ANSYS®

Quick Guide to Setting Up LES-type Simulations

Version 1.4
May, 2016

For updates and latest versions please check here

ANSYS Customer Portal: Solution 2039348
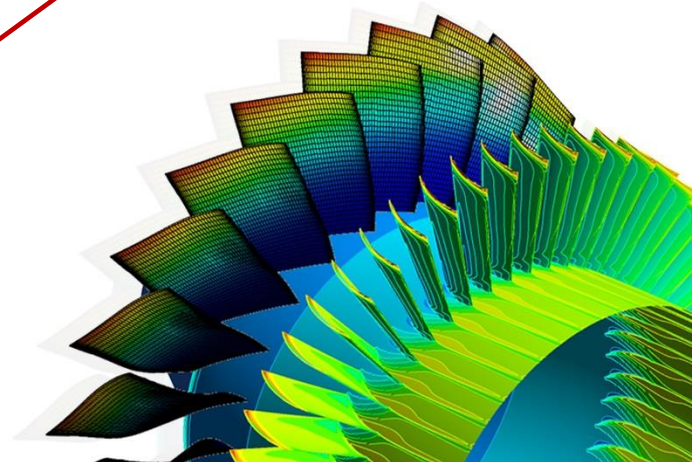
Prepared and compiled by

Dr Aleksey Gerasimov

European Technology Group

ANSYS Sweden AB

Aleksey.Gerasimov@ansys.com

Table of Contents

ANSYS

Procedure for LES

- **Evaluate adequate RANS inlet profiles for velocity and turbulent quantities.**

- **Compute the mean flow with steady RANS.**

- **Make assessments for required mesh resolution.**

- **Create a new LES-friendly mesh.**

- **Interpolate the original RANS solution onto the new LES mesh or perform another RANS simulation on the new LES mesh.***

- **Adapt the mesh in the near-wall regions, if necessary.**

- **Make assessments for required temporal resolution (time step, Δt).**

- **Superimpose the synthetic turbulence on the mean flow in the computational domain.**

  TUI: `/solve/initialize/init-instantaneous-vel`**

- **Switch to LES, select the SGS turbulence model of your choice.**

* ** – These steps should not be used for reacting flows as any synthetic perturbations to the stoichiometry field are likely to lead to severe stability issues. It is much easier and quicker to patch the fluid domain with the oxidant species and then gradually introduce fuel and let reaction to take place gradually.

       May 23, 2016   **ANSYS Confidential**

Procedure for LES (2)

- Select the solver algorithm (e.g., ITA (refrain from using NITA at the very beginning), PISO or SIMPLEC) and the discretisation schemes.

- Set the time-step (Δt) and adjust the solver parameters if needed (e.g., URFs, convergence criteria)

- Enable the generation of synthetic turbulence at the inlets. Vortex Method is preferred over the Spectral Synthesizer.

- Assess the number of cell faces at each of the inlets N :
  - GUI: Surface / Manage -> Select inlet boundary from the "Surfaces" list, the number faces will be shown in the "2D Facets" field
  - TUI: `/mesh/mesh-info 0` and then find a number of faces for a specific zone ID. The required zone ID can be found from: `/mesh/modify-zones/list-zones`

- Set the number of vortices for the Vortex Method to N/4

- Set the monitors with relevant global (e.g., forces/moments) and local quantities (e.g., velocity, pressure)

- Set the autosave of the data files (*e.g.,* every few hundred time-steps)
  GUI: File/Write/Autosave…

ANSYS®

Procedure for LES (3)

- **Start the transient run and use higher number of iterations per time step at the very beginning of the simulation. Consider using as many as 100 iterations for the first time step.**

- **A smaller time step ($\Delta t_{initial} = \Delta t/10$) might also be necessary at the very beginning*. A higher level of reduction might be required for reacting flows.**

- **Gradually reduce the number of iterations per time step, from 100** to 10, and ramp up the time step from $\Delta t_{initial}$ to Δt as simulation progresses.**

- **Consider using NITA with FSM or PISO pressure-velocity coupling methods for incompressible or weakly compressible flows in order to speed up the simulation, remember to remain conservative with Δt.**

- **Continue until a statistically averageable state is reached.**

- **Select the right viewpoints, graphic displays and the right fixed levels of colourmaps, so that the right pictures could be created at specified intervals for animations once the statistically averageable state is reached.**

- **Write a journal file for easier and automated postprocessing.**

* ** – These are very conservative figures and it is worth mentioning that the time step increase as well as the reduction of the number of iterations per time step can be performed within the first few dozens of time steps, depending on residuals behaviour.

Procedure for LES (4)

- **Start sampling the data (to compute mean and r.m.s. values)**
  GUI: Solve/Iterate… (click on the "Data Sampling for Time Statistics" button)
  GUI: Solve/Initialize/Reset Statistics to reset the data sampling

- **Continue sampling for a sufficiently long period of time**
  Several flow-through times (L / U_0)
  Mean fields need to recover homogeneity

- **Post-process the results (mean, r.m.s., power spectra, *etc*.)**

Precursor RANS Simulation

- **All types of Scale-Resolving Simulations (SRS) such as LES, DES, SAS and their flavours should start from a RANS Simulation**

  - **Typical meshes used in RANS might not be suitable for LES.**

  - **The assessment of the local mesh requirement can be based on an existing RANS solution for the same domain.**

  - **Once the new LES-type mesh has been created, the assessments for the time-stepping can be made.**

  - **The initial synthetic turbulence can be generated based on RANS solution in order to achieve a quicker statistically averageable state in LES.**

ANSYS®

Requirements for Precursor RANS

- **As a bare minimum a 2-equation eddy-viscosity model should be used for precursor RANS simulations**

  - **The choice of the EVM should fall for the one that is most suitable & accurate for a given type of flow/scenario.**

  - **Consider using RKE, SST or RNG.**

  - **Refrain from using the Standard k-ε or 1-equation Spalart-Allmaras*** **models.**

  - **In some cases even the RSM solutions might be required.**

- **For internal flows, the prescribed flat inlet profiles for velocity and turbulent quantities can be detrimental and, therefore, must not be used.**

- **Fully-developed inlet boundary conditions can be of great benefit to the precursor RANS and subsequent LES simulation.**

*** - The precursor RANS simulation for DES-SA model will be the only exception from this rule**
for comparison purposes: DES-SA vs RANS-SA

ANSYS®

Evaluation of the Fully Developed Inlet Boundary Conditions

- **Take the existing flow domain and extrude three rows of cells away from the current inlet(s) by using TUI:**

  mesh modify-zones extrude-face-zone-delta
  specify the face zone id/name   **i.e.   inlet boundary name or zone ID**
  distance delta 1   0.2*
  distance delta 2   0.2
  distance delta 3   0.2
  distance delta 4   **just press enter as the 3-cells-thick domain should be sufficient**.

- **Delete the original fluid zone with your current existing geometry and save the file.**

- **Make the offset faces (inlet-outlet) in the new fluid zone periodic by using TUI:**

  mesh modify-zones make-periodic

*   – The offset value does not need to be 0.2 and can be equal to the biggest cell edge length at the inlet face.
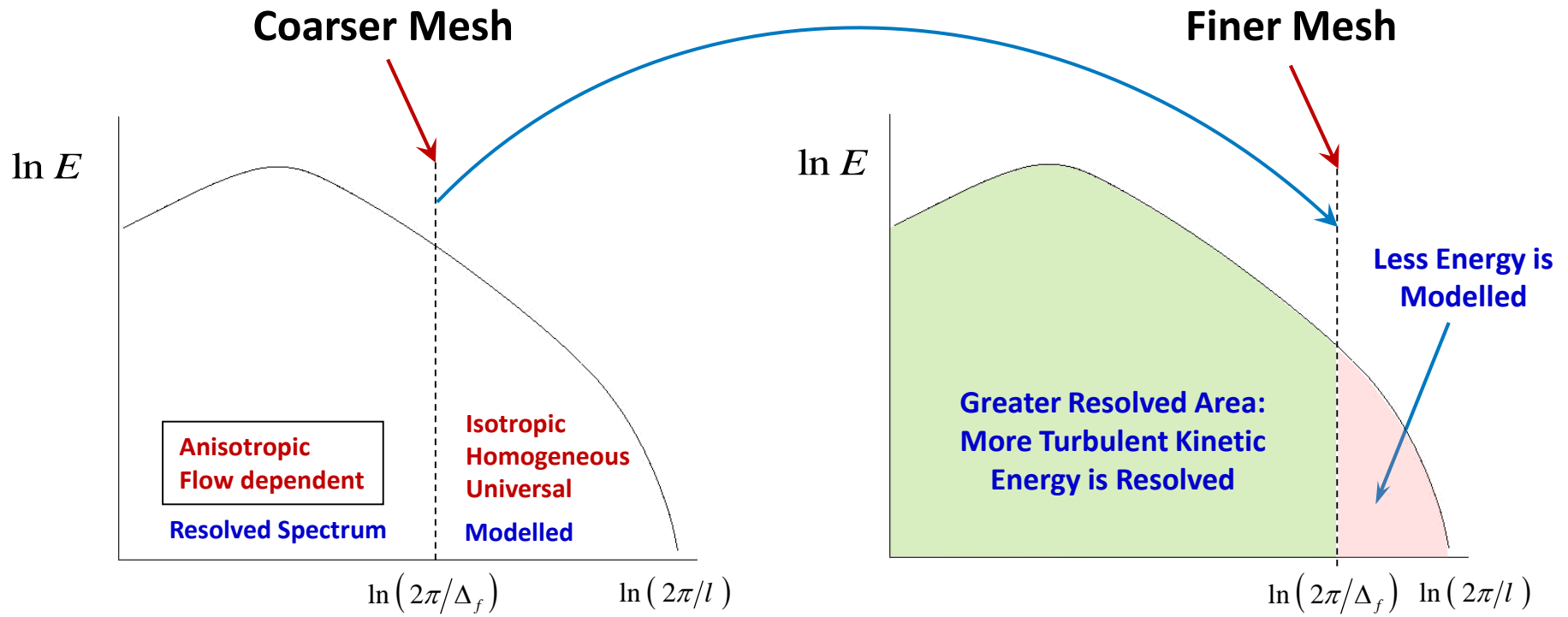** – Thus a new fluid zone will be formed

Evaluation of the Fully Developed Inlet Boundary Conditions (2)

- **Set up the periodic case, by specifying the flow rate and ensuring the consistency of the existing boundary conditions, models *& properties.**

- **Ensure that the surrounding side boundaries are consistent with the original geometry, for example the periodic side boundaries of the original geometry might become walls in the extruded zone.**

- **Iterate and obtain the fully periodic solution.**

- **Check the obtained periodic solution: flow rate, direction *& convergence**

- **Write the profile file with the values of *U, k* (or stresses) and ε (or ω).**

- **Save and close the "periodic" Fluent session.**

- **Open the original Fluent session and read in the inlet profiles for *U, k* (or stresses) and ε (or ω).**

ANSYS®

Mesh Resolution in LES

- **LES requires mesh and the time-step sizes sufficiently fine to resolve the energy-containing eddies**
  - **The mesh resolution determines the fraction of turbulent kinetic energy directly resolved**

Coarser Mesh　　　　　　　　　　　　　　**Finer Mesh**

$\ln E$

Anisotropic Flow dependent

Isotropic Homogeneous Universal

Resolved Spectrum　　**Modelled**

$\ln\left(2\pi/\Delta_f\right)$　　$\ln\left(2\pi/l\right)$

$\ln E$

Less Energy is Modelled

Greater Resolved Area: More Turbulent Kinetic Energy is Resolved

$\ln\left(2\pi/\Delta_f\right)$　$\ln\left(2\pi/l\right)$

Energy spectrum against the inverted length scale (log scales)
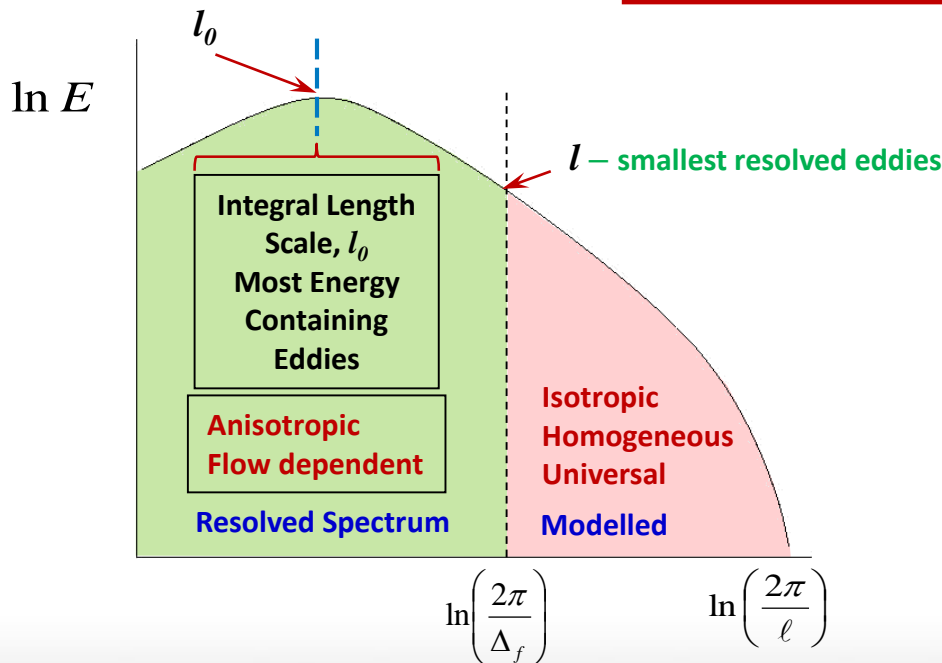**(Please note, this is a local graph: would be different for each and every point in the domain)**

Mesh Resolution in LES (2)

- **Integral length scale l_0**
  - **Turbulent kinetic energy peaks at integral length scale. This scale must be sufficiently resolved**
  - **Approximate estimation for l_0**

   **From the precursor RANS simulation l_0 can be computed as**

    $$l_0 = k^{1.5}/\varepsilon \ \text{ or } \ l_0 = k^{0.5}/(C_\mu \omega)$$

    $C_\mu = 0.09$

$\ln E$

l_0

Integral Length Scale, l_0
Most Energy Containing Eddies

Anisotropic Flow dependent

Isotropic Homogeneous Universal

Resolved Spectrum

Modelled

l — smallest resolved eddies

$\ln\left(\dfrac{2\pi}{\Delta_f}\right)$

$\ln\left(\dfrac{2\pi}{\ell}\right)$

- **Smallest resolved length scale l**
 - **At least, a couple of cells needed in each direction to resolve an eddy with a length scale l**

$$\Delta = \frac{l}{2}$$

l

Δ

ANSYS
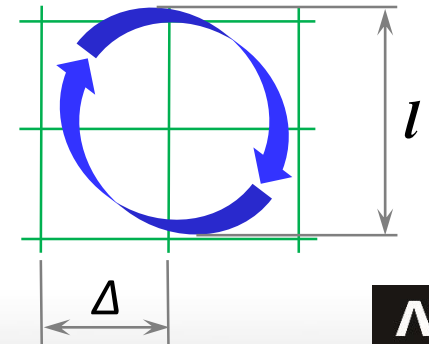
Mesh Resolution in LES (3)

- **Suppose we want to resolve 80% of the turbulent kinetic energy**

- **Then, we need to resolve the eddies whose sizes are larger than roughly half the size of the integral length scale l_0.**

 - **Approximately 5 cells will be needed across the integral length scale l_0.**

**Cumulative TKE against length-scale of eddies based on the Kolmogorov's energy spectrum**

$$\frac{k(\ell)}{k}$$
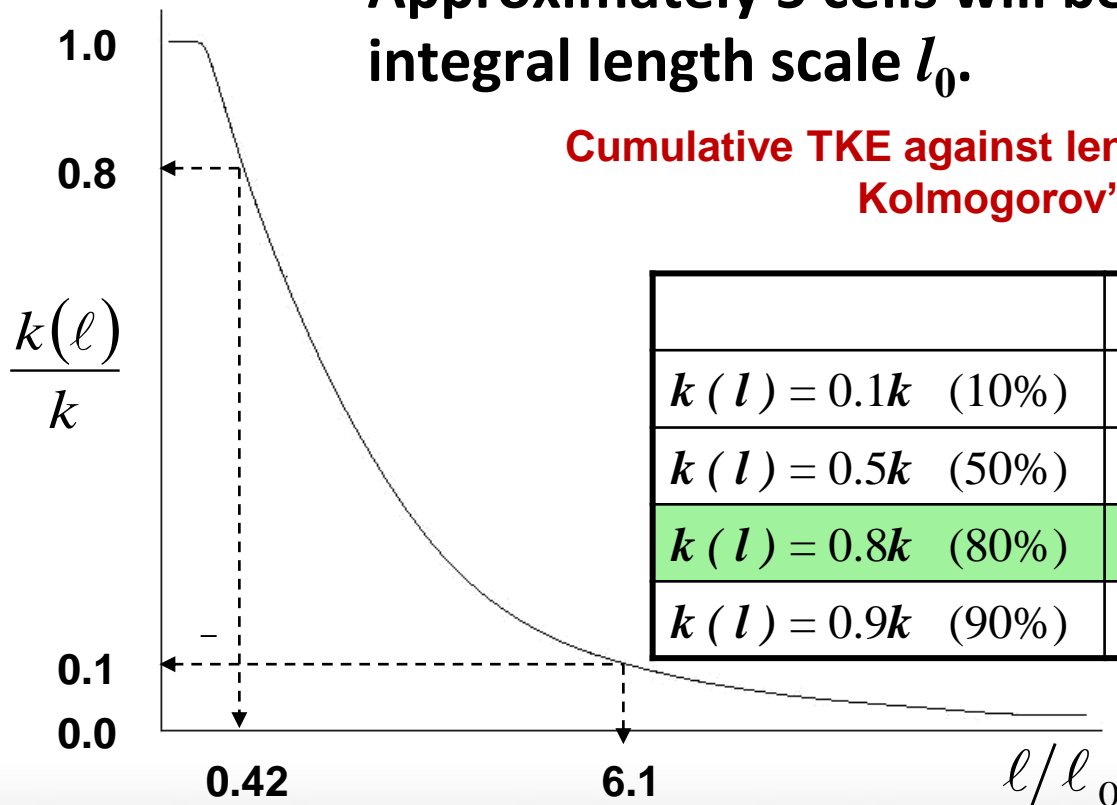
	$l\,/\,l_0$	$l_0\,/\Delta$
$k\,(\,l\,) = 0.1k$ (10%)	6.10	0.33
$k\,(\,l\,) = 0.5k$ (50%)	1.6	1.25
$k\,(\,l\,) = 0.8k$ (80%)	0.42	4.8
$k\,(\,l\,) = 0.9k$ (90%)	0.16	12.5

1.0
0.8
0.1
0.0

0.42   6.1

$\ell\,/\,\ell_0$

ANSYS

Mesh Resolution in LES (4)

- **It is a good practice to have at least** 5 cells across l_o to resolve 80% **of the** k **spectrum and** 10-12 cells per l_o can warrant a better 90% **resolution of the energy spectrum in the regions of interest.**

- **There are a number of methods to perform an assessment of the resolved** k **spectrum. The quickest and easiest assessment is based on the precursor RANS simulation*.**
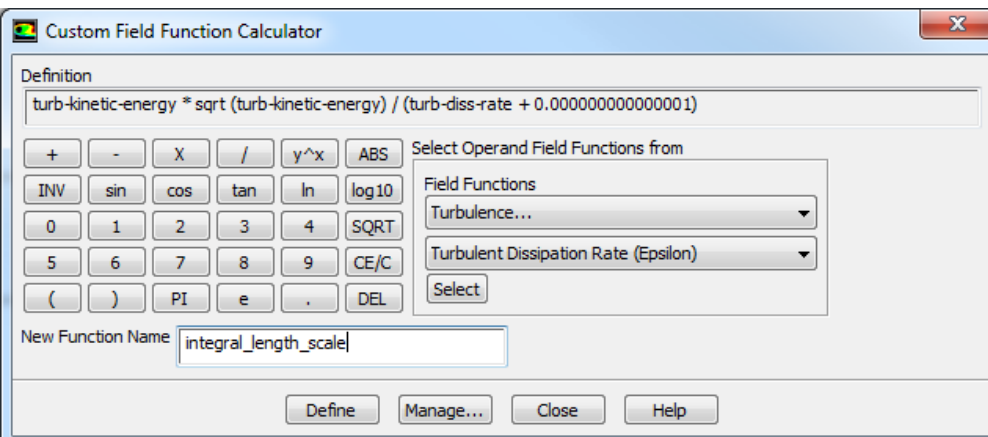
- **The integral length scale** l_o **is a local quantity and a custom-field function (CFF) can be evaluated using** k, ε **(or** ω**) values from the RANS simulation.**
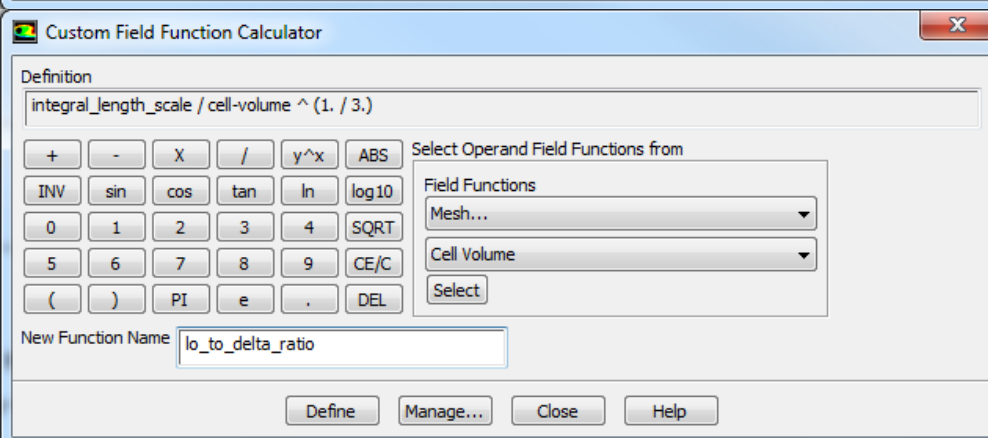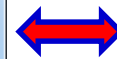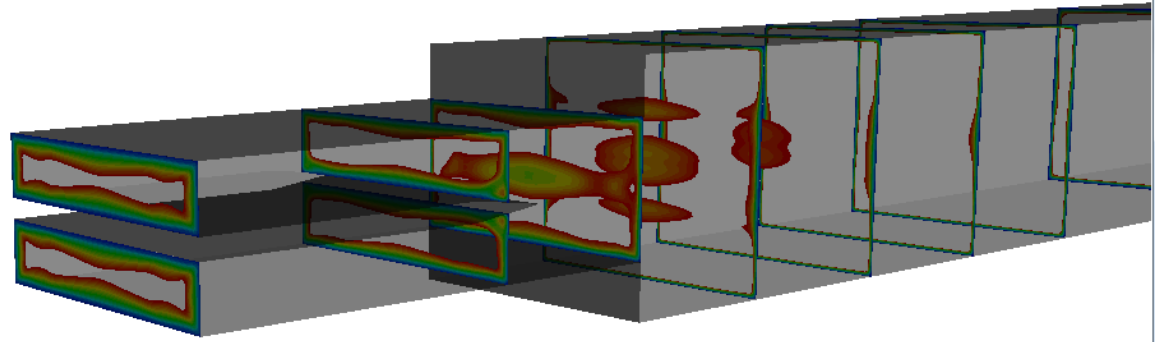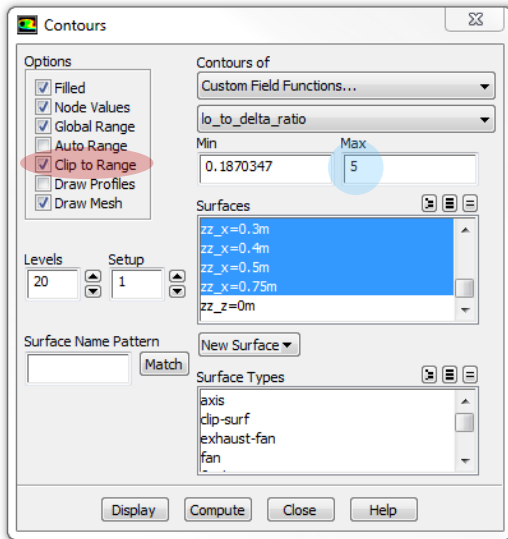
- **The number of cells per** l_o **can be assessed by assuming that**

$$\Delta \approx \sqrt[3]{(Cell\ Volume)} \quad \text{**}$$

- **Another CFF of** l_o / Δ **will return the number of cells per** l_o
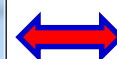
* − Might not be accurate due to differences between RANS & LES.
**− Accurate if the aspect ratios are modest (not far from 1)

Mesh Resolution in LES (5)

- Contours showing the ratio l_0/Δ need to be plotted at different predefined surfaces.
- The upper values of l_0/Δ can be clipped so that the well-resolved areas do not appear and the not-so-well resolved regions could be identified easily.

Under-resolved Mixing Layer
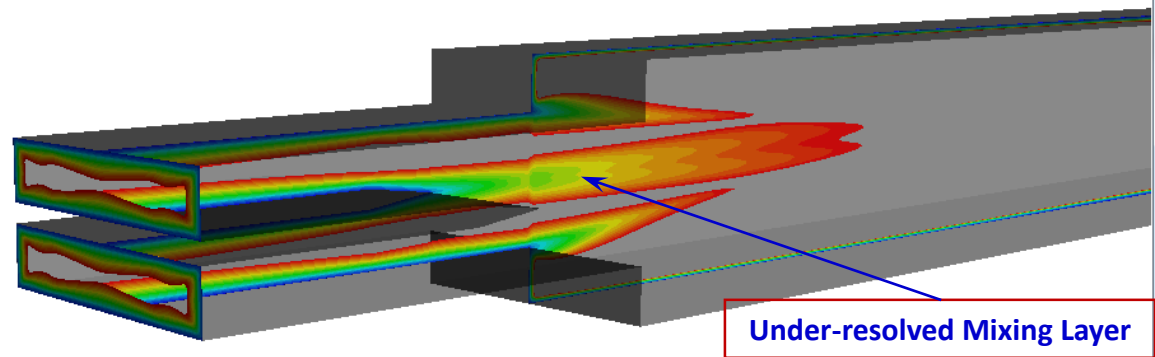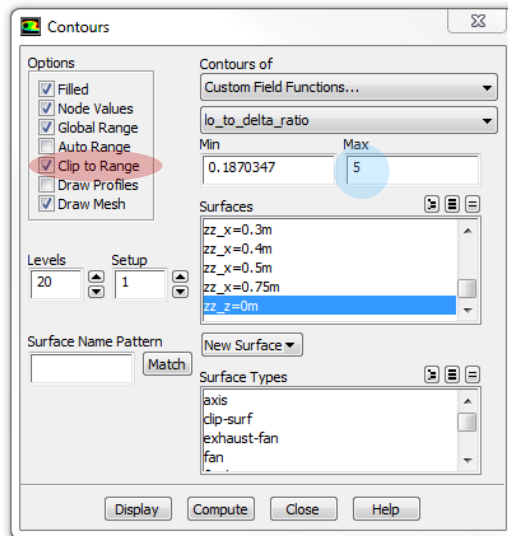
- Critical regions of a particular interest will need remeshing or local mesh adaption, if appear as under-resolved.
- The near-wall regions always pose challenges to LES and a number of techniques are available to address that.

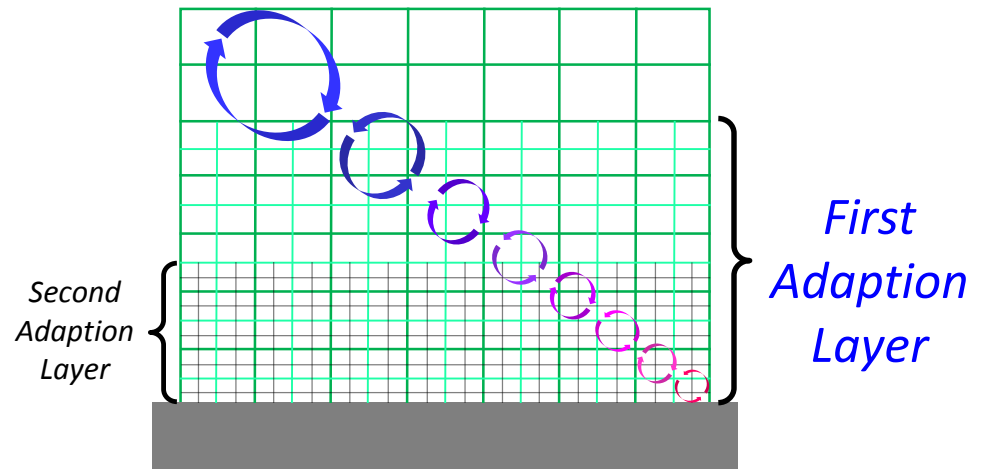# LES: Walls & Near-Wall Resolution

- **In LES, unlike in RANS, both the wall-normal and wall-parallel (stream-wise & span-wise) spacing must also decrease to resolve smaller eddies, so the density of grid points should, ideally, increase in all 3 directions as a wall is approached.**

- **Conventional RANS meshes can still be used in LES, if:**
  - The local aspect ratios are not too high. An order of magnitude can still be acceptable if the near-wall behaviour is not of primary importance and influence.
  - The outer region away from walls is sufficiently well resolved.

- **Hybrid approaches such as DES & SAS need to be considered if higher near-wall aspect ratios are unavoidable.**

Second Adaption Layer

First Adaption Layer
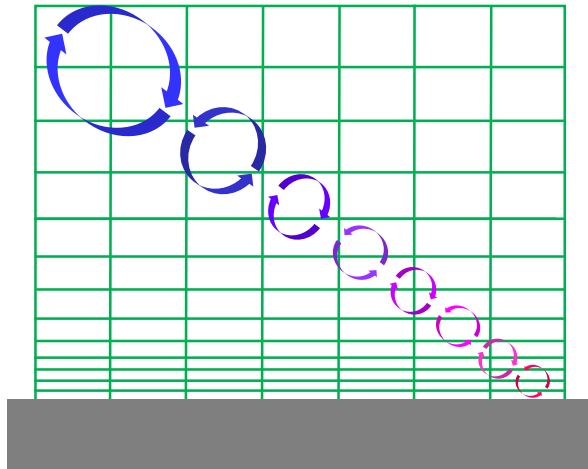
Conventional grid compression: high aspect ratios near the wall
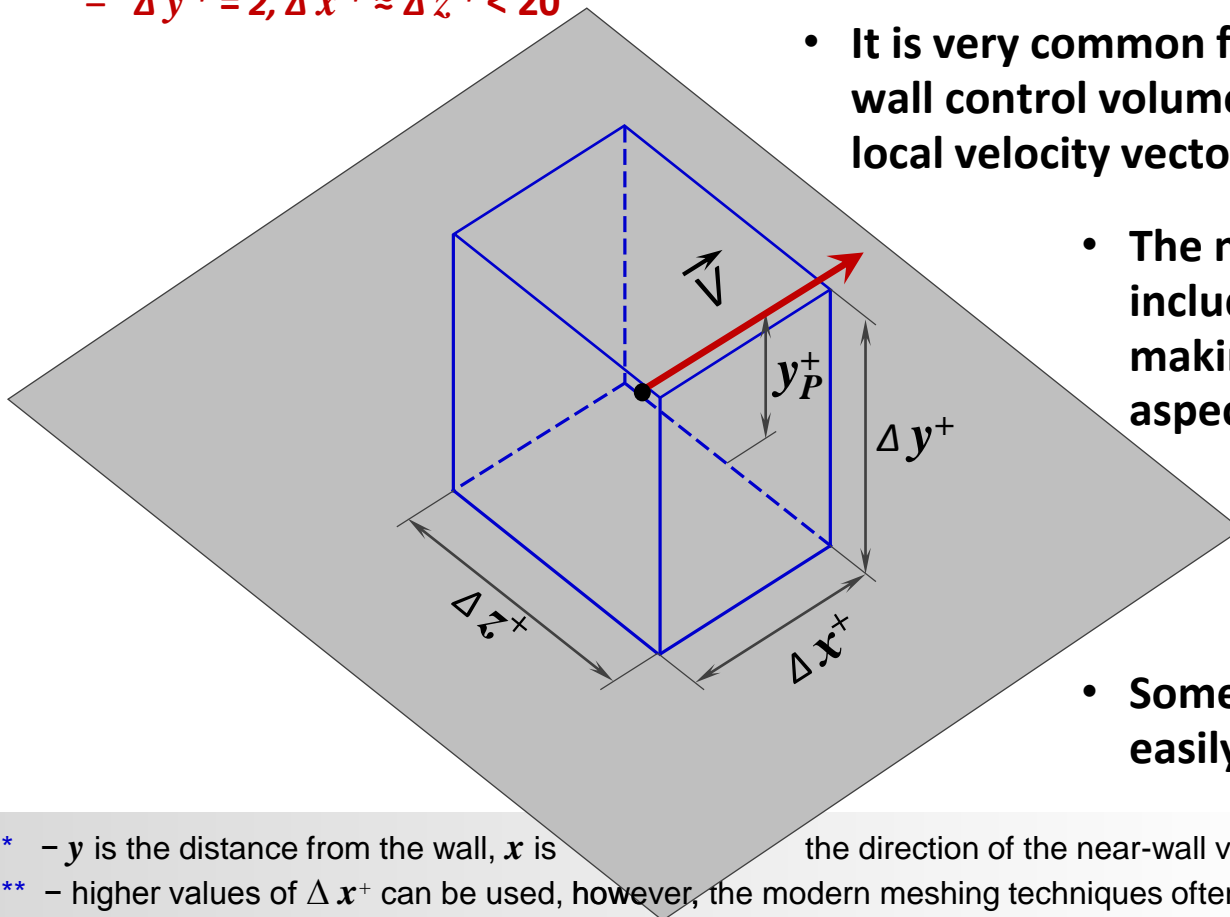
Adapted mesh: Possible solution to preserve the aspect ratio

LES: Walls & Near-Wall Resolution (2)

- **The ideal near-wall mesh resolution in LES is $y_p^+ = 1$ ($\Delta y^+ = 2$) and aspect ratio of 1.**
  - $\Delta y^+ = \Delta x^+ = \Delta z^+ \leq 2$ *

- **Higher aspect ratios can still return good results in simple shear flows.**
  - $\Delta y^+ = 2$, $\Delta x^+ \approx \Delta z^+ < 20$ **

- **It is very common for the orientation of the near-wall control volumes to be misaligned with the local velocity vectors.**

- **The near-wall control volumes can include prisms & pyramids, thus making the clear evaluations of the aspect ratios based on the directionality of the velocity vectors rather difficult.**

$\vec{V}$

y_P^+

Δy^+

Δz^+

Δx^+

- **Some rough assessments can be easily made and are sufficient.**

* − y is the distance from the wall, x is   the direction of the near-wall velocity vector, z is normal to x & y
** − higher values of Δx^+ can be used, however, the modern meshing techniques often result in the cell face aspect ratios being ~ 1
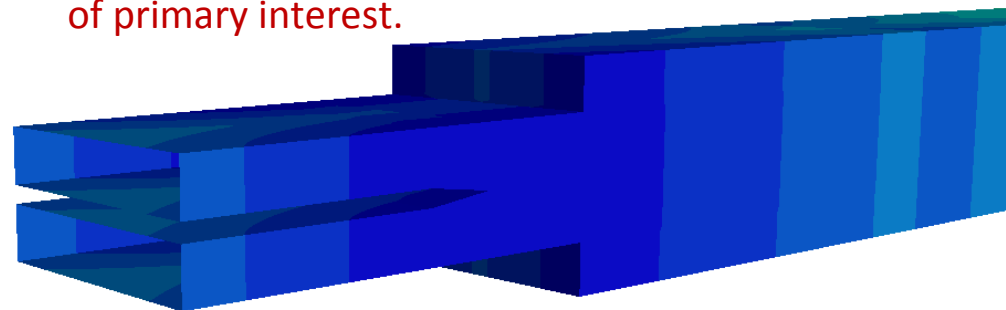
LES: Walls & Near-Wall Resolution (3)

- **The suitability of the geometrical near-wall cell characteristics for LES simulation can be obtained directly from the mesh analysis.**
  - The aspect ratios between the non-dimensional coordinates x^+, y^+ & z^+ and physical coordinates x, y & z are exactly the same for any given near-wall cell:

$$\frac{x^+}{x} = \frac{y^+}{y} = \frac{z^+}{z}$$

- **Meshing applications usually have adequate tools for mesh diagnostics.**

- **One can use ANSYS Fluent for the same purpose of the near-wall mesh diagnostics**

- **First the aspect ratios[*] of the wall-surface face sizes Δx / Δz can be postprocessed.**

– Ideally, these values should be in the range of [0.5;2] in the regions of primary interest.

[*] – The "Aspect Ratio" is not available by default, a TUI command needs to be executed first, so that more postprocessing quantities become available

```
> mesh check-verbosity 2
```
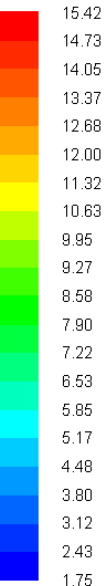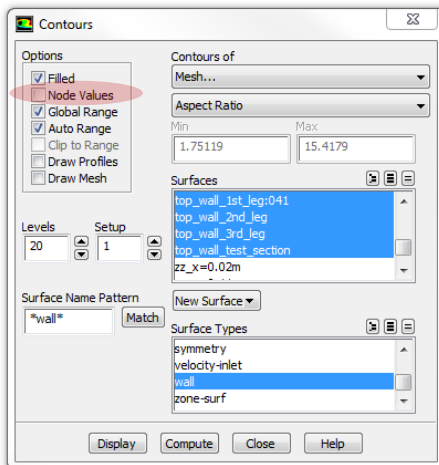
LES: Walls & Near-Wall Resolution (4)

- **Then the ratio of a typical cell-base length scale (*Δx or Δz* or their average) to the cell height *Δy* should be evaluated:**

  - This characteristic is not readily available and need to be evaluated via a custom-field function

  - Values ~ 1 are ideal

  - Values < 10 can be acceptable
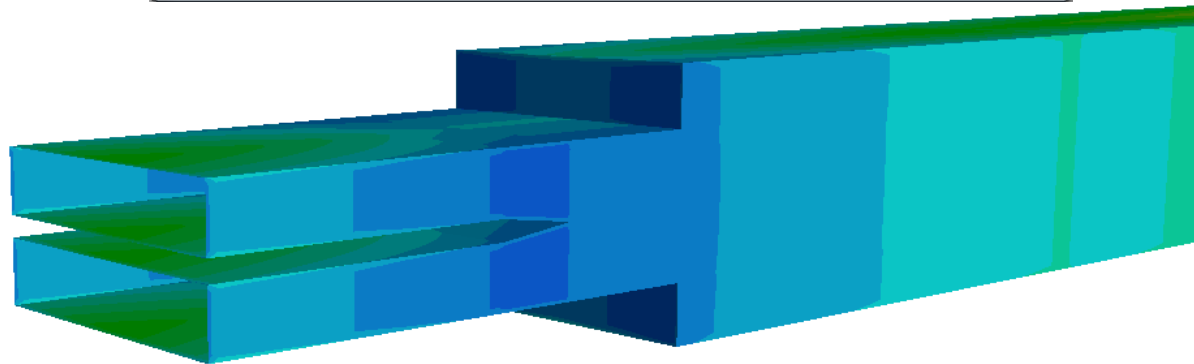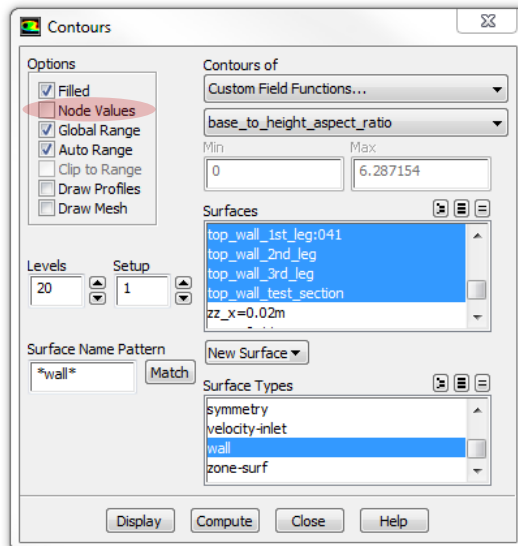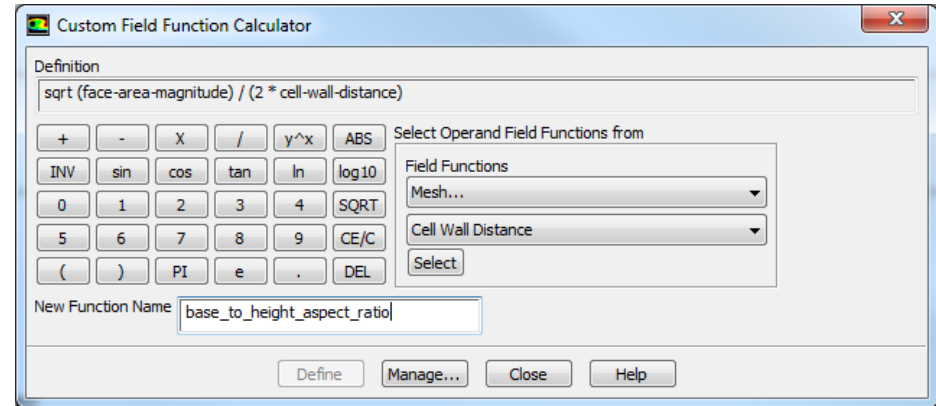
- Similarly, the approximate values of Δx^+ and Δz^+ can be evaluated via CFF as

  `sqrt(face-area-magnitude)*y-plus/cell-wall-distance`
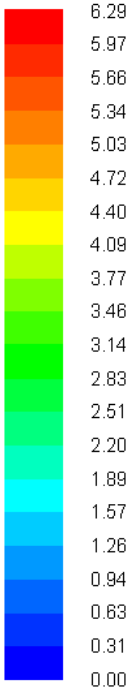
LES: Walls & Near-Wall Resolution (5)

- **The wall-normal y^+ values can be also plotted. They show how fine the near-wall cells are.**

- **The region between the first near-wall plane of cells and the outer flow must be adequately resolved.**
  - It is recommended to have around 30 cells across the boundary layer.
  - The assessment and tracking of the boundary layer thickness is impractical in real-life applications.
  - Boundary layers do not exist in the vicinity of the impingement, separation & reattachment regions.

- **It is recommended to create at least 20 inflation layers with y^+ =1 for the first near-wall cell and the expansion rate of 1.15. This approach is good for:**
  - DES or SAS
  - LES as long as:
    - the near-wall aspect ratio is not very high, or
    - stretched near-wall mesh exists far away from the regions of main interest
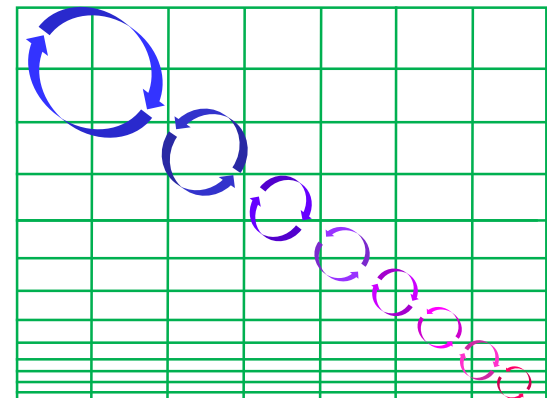
LES: Walls & Near-Wall Resolution (6)

- **Nested Adaption is a more LES-friendly meshing approach (albeit more expensive)**

- **For the well-resolved near-wall LES with $y^+ =1$ for the first near-wall cell one needs 5-6 consecutive adaption layers with 6 adapted cells at each layer.**

  - An approximate region of $y^+= 300$ should be identified from a precursor RANS simulation.
  - A fairly uniform, preferably hexahedral, mesh needs to be created with the intention of having 9-12 cells across that near-wall $y^+ \in [0; 300]$ region
  - The first ½ of that near-wall region (i.e. 4-6 cells) needs to be adapted.
  - Then a further ½ of this adapted layer needs to be adapted again.
  - In total 5 or 6 consecutive adaptions might be needed to achieve $y^+ =1$ for the first near-wall row of cells

- **A similar mesh distribution and refinement can be achieved using a very efficient Cut-Cell Meshing technology.**

- **Please note that the mesh adaption is not suitable for Cut-Cell Meshing and the required mesh distribution should be achieved via imposed mesh controls.**

Second Adaption Layer

First Adaption Layer
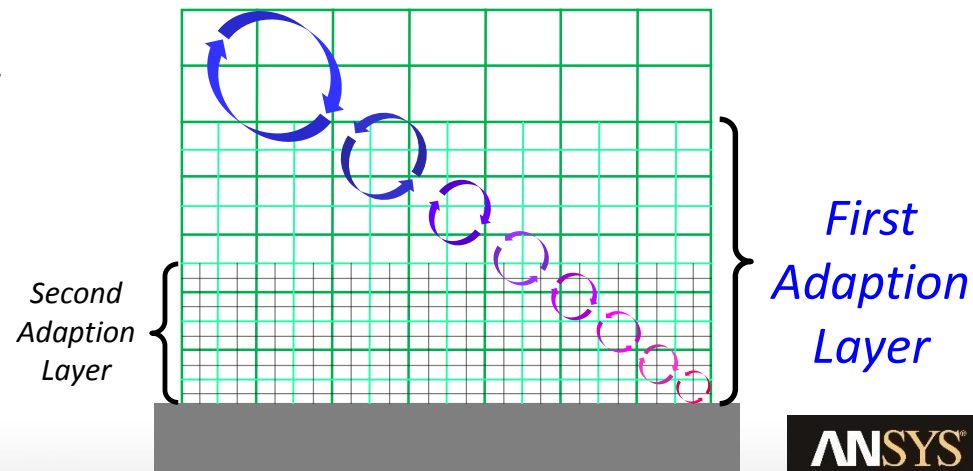
LES: Walls & Near-Wall Resolution (7)

- Please note that for fluids with Prandtl number > 1, the $y_P^+ = 1$ requirement for the first near-wall cell is no longer sufficient as the thermal boundary layer and thermal diffusion sublayer become thinner than the corresponding hydrodynamic layers:

$$\frac{\delta_{\text{hydrodynamic b.l.}}}{\delta_{\text{thermal b.l.}}} \approx \frac{y_{\text{viscous s.l.}}}{y_{\text{thermal s.l.}}} \approx \sqrt{Pr} \rightarrow y_P^+ = \frac{1}{\sqrt{Pr}}$$

- A similar rule applies for very well-resolved near-wall or free-surface LES with multispecies transport if the Schmidt number is considerably larger than unity.

$$y_P^+ \sim \frac{1}{\sqrt[3]{Sc}}$$

- The thinnest of the diffusion sublayers need to be resolved.

What happens in LES if the smallest near-wall cell size is well above the $y_P^+ = 1$ requirement ?

      May 23, 2016   **ANSYS Confidential**

ANSYS®

LES: Walls & Near-Wall Resolution (8)

- There are situations where either the well-resolved near-wall LES cannot be afforded or is not needed, whilst there would be benefits of having the resolved eddies in the mean flow.

- In such situations, LES wall functions are used.

- There is no need to choose between the wall-treatment options for various y_P^+.

- The LES wall functions are applicable for the whole range of y_P^+ below 300. There are no additional stability and accuracy problems in applying the LES wall functions across the buffer layer (unlike in RANS, where y^+=11.2 value introduces some abrupt changes to the solved turbulence equations).

- The finer the mesh the better it is *i.e.* y_P^+ = 10 would be better than y_P^+ = 50.

- **Fewer or no adaption layers needed if the LES wall function were used.**

1st Adaption Layer

2nd Adaption Layer
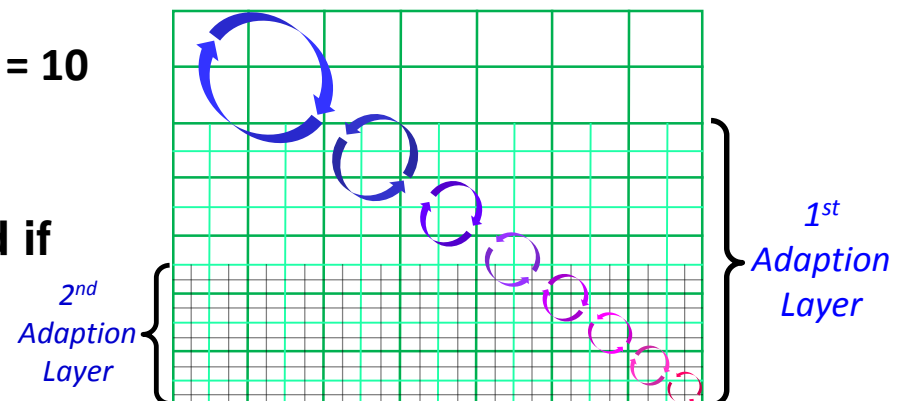
ANSYS®

LES: Walls & Near-Wall Resolution (9)

- **Choice of LES Wall Functions in ANSYS Fluent:**

  - **By default the standard blended log-law type wall function is used for LES.**
    - The blend of the linear and logarithmic laws of the wall is used to relate instantaneous wall shear stress to wall parallel instantaneous velocity at the centroid of wall adjacent cells.

  - **Werner-Wengle wall functions are also available through TUI.**

    ```
    /define/models/viscous/near-wall-treatment/werner-wengle-wall-fn?
    ```
    - The blend of the linear and the power laws of the wall is used.

- **The combination of the subgrid scale (SGS) WALE (or Dynamic) model, giving correct levels of SGS viscosity in the near-wall regions and Werner-Wengle near-wall approximation is very effective in returning the closest fidelity at moderate $y_P{}^+$ values[*] to the highly resolved solution in some scenarios[**].**

- **Please note that for higher $y_P{}^+$ values the conventional LES & RANS wall functions do suffer from the very same inherent limitations and can only return adequate results for near-wall flows where turbulence is in the equilibrium state i.e. in situations far removed from practical applications.**

[*] – Provided that the $y_P{}^+$ values are below 20 (*i.e.* $\Delta y^+ \leq 40$).
[**] – Temmerman & Leschziner, "Large Eddy Simulation of Separated Flow in a Streamwise Periodic Channel Constriction", 2001
      http://www.ercoftac.org/fileadmin/user_upload/bigfiles/sig15/database/9.2/TSFP2_09032001.pdf

Cost of LES: Time-Step Size

- **The temporal resolution should match or exceed the spatial resolution in LES**

  - Let's say we have a cell with largest edge length of Δx and local average velocity, U

  - It takes a time interval Δt for the flow to travel across the cell

  - The time step should be small enough to provide an adequate temporal resolution of the flow as it passes through the cell

  $$\Delta t \leq \frac{\Delta x}{U}$$

  - The real velocity can be higher than the averaged velocity

  - Pre-cursor RANS simulation is used for the assessment of Δt

- **Good practice to account for differences between instantaneous and averaged velocities as well as for the errors introduced from RANS by**

  $$\Delta t \approx \frac{\Delta x}{2U} \quad \Rightarrow \quad Courant\ Number = \frac{U\Delta t}{\Delta x} \approx 0.5$$

ANSYS®

Cost of LES: Time-Step Size (2)

- **Custom-Field Function for local time scales can be**

  – **Evaluated**

  – **Postprocessed**

  – **Minimum time scale can be calculated and the rounded value for the required time step can be obtained**

Custom Field Function Calculator

Definition

cell-volume ^ (1. / 3.) / (|V| + 0.000000000000001)

+	-	X	/	y^x	ABS
INV	sin	cos	tan	ln	log10
0	1	2	3	4	SQRT
5	6	7	8	9	CE/C
(	)	PI	e	.	DEL

Select Operand Field Functions from

Field Functions

Velocity...

Velocity Magnitude

Select

New Function Name time_scale

Define   Manage...   Close   Help

5.00e-02
4.75e-02
4.50e-02
4.25e-02
4.00e-02
3.75e-02
3.50e-02
3.26e-02
3.01e-02
2.76e-02
2.51e-02
2.26e-02
2.01e-02
1.76e-02
1.51e-02
1.26e-02
1.01e-02
7.63e-03
5.14e-03
2.65e-03
1.56e-04

Contours

Options
☑ Filled
☑ Node Values
☑ Global Range
☐ Auto Range
☐ Clip to Range
☐ Draw Profiles
☑ Draw Mesh

Levels   Setup
20   1

Surface Name Pattern

Match

Display   Compute   Close   Help

Contours of
Custom Field Functions...
time_scale

Min   Max
0.000155805   0.05

Surfaces
zz_x=0.3m
zz_x=0.4m
zz_x=0.5m
zz_x=0.75m
zz_z=0m

New Surface ▼

Surface Types
axis
clip-surf
exhaust-fan
fan

Please note that a larger Δt can be chosen if:

- The minimum time scales are detected far away from the regions of main interest.
- The regions with smallest time scales are not posing influence onto the rest of the domain.

Mesh Resolution for Hybrid Approaches

- The cost of resolving the near-wall region with LES increases very quickly and can become prohibitive as the Reynolds number increases.

- LES wall functions can be satisfactory in some situations.

- In many cases, the hybrid RANS/LES models, such as DES, WMLES & SAS can be beneficial by offering the LES resolution in the outer regions and RANS modelling in the near-wall regions*.

- The main savings for DES, WMLES & SAS come form:
  - Much higher, RANS-like, aspect ratios can be used close to walls (i.e. lower mesh count).
  - Larger Δt can be used as the expensive-to-resolve near-wall region in (D)DES is occupied by RANS while the stringent $Co \approx 1$ limitation remains applicable for the outer LES region only.
  - SAS allows <u>larger Δt</u> that will be matched by the spatial resolution of the resolved eddies.

- **Please note** that $y_P^+ \leq 1$ resolution must still be applied in wall-normal direction and there will be hardly any benefit from DES, WMLES & SAS if this criteria is broken.

- Embedded LES (ELES) is a different hybrid approach where the RANS and LES regions have a predefined interface. Specific RANS or LES mesh requirements apply in each of these regions. The mesh interface can be non-conformal.

* − For more detailed information please see: Spalart, P.G., "Young-Persons's Guide to Detached-eddy Simulation Grids", 2001, NASA/CR-2001-211032

ANSYS

Meshing Methods for LES:

- **General mesh recommendations**
  - **Use hex meshes for the best accuracy.**
  - **Take advantage of hybrid meshing capability and local mesh refinement.**
  - **Polyhedral meshes can also be used.**

- **ANSYS Meshing methods for LES**
  - **Multizone method (preferred method)**
    - Based on blocking approach of ANSYS ICEM CFD Hexa.
    - High quality mesh, but might require some manual decomposition.
    - Might experience some robustness issues.
  - **Assembly Meshing (Cut-Cell approach)**
    - Very robust algorithm. Good quality mesh, but step changes usually present in the domain. Not suitable for further adaption.

- **ANSYS Fluent Meshing methods for LES**
  - **Cut-Cell Method**   – **Hexcore Metod**

ANSYS®

LES: Numerical Discretisation

- **Spatial discretisation.**
  - **Use with CD or BCD for momentum equation.**
    - For ELES, 2nd order upwind can be used for the momentum equation in RANS zones and for turbulence transport equations
  - **Use the Least Squares cell-based or Green-Gauss node-based gradient option.**
  - **When undershoot/overshoot of the solution fields becomes an issue, use high-order upwind schemes (QUICK, MUSCL, SOU) for scalars.**

- **Time discretisation.**
  - **Use the second-order scheme.**
  - **Use the NITA/fractional-step method for incompressible or weakly compressible flows and make sure that the Δt assessments are conservative.**
  - **With the fully-iterative scheme, use SIMPLEC with high URF's (0.8– 0.9) for pressure and momentum equations.**

ANSYS®

Choice of the Subgrid-Scale (SGS) Modelling:

- **SGS modelling. Points to consider:**
  - **WALE** or the **Dynamic Smagorinsky** model are good starting points.
  - **Dynamic TKE** model can potentially benefit highly non-equilibrium flows and reacting flows.
  - **Dynamic TKE** model is the only SGS model from ANSYS arsenal that can easily show how much of modelled spectrum is resolved and how much is modelled.
  - **Dynamic Smagorinsky** & **Dynamic TKE** models are much better in recovering the real turbulence levels from the synthetic turbulence levels imposed at the inlets.
  - Consider **DDES** or other hybrid methods for external high-Re wall-bounded flows with large flow separation.
  - **WMLES** is effectively a 0-equation algebraic model DES which is placed under SGS models. Similarly to other DES models, it is suitable to situations with high near-wall cell aspect ratios and requires $y_P^+ \leq 1$.

ANSYS®

Postprocessing

- **The use of "Q-criterion" is one of the best ways to visualise the turbulent vortical structures**[*].

  - First assess the range of calculated Q-criterion. This can be done via either the Display Contours or via the Iso-Surface panel.

  - Create a set of iso-surfaces for a number of fixed positive values as shown in the picture below. Please note that the best visualisation can be achieved with the iso-surfaces created in the range of 5% - 25% of the positive range i.e. if the maximum value, say, 4,000,000, then consider creating more iso-surfaces within the sub-million range (800,000; 600,000; 500,000; 400,000 etc.)
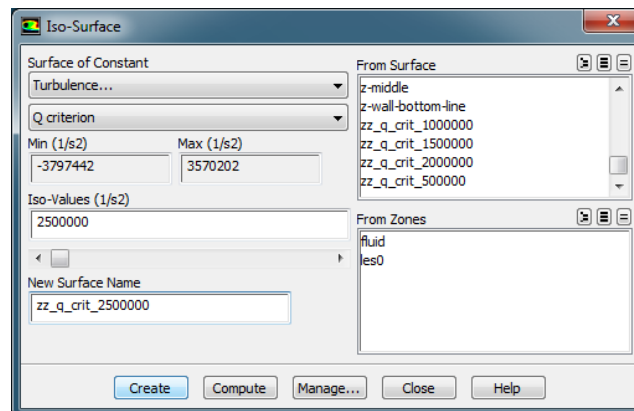
Please consider naming all the new surfaces with the "zz_" prefix so that all the new postprocessing surfaces appear at the bottom of the list, for convenience.

  - Visualise the contours of either the velocity magnitude or vorticity on different newly created Q-criterion iso-surfaces and select the most informative pictures.

[*] – Please see a separate Solution No. 2041496 on this subject: : "Vortex Visualisation in ANSYS Fluent, Part 1: SRS"

Postprocessing (2)

- **Please refrain from using the "Normalised Q-criterion" for general visualisation of the vortical structures inside the flow domain.**
    - **This quantity is best suited for visualising and analysing the dynamics of synthetically generated eddies at the inlet to the LES domain where either the Vortex Method or Spectral Synthesizer is used.**

- **Typical contour plots of different quantities at different cross-sections in the domain are as informative in LES as they are in RANS:**
    - **Instantaneous quantities can show their ranges and can indicate whether the flow has purged through the domain or not i.e. whether it is a good time to start gathering the unsteady statistics or not (please note that the simple monitor points are better for this purpose).**
    - **Averaged quantities become available when the "Data Sampling for Time Statistics" option is ticked in the "Run Calculation" panel. The smoothness of the statistical plots can indicate how well averaged the solution is*.**

- **A sufficient amount of monitor points in the key areas of the domain as well as in the proximity of the outlet are important for ensuring the quality of the gathered unsteady statistics.**

* – Please note that it takes considerably longer to gather reliable statistics for the rms values than for the mean values

ANSYS

Postprocessing (3)

- The instantaneous vector plots are hard to read and understand and, therefore, there is very little point in analysing them

- The averaged velocity vectors are not available by default, however, there is a way to visualise them by using the "Custom Vectors" option from the Vectors Panel.
    - Set up the custom vector* as shown in the picture below
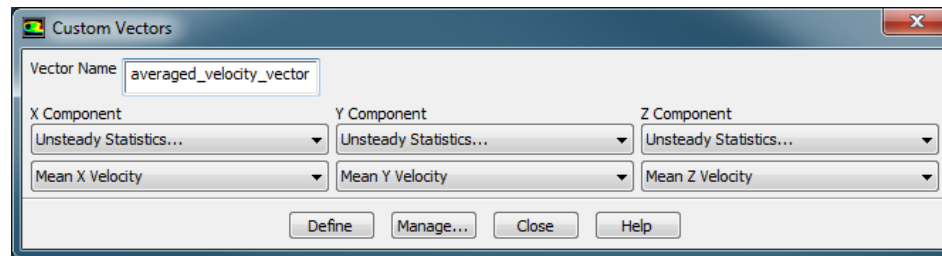
- A number of custom-field functions (CFFs) for instantaneous as well as for the averaged quantities can be defined and plotted.

- Sampling can be gathered for CFFs that are based on instantaneous values from the "Sampling Options" tab in the "Run Calculation" panel.
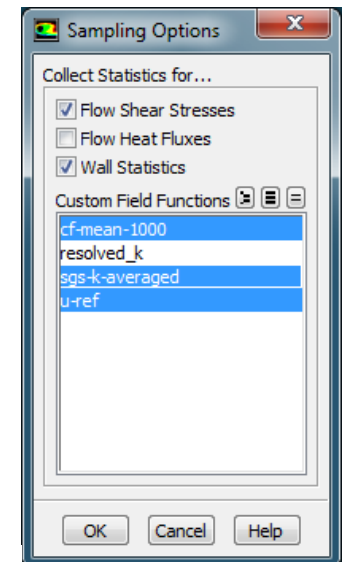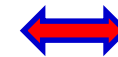
* – Please note that all created CFFs as well as the Custom Vectors can be saved and transferred to a different case file and, therefore, there is no need to evaluate them for each and every case. It can be done just once!

Postprocessing (4)

- **The resolved part of the turbulent kinetic energy can be calculated via a CFF using the rms velocity values from the gathered unsteady statistics.**
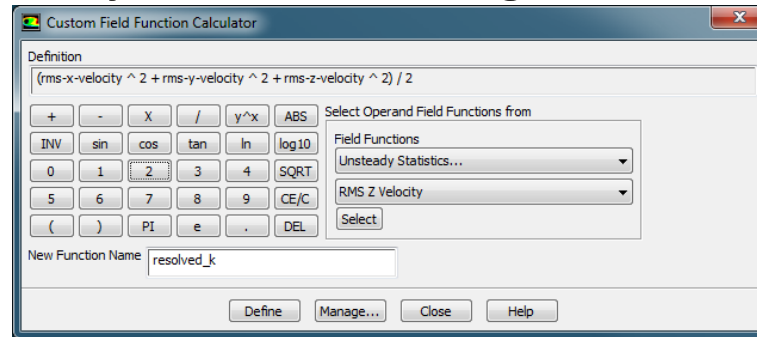
- **The obtained values should be compared with the turbulent kinetic energy values from the precursor RANS simulation**

  - **This will give some rough indication on how much energy is modelled by taking the difference between the total amount of turbulent kinetic energy in RANS and resolved turbulent kinetic energy in LES (DES, SAS)**

- **If the Dynamic Kinetic Energy Transport SGS model is used in LES then the ratio of resolved "Subgrid Kinetic Energy" vs total turbulent kinetic energy can be evaluated directly by creating a CFF:**

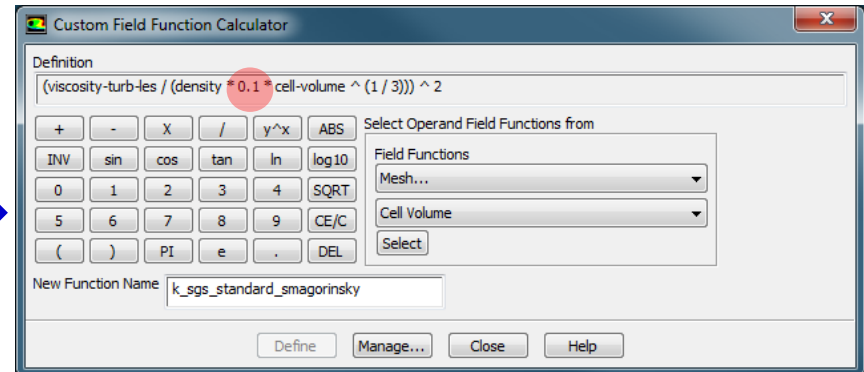 resolved_spectrum = resolved_k / (resolved_k + sgs_k_averaged[*])

* – Fluent does not gather statistics for the Subgrid Kinetic Energy automatically and, therefore, a custom-field function **sgs_k_averaged** needs to be defined first so that sampling can be performed. This can only be done for DKE SGS!

Postprocessing (5)

- The rest of the available SGS models do not solve a separate equation for the turbulent kinetic energy.

- The modelled component of the turbulent kinetic energy can be approximated from the subgrid scale viscosity and length scale.

For Standard Smagorinsky model:

$$k_{sgs} = \left(\frac{\mu_{sgs}}{\rho l_{sgs}}\right)^2 = \left(\frac{\mu_{sgs}}{\rho \min[\kappa y, C_s \Delta]}\right)^2$$

Custom Field Function Calculator

Definition

(viscosity-turb-les / (density * 0.1 * cell-volume ^ (1 / 3))) ^ 2

New Function Name: k_sgs_standard_smagorinsky

- Please note that for the Standard Smagorinsky[*] model the choice of the C_s constant value should be consistent with the value provided in the model set up (default = 0.1). Strictly speaking, the value for the SGS length scale $l_{sgs} = \min[\kappa y, C_s \Delta]$ includes a logical "min" expression and, therefore, it is best approached via a simple UDF[**] instead of the CFF.
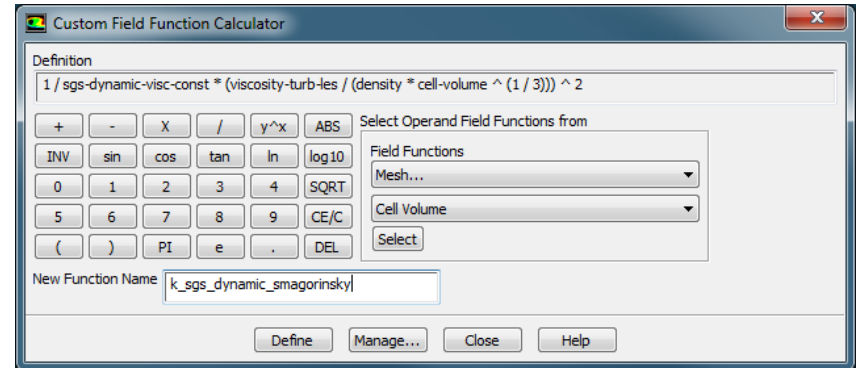
[*] − Please note that WALE model uses a different constant C_w = 0.325 instead of C_s =0.1
[*] − WMLES uses a different C_s=0.2 and the reconstruction of the filter expression is different and would require a UDF.

Postprocessing (6)

- **A similar procedure can be established for the Dynamic Smagorinsky SGS model:**

$$k_{sgs} = \left(\frac{\mu_{sgs}}{\rho l_{sgs}}\right)^2 = \left(\frac{\mu_{sgs}}{\rho\, C_{DS}\Delta}\right)^2 = \frac{1}{C}\left(\frac{\mu_{sgs}}{\rho\Delta}\right)^2$$

- **Typically, the subgrid dynamic constant C_{DS} is the one that gets mentioned and compared to the C_S constant of the Standard Smagorinsky model. The value of C_{DS} is allowed to vary from 0 to 0.23**

  - **Please note that at present (R17) ANSYS Fluent uses the value of $C = (C_{DS})^2$ for postprocessing.**

- **The above quantity k_{sgs} should not be mistaken for "Subtest Kinetic Energy" which is a quantity associated with a different test filter.**

- **The ratio of resolved vs modelled turbulent kinetic energy can be evaluated directly by creating a CFF:**

resolved_spectrum = resolved_k / (resolved_k + k_sgs[*])

*   –   Statistics need to be gathered for the custom-field function **k_sgs** (either k_sgs_standard_smagorinsky or k_sgs_dynamic_smagorinsky)

Postprocessing (7)

- The subgrid viscosity ratio can be used to come up with some approximate assessment for the quality of the scale-resolved simulations too.

- The LES index of quality was proposed by Celik *et al* [*]:

$$LESIQ_v = \frac{1}{1 + 0.05\left[\frac{(\mu + \mu_{sgs})}{\mu}\right]^{0.53}}$$

μ_{sgs}/μ	5	20
$LESIQ_v$	0.89	0.8

- This is a number between 0 and 1. The constants are calibrated such that the index is perceived similar to the ratio of resolved to total turbulent kinetic energy i.e. the higher the value the better the resolution is (0.8 or above).

- This is a single-point indicative measure which is not particularly accurate for anisotropic turbulence and need to be used with caution.

- A custom-field function can be evaluated for this index

- The SGS viscosity ratio can be compared to the turbulent viscosity ratio from the precursor RANS. It should be much less.
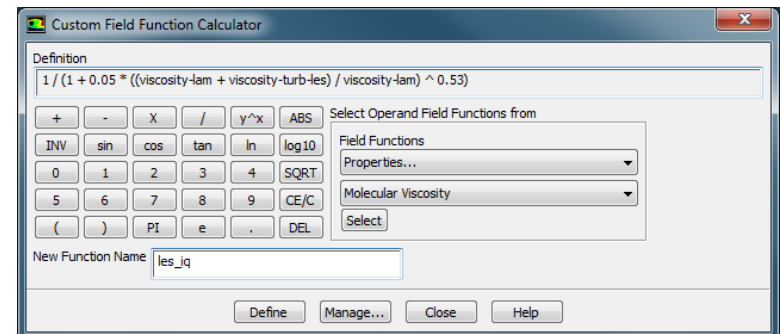
[*] – Celik, Cehreli & Yavuz. Index of resolution quality for large eddy simulations. ASME Journal of Fluids Engineering, 127:949-958, 2005

Postprocessing (8)

- **The balance between the RANS and LES regions of DES can be assessed by looking at the contours of DES TKE Dissipation Multiplier:**

  - **For DDES-RKE & DDES-SA: the value of 1 indicates LES region and the value of 0 shows the RANS region.**

  - **There is no automatic option to distinguish the RANS and LES regions with DES-RKE & DES-SA. For Release 12 and earlier versions there was an option called "Relative Length Scale" that could have been used for this purpose. This, however, can be remedied by writing a simple UDF where a UDM needs to be defined: UDM=0 for RANS, UDM=1 for LES**

    - DES-RKE: UDM = 0 if $l_{RKE} < l_{DES}$, or otherwise UDM = 1 ($l_{RKE} = k^{3/2} / \varepsilon$, $l_{DES} = C_{DES} \Delta^*_{max}$)
    - DES-SA: UDM = 0 if $d < l_{DES}$, or otherwise UDM = 1 (d – distance from wall, $l_{DES} = C_{DES} \Delta^*_{max}$)

  - **For (D)DES-SST : the value of 1 indicates the RANS region and values in excess of 1 indicate LES regions. Please disable the "Auto Range" for the contours of "Turbulence" → "DES TKE Dissipation Multiplier" and clip the range from the minimum calculated value to the maximum of 1.01 – The RANS part of the domain will be visible on selected surfaces. Similarly, the clipped values from 1.01 to the maximum range value will show the LES region.**

---

* – Please note that the filter width in DES, Δ_{max}, is based on the maximum local grid spacing $\Delta_{max} = \max [\Delta x, \Delta y, \Delta z]$ instead of $\Delta = $ (Cell Volume)$^{1/3}$, whereas $C_{DES} = 0.61$ for DES-RKE, $C_{DES} = 0.65$ for DES-SA

Summary of SRS Models

- ## LES

  - **Suitable for outer flows as well as for wall-bounded flows at low and moderate Reynolds numbers**

  - **Well-resolved LES ($y^+ \approx 1$) is not feasible if wall boundary layers are at high Re.**

    - Spatial and time resolution requirements are excessive for such flows.

    - LES with wall functions ($y^+ > 1$) can be used at high Re if accuracy in the near-wall region is not vital.

- ## DES

  - **Explicit combination of RANS and LES models:**

    - LES – in outer flow

    - RANS – in the wall adjacent regions.

  - **Allows computation of flows with weaker instabilities than needed for SAS.**

  - **Good for external aero flows, lesser success in internal flows.**

  - **Classic DES models do not have shielding i.e. there is no protection of the boundary layer: the resolved eddies can disturb the boundary layer and affect the accuracy of the RANS near-wall solution.**

  - **Grid independence in RANS regions requires shielding. Delayed DES (DDES) models prevent the resolved structures from entering into the boundary layer regions. The strength of such protection is determined by shielding functions.**

ANSYS®

Summary of SRS Models (2)

- **DES (continued)**

  - **DDES-SA & DDES-RKE models offer the predetermined shielding functions**

    - Use these flavours of DDES if you know that that the predictions with SA or RKE RANS models are superior to SST for the near-wall flow in question. Otherwise use DDES-SST.

  - **DDES-SST has a choice of 4 shielding functions**

    - SST F2 Function – Not recommended as it might delay formation of resolved structures.
    - DDES – Stronger shielding. Might struggle to pick up instability in weakly disturbed flows.
    - IDDES – Preferred option*. Picks up instabilities easier. Somewhat weaker shielding.
    - SST F1 Function – Least amount of shielding.

    Shielding Strength ↑

- **SAS**

  - **Lesser grid sensitivity in RANS zone, capable to work at higher Co numbers.**

  - **For many flows results are similar to DES.**

  - **Needs a strong flow instability to generate resolved turbulence.**

    - Stays in RANS/URANS mode if instability is missing or too weak.
    - The RANS mode of SAS might not reduce fully to the underlying RANS k-ω model.

*   – This does not mean that IDDES is always better. For example, DDES might be better in some strongly unstable flows. At times, even simple DES can give better results. For design optimisation studies, it is important to ensure consistency between the model used and the mesh resolution.

Summary of SRS Models (3)

- **ELES**
  - **Useful to limit LES zone to critical area.**
  - **Allows manual definition of LES zone.**
    - Unlike SAS or DES that depend on both the grid and the solution.
  - **Synthetic turbulence generated at RANS-LES interface.**
  - **Can also be used to trigger SAS/DES into unsteady mode.**

- **WMLES**
  - **The inner part of the boundary layer is covered by a 0-equation algebraic RANS and the central part by LES, thus reducing the LES grid requirements for high Re flows.**
  - **Less widely applicable for external flows as the 0-eq algebraic RANS model is no match to 2-equation models in DDES-SST or in DDES-RKE. It can be classified as the leanest DES.**
  - **Can return better results for internal flows when compared to other flavours of DES.**
  - **WMLES S-Ωmega (Strain-Vorticity) option should be preferred.**
  - **No shielding, eddies can penetrate into the boundary layer, the RANS region is very thin.**
  - **Boundary layer resolution:**
    - y_P^+ should be ≤ 1
    - Cell Courant number in boundary layer = 0.3. Therefore, **WMLES is not necessarily cheaper than the other flavours of DES**, that solve 1 or 2 turbulence equations across the boundary layer but are fine with the cell Courant number ~ 1.

Best-Practice Guide for Scale-Resolving Simulations

Best Practice: Scale-Resolving Simulations in ANSYS CFD

Version 2.00

F.R. Menter
ANSYS Germany GmbH

November 2015
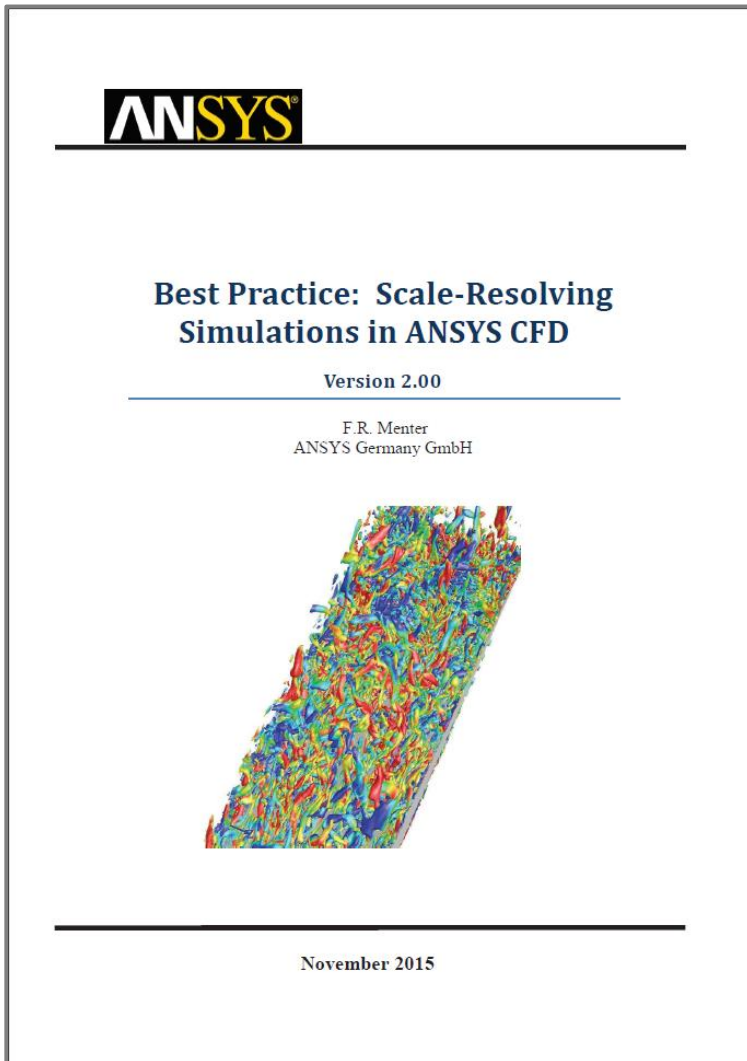
- Comprehensive & Detailed Document

- Particular Emphasis on Hybrid Approaches

- New hybrid SDES & SBES technologies explained.

- Theoretical Background Provided

- A Number of Examples Discussed

- Freely available from ANSYS Customer Portal

Additional sources are these books:

Johan Meyers
Bernard Geurts
Pierre Sagaut
Editors

ERCOFTAC Series

Quality and Reliability of Large-Eddy Simulations

Springer

Vincenzo Armenio
Bernard Geurts
Jochen Fröhlich
Editors

ERCOFTAC Series

Direct and Large-Eddy Simulation VII

Proceedings of the Seventh International ERCOFTAC
Workshop on Direct and Large-Eddy Simulation,
held at the University of Trieste, September 8–10, 2008

cost   Springer

ANSYS

Typical Mistakes

- **Use of symmetry boundary conditions is not permitted in LES**

  - Consider using either a few sets of translational periodics or pressure inlet/outlet boundary conditions or walls if appropriate. The boundary should not reflect any eddies in an unrealistic way.

- **Using PRESTO scheme for pressure spatial discretization**

- **Imposition of the Vortex Method (VM) onto flat inlet turbulence profiles for wall-bounded flows where VM interface comes in contact with the walls**

- **Using NITA from the onset of a simulation**

- **Use of the "Normalised Q-criterion" for visualisation of the turbulent eddies. Please use the non-normalised conventional "Q-criterion" instead.**

Common Questions

- How to make a correct assessment of the resolved/modelled parts of the energy spectrum, especially if the Dynamic TKE model is not used? Please see slides 13-15 and 34-37.

- How to interpret the "DES TKE Multiplier"?   Please see slide 38.

- I see some overshooting of some variable like Temperature in my LES simulation?   Please check convergence, Courant number and compare the numerical settings with the ones discussed in this document.

- How long do I have to run my simulation for taking averages and when should I start sampling? The flow in the region of interest should become statistically averageable and the best way to assess that is through the local monitor points.

- The LES velocity field looks plausible but the pressure shows very wild unrealistic oscillations.   The most common reason for that is the use of vortex method on unrealistic, typically flat, inlet profiles. Please see slides 9 & 10.

ANSYS

Common Questions:
Why am I getting worse LES results on a finer mesh?

- **There is a number of factors that influence LES results:**
  - Mesh & Filtering
  - Numerical Methods
  - Models and Submodels
  - Boundary Conditions
  - Convergence Criteria
  - Statistical Sampling

- **All of the above factors introduce certain errors into a simulation. These errors might cancel each other, at least, partially.**

- **Thus reducing one of these errors, say by using a finer mesh, might make the simulation results look worse.**

What can I do about it?

- **Make sure that your new spatial resolution is matched by an amended time step, so that the Courant number is acceptable.**

- **Consider all of the above influencing factors and mitigation measures:**
  - Numerical Scheme: CD vs BCD
  - Convergence Criteria.
  - Mesh growth rate in critical regions
  - SGS Model: Dynamic (TKE) vs WALE
  - Effect of the VM (or SS) and its inputs
  - ITA vs NITA

ANSYS®

Ways to speed up LES-type simulations (1)

- **Typically, the use of NITA for incompressible or weakly compressible flows and/or Hybrid approaches such as DDES can offer significant savings.**
  - ITA often needs to be used at the initial stages of the simulation. Therefore, NITA will need to be switched on only once the simulation showed good convergence with ITA.
  - NITA requirement for Courant number $Co \leq 1$ is more stringent than for ITA.

- **Use of ITA with a small number of iterations per time step for compressible flows.**
  - It has been proven that decent results can be achieved in some cases by reducing the number of iterations per time step from a typical value of 10 to as low value as 3. In such cases it is important to ensure that $Co \leq 1$.
  - Use PISO instead of SIMPLEC.
  - Consider doing a preliminary test comparing the averaged quantities for PISO with 10 iterations per time step and for PISO with 3 iterations per time step.

ANSYS®

Ways to speed up LES-type simulations (2)

- **Often there is a question about using Very Large Eddy Simulation VLES i.e. conventional LES on relatively coarse meshes, and with a larger time step, where only a tiny fraction of spectrum (<50%) is resolved in some parts of the flow.** **Is there a value in such analysis** or **is it plain dangerous****?**

- **This might not withstand any academic scrutiny, however, this might still be a viable engineering approach in some cases. Please see the slide "Common Questions:** **Why am I getting worse LES results on a finer mesh?"**

- **In some cases, the cancellation of errors can be very favourable for a specific set-up. Should this be _confirmed by other means_, then the VLES can be used for a similar related scenario, provided that all the settings stay the same and mesh topology/distribution is similar.**

Scenario where this might work:

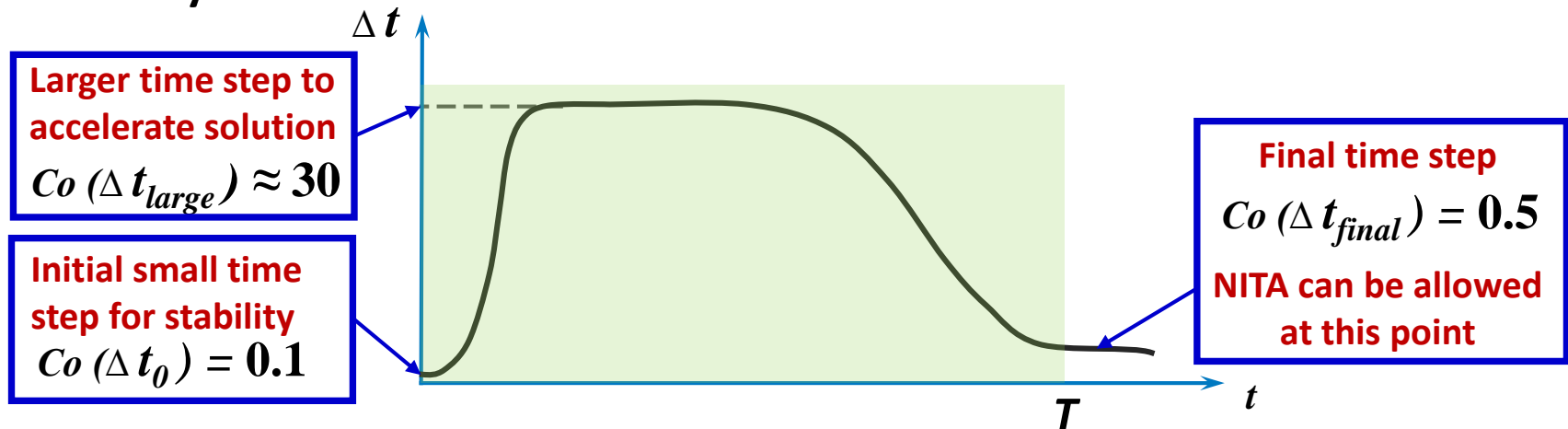- **External Aero: Prediction of the wake behind a car, wing tip vortices, urban terrain, etc…**

Typical Scenario where this will not work:

- **Heat transfer predictions at walls.**

ANSYS®

Ways to speed up LES-type simulations (3)

- **To achieve the statistically averageable state quicker one can start with a larger time step reducing it gradually to meet the Courant number requirement. This is somewhat questionable from the academic point, however, this is a sensible engineering method that can speed up the LES simulation by up to an order of magnitude on its run up to the statistically averageable state.**

- **Having the larger time step size in LES from the onset of a simulation might introduce some stability issues. Therefore, an initial ramping up might be necessary.**

Δt

Larger time step to accelerate solution
$$Co \, (\Delta t_{large}) \approx 30$$

Initial small time step for stability
$$Co \, (\Delta t_0) = 0.1$$

Final time step
$$Co \, (\Delta t_{final}) = 0.5$$

NITA can be allowed at this point

T

t

- **ITA (and not NITA) will need to be used in this approach with larger Δt**

T – Flow-through time i.e. time that is required for the flow to go through the computational domain from the inlet to the outlet

... and to finalise:

- **More information to be included in the next versions:**
  - How to use the FFT tool in ANSYS Fluent.
  - Postprocessing the spectra and using the "Confidence Intervals" method.
  - The best way to create animations for the Scale-Resolving Simulations.
  - Creating journal files for LES.
  - LES & Multiphase: How to make it work.

- **Acknowledgement**
  - Special thanks go to my fellow colleagues from ANSYS technical services teams Domenico Caridi, Marco Oswald & Ulrich Postl for their comments, suggestions and for the long list of common frequently asked LES questions that are now addressed in this document.
  - I am also grateful to the members of our development organisation Davor Cokljat, Florian Menter & Richard Lechner for answering and clarifying certain theoretical and implementation-related questions so swiftly.
  - Last, but by no means least, I would like to thank my former colleagues with vast academic and industrial experience in the field of turbulence modelling: Simon Gant of Health & Safety Laboratory, UK and Fredrik Carlsson at FS Dynamics, Sweden for their detailed reviews of this material.

ANSYS