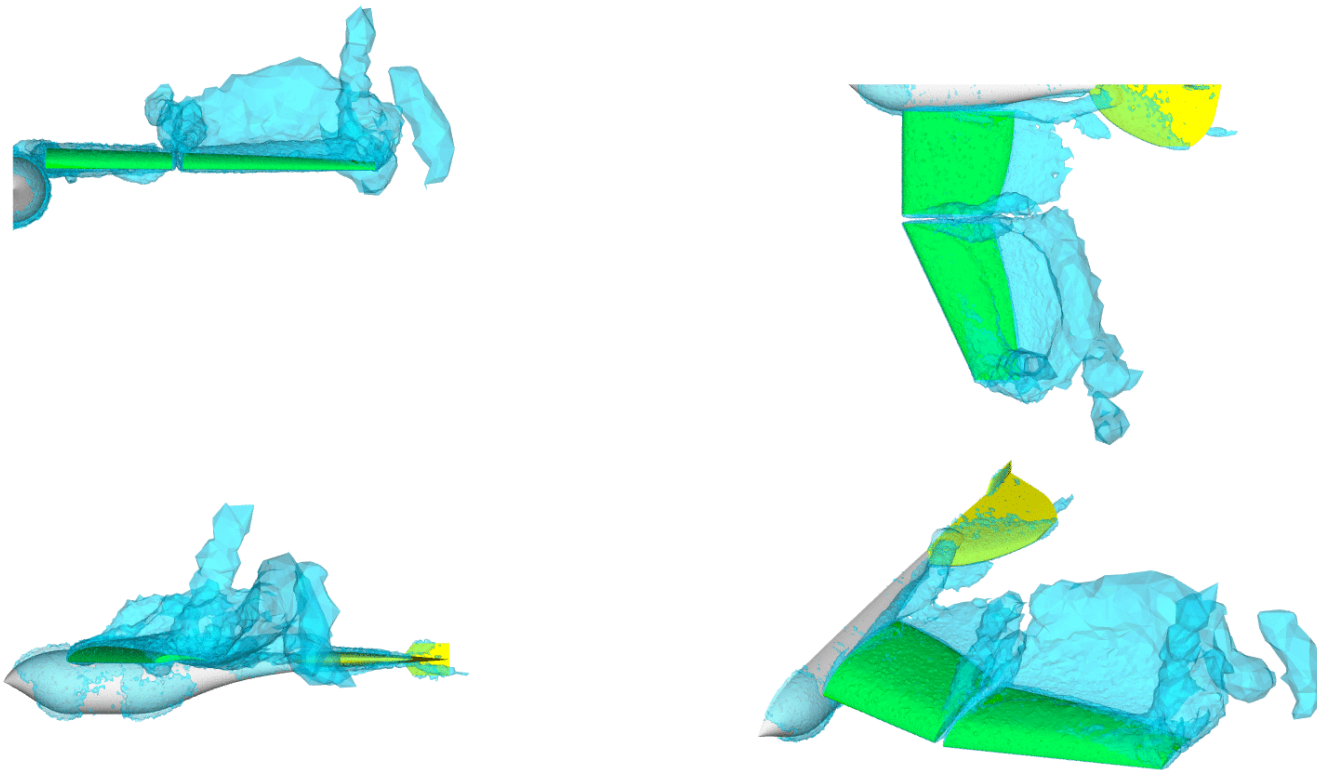
SAS simulation



Iso surfaces of vorticity magnitude.

Crash introduction to turbulence modeling

URANS simulation



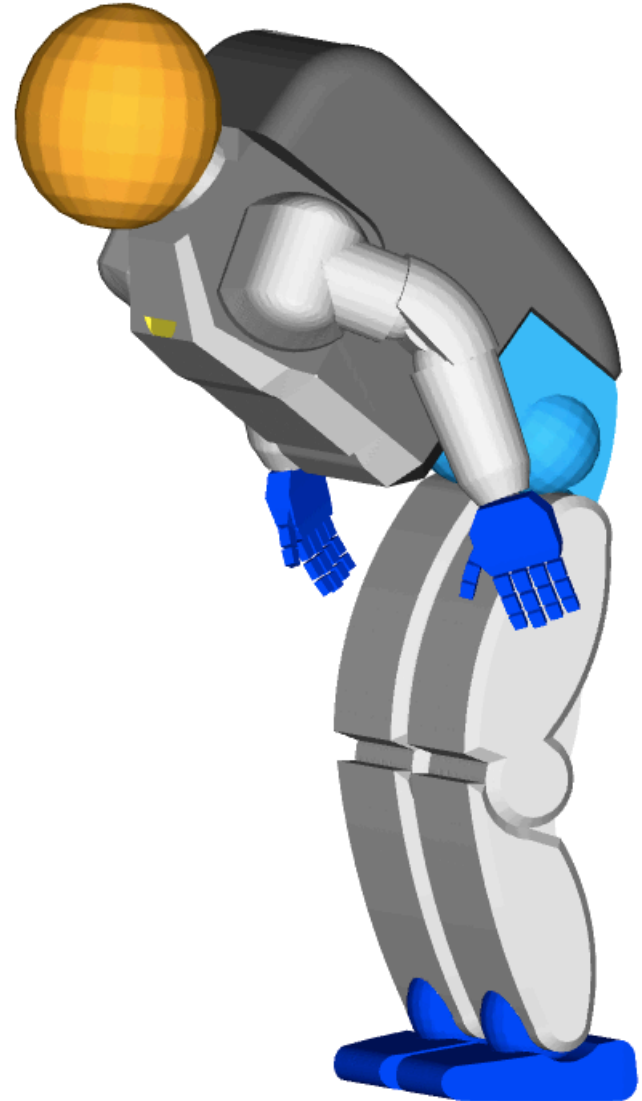
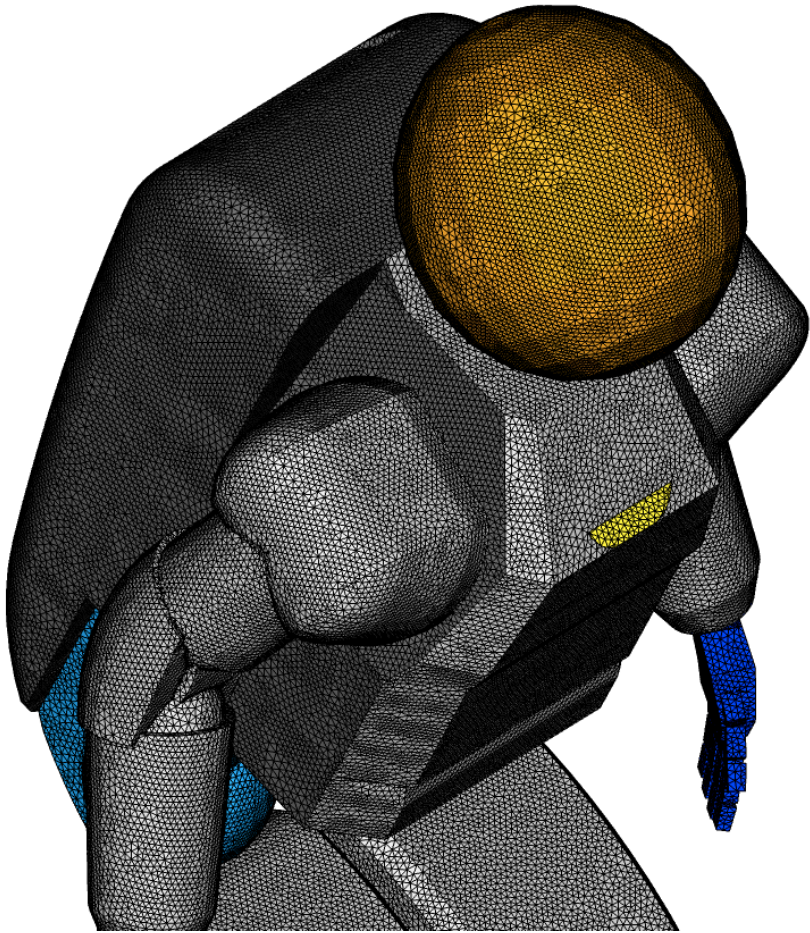
Iso surfaces of Q criterion.

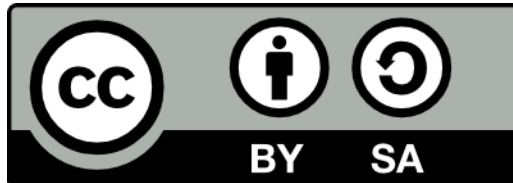
Crash introduction to turbulence modeling

References

- Turbulence Modeling for CFD
David C. Wilcox
- Large Eddy Simulation for Incompressible Flows
Pierre Sagaut

Thank you for your attention





These lectures notes are licensed under the Creative Commons Attribution-ShareAlike 4.0 International License.

To view a copy of this license, visit

<http://creativecommons.org/licenses/by-sa/4.0/>