HPC and turbulent boundary layers on airplane wings

Philipp Schlatter
Armin Hosseini, Ricardo Vinuesa, Ardeshir Hanifi, Dan Henningson

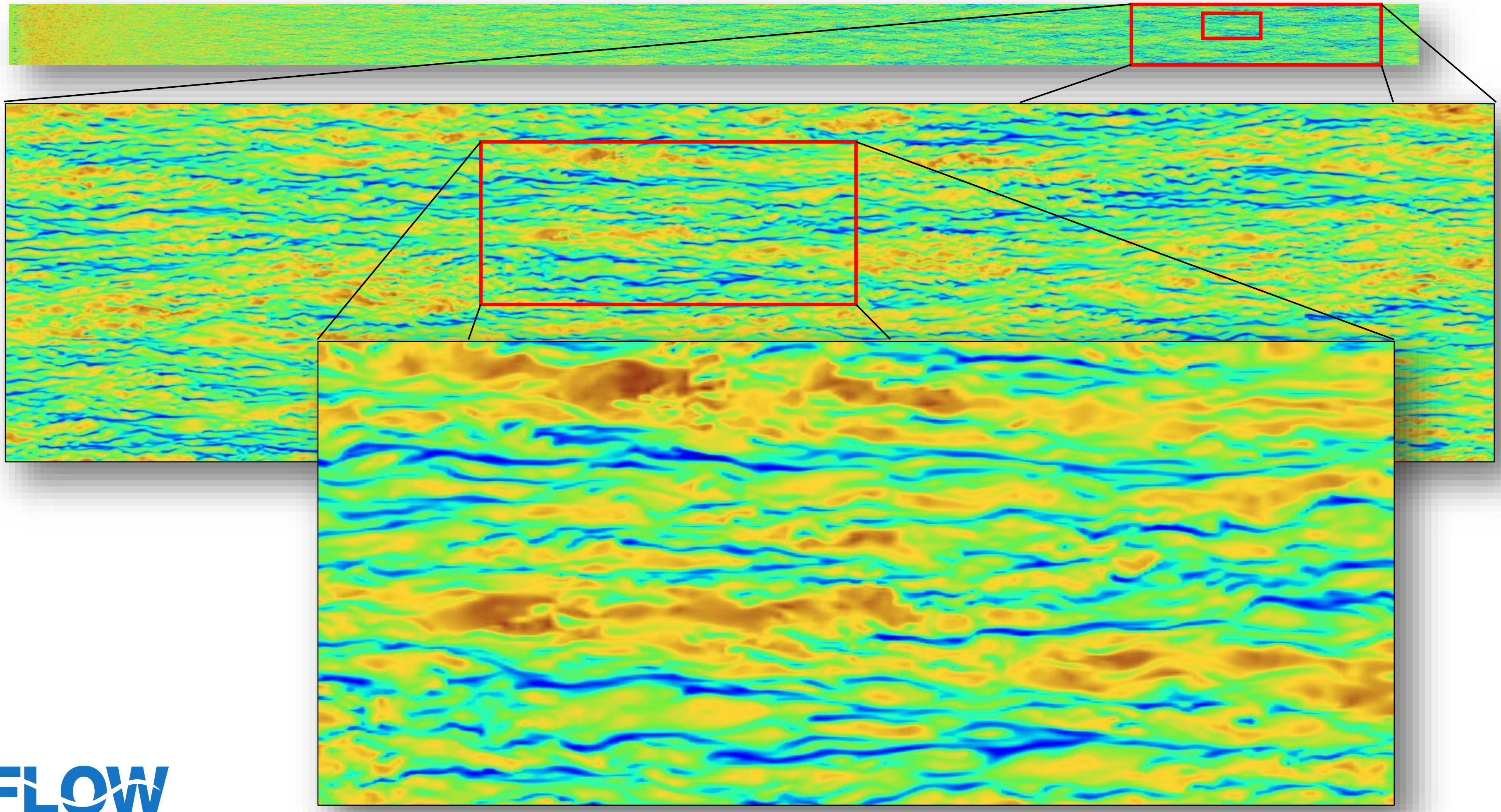
Linné FLOW Centre and
Swedish e-Science Research Centre (SeRC),
KTH Mechanics, Stockholm, Sweden

Turbulent flow close to solid walls...

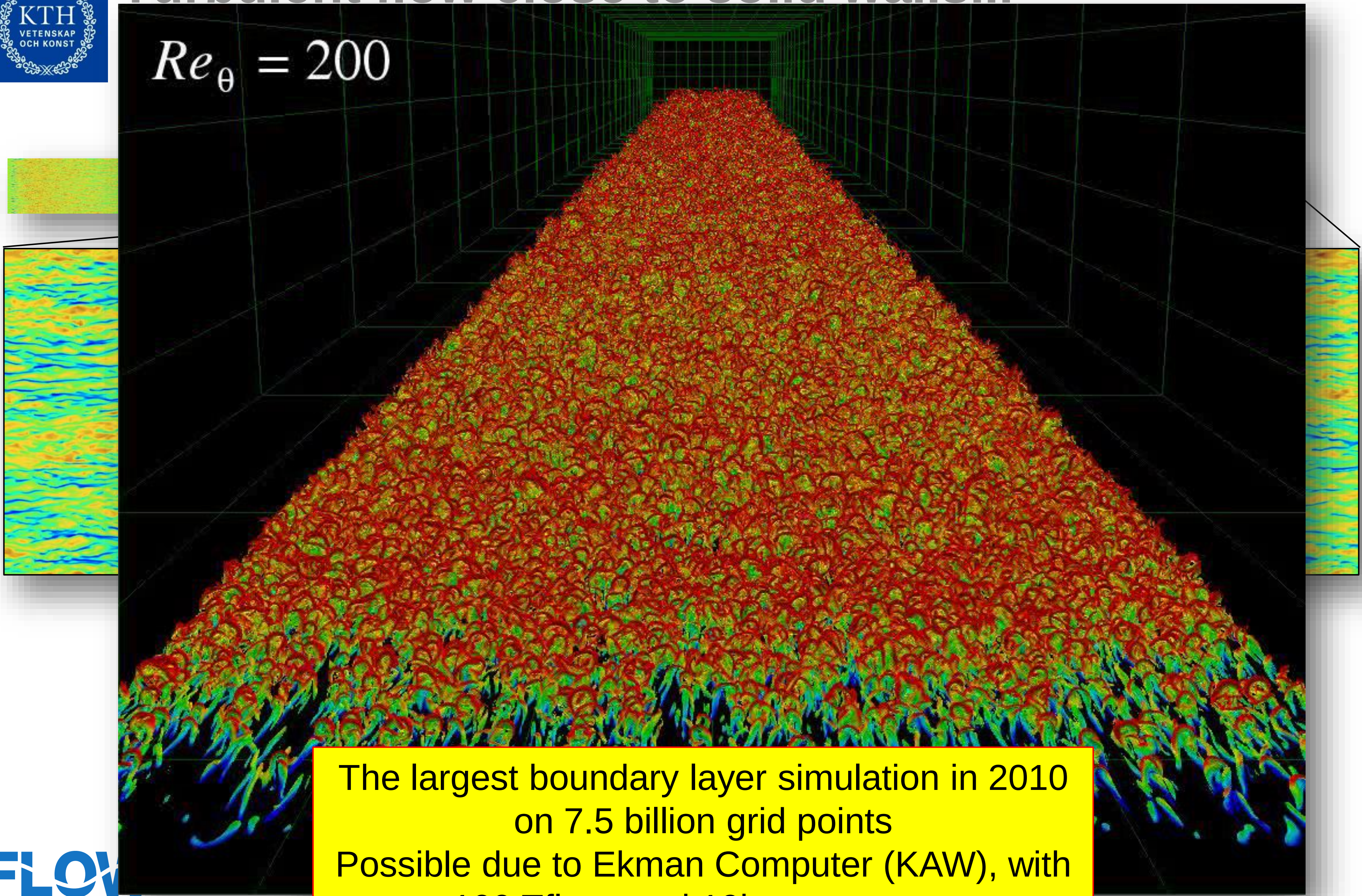
FLOW
LINNÉ FLOW CENTRE

Turbulent flow close to solid walls...

simulation result



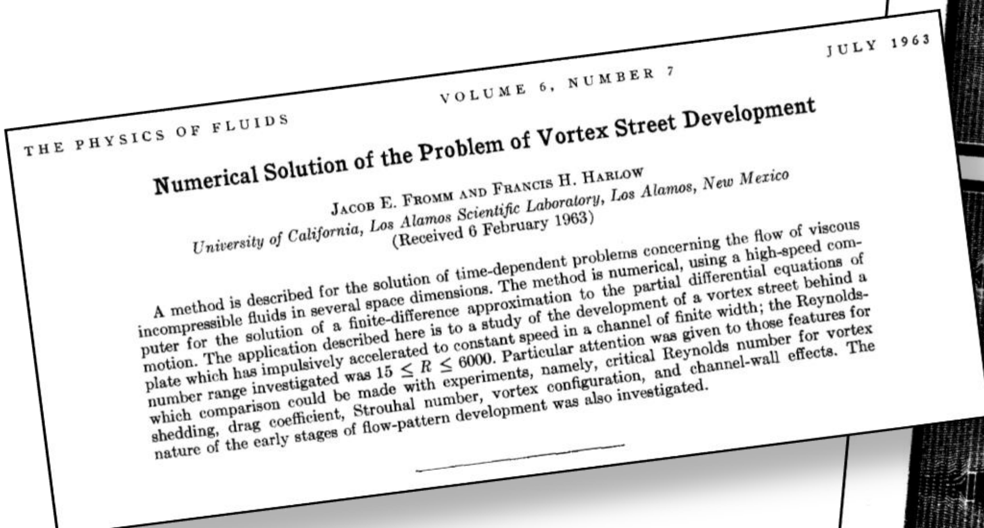
Turbulent flow close to solid walls...



The largest boundary layer simulation in 2010 on 7.5 billion grid points
Possible due to Ekman Computer (KAW), with 100 Tflops and 10k processors.

Brief History (1/4): - 1960's

- ”First simulations” (NWP) by Lewis Fry Richardson 1920: Eight hours weather prediction in 6 weeks, using **2000 ”human” computers**
- Low-*Re* cylinder wakes by Thom (1933), Kawaguti (1953) and Fromm & Harlow (1963), Los Alomos



THE PHYSICS OF FLUIDS VOLUME 6, NUMBER 7 JULY 1963

Numerical Solution of the Problem of Vortex Street Development

Jacob E. Fromm and Francis H. Harlow

University of California, Los Alamos Scientific Laboratory, Los Alamos, New Mexico

(Received 6 February 1963)

A method is described for the solution of time-dependent problems concerning the flow of viscous incompressible fluids in several space dimensions. The method is numerical, using a high-speed computer for the solution of a finite-difference approximation to the partial differential equations of motion. The application described here is to a study of the development of a vortex street behind a plate which has impulsively accelerated to constant speed in a channel of finite width; the Reynolds-number range investigated was $15 \leq R \leq 6000$. Particular attention was given to those features for which comparison could be made with experiments, namely, critical Reynolds number for vortex shedding, drag coefficient, Strouhal number, vortex configuration, and channel-wall effects. The nature of the early stages of flow-pattern development was also investigated.

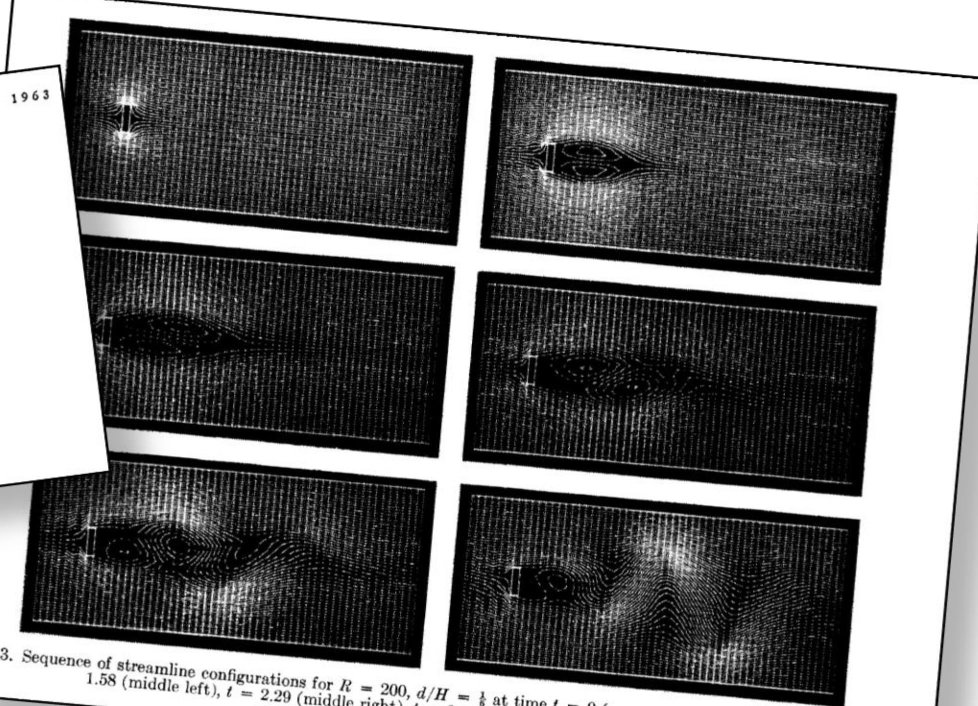


Fig. 3. Sequence of streamline configurations for $R = 200$, $d/H = \frac{1}{6}$ at time $t = 0$ (upper left), $t = 0.70$ (upper right), $t = 1.58$ (middle left), $t = 2.29$ (middle right), $t = 2.64$ (lower left), and $t = 3.16$ (lower right).

1440 grid points!

Brief History (2/4): 1960's

- 1965: **MAC** (Marker&Cell) method (Harlow&Welch): staggered grid
- 1966: Journal of Computational Physics founded
- 1968/1969: **Numerical methods for NS with pressure projection**: Chorin and Temam.

MATHEMATICS OF COMPUTATION
October, 1968, Vol. 22, No. 104
Pp. 745–762

Numerical Solution of the Navier-Stokes Equations*

By Alexandre Joel Chorin

Abstract. A finite-difference method for solving the time-dependent Navier-Stokes equations for an incompressible fluid is introduced. This method uses the primitive variables, i.e. the velocities and the pressure, and is equally applicable to problems in two and three space dimensions. Test problems are solved, and an application to a three-dimensional convection problem is presented.

Sur l'Approximation de la Solution des Équations de Navier-Stokes par la Méthode des Pas Fractionnaires (II)

R. TÉMAM

Mémoire présenté par J. L. LIONS

Introduction

Comme dans deux travaux précédents [8, 9], nous nous intéressons ici à l'approximation du problème de Navier-Stokes suivant: étant donné un ouvert borné Ω de R^2 et un nombre $T>0$, trouver les fonctions $u=\{u_1, u_2\}$ et p définies dans $\Omega\times[0,T]$ et qui vérifient les équations

$$\frac{\partial u}{\partial t}-\nu\Delta u+\sum_{i=1}^{2} u_i\frac{\partial u}{\partial x_i}+\operatorname{grad} p=f \quad (f \text{ donné}) \tag{0.1}$$
$$\operatorname{div} u=0$$

et les conditions aux limites et initiales suivantes:

$$u(x,t)=0 \quad \text{si } x\in\Gamma \text{ (la frontière de } \Omega) \tag{0.2}$$
$$u(x,0)=u_0(x) \quad (u_0 \text{ donné}).$$

La méthode que nous envisageons ici diffère des deux méthodes respectivement considérées en [8] et [9] en ce sens qu'elle ne repose pas sur une méthode de perturbation; par contre, comme en [9] la méthode considérée ici est encore une méthode discrète de pas fractionnaires [4, 6, 7].

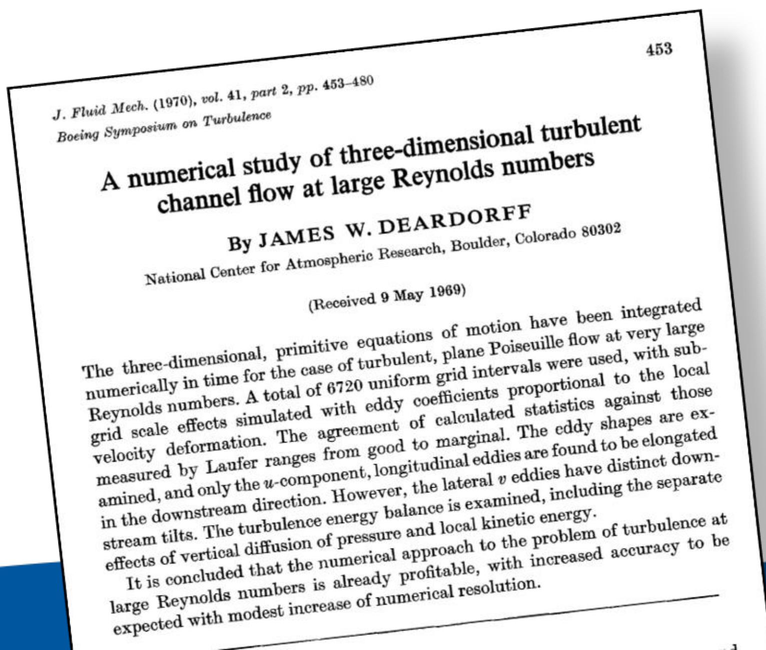
Rappelons rapidement en quoi consiste la méthode des pas fractionnaires lorsqu'il s'agit d'approcher une équation d'évolution

$$\frac{\partial u}{\partial t}+Au=f \tag{0.3}$$

LINNÉ FLOW CENTRE

Brief History (3/4): 1970's

- 1970: first channel-flow **large-eddy simulation**: Deardorff (6720 grid points), based on Smagorinsky model (1963)
- 1972: k-ε **turbulence model** (RANS): Spalding & Launder
- 1972: SIMPLE (semi-implicit method for pressure-linked equations): Patankar & Spalding
- 1973: The abbrevation **CFD** (**Computational Fluid Dynamics**, not "Colours for Directors"...) is coined



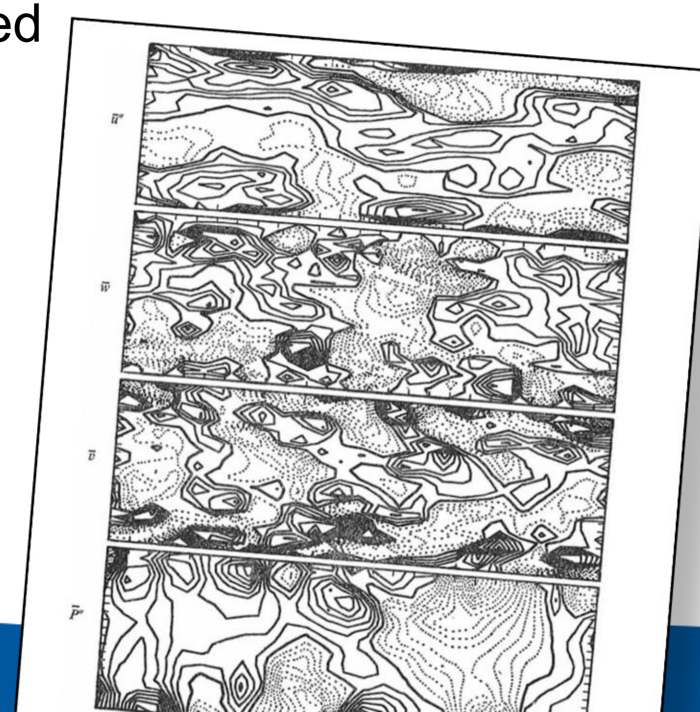
J. Fluid Mech. (1970), *vol.* 41, *part* 2, *pp.* 453–480
Boeing Symposium on Turbulence

A numerical study of three-dimensional turbulent channel flow at large Reynolds numbers

By JAMES W. DEARDORFF
National Center for Atmospheric Research, Boulder, Colorado 80302

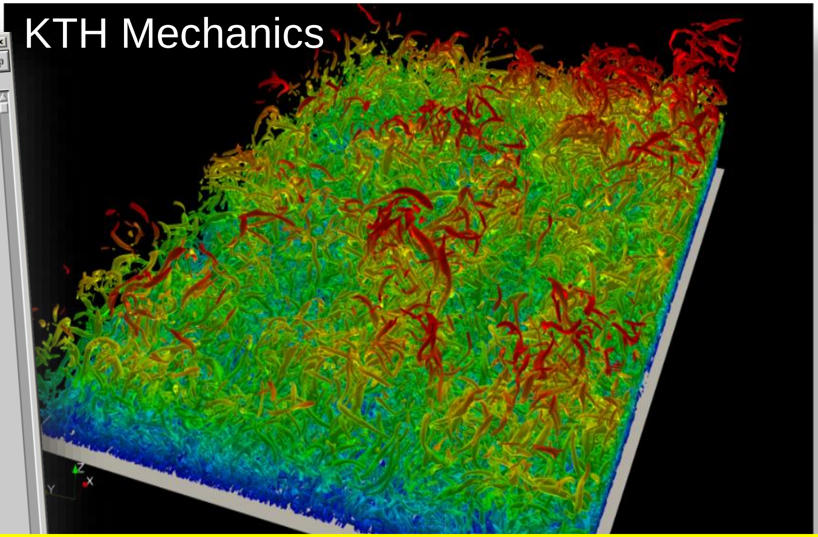
(Received 9 May 1969)

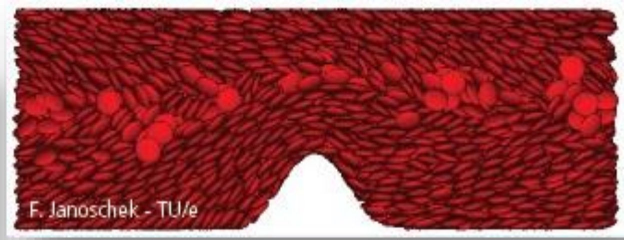
The three-dimensional, primitive equations of motion have been integrated numerically in time for the case of turbulent, plane Poiseuille flow at very large Reynolds numbers. A total of 6720 uniform grid intervals were used, with sub-grid scale effects simulated with eddy coefficients proportional to the local velocity deformation. The agreement of calculated statistics against those measured by Laufer ranges from good to marginal. The eddy shapes are examined, and only the *u*-component, longitudinal eddies are found to be elongated in the downstream direction. However, the lateral *v* eddies have distinct downstream tilts. The turbulence energy balance is examined, including the separate effects of vertical diffusion of pressure and local kinetic energy.
It is concluded that the numerical approach to the problem of turbulence at large Reynolds numbers is already profitable, with increased accuracy to be expected with modest increase of numerical resolution.



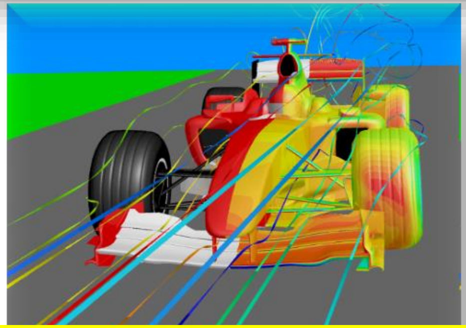
Brief History (4/4): 1980's -

- 1980 - : CFD codes used in engineering (e.g. Fluent, ANSYS, *etc.*); first for aircrafts, then also automotive *etc.*
- 1987: **First fully resolved DNS of channel flow ($4\cdot10^6$ grid points):** Kim, Moin & Moser





TU Eindhoven



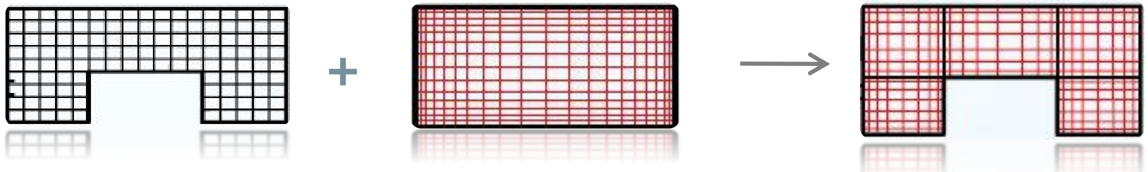
Fluent

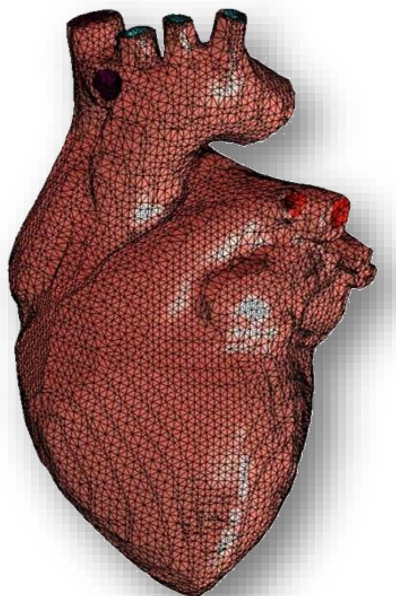
today:
- Computational fluid dynamics is integral part of both engineering and research, calculations up to **$50\cdot10^9$ grid points** and **1'000'000 cores** "easily" possible
- Data post processing! Storage! Visualisation!

LINNÉ FLOW CENTRE

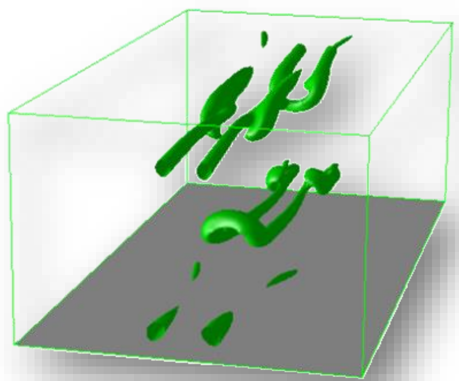
Spectral Element Method (Patera 1984)

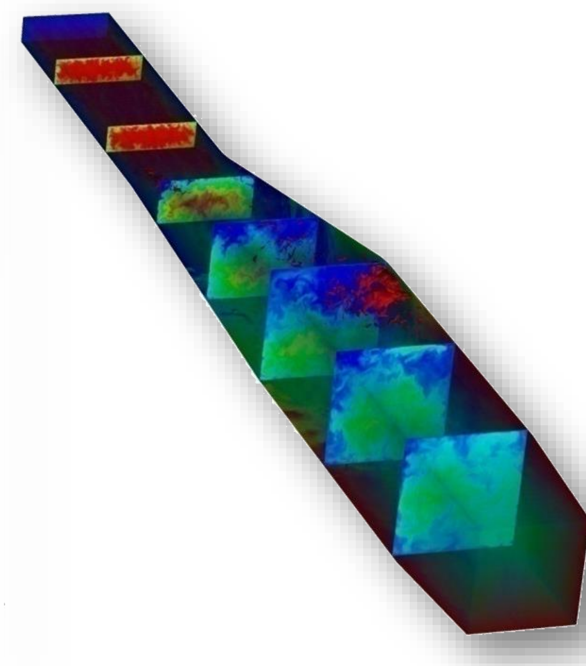
Finite element (FE) + Spectral → Spectral element (SE)





(Zhang et al. 2004)



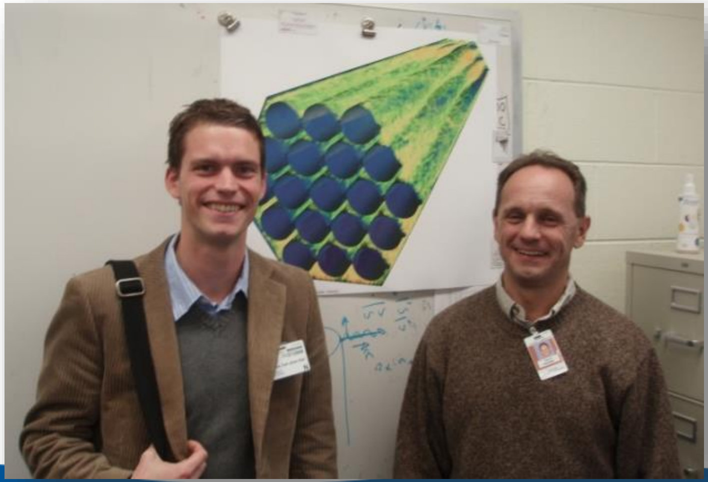


Nek5000 – Spectral Elements

- SEM code by **Paul F. Fischer**, Argonne National Lab, USA
  Open source: `nek5000.mcs.anl.gov`
- 80 000 lines of **Fortran 77** (some C for I/O), MPI (no hybrid)
- **Gordon Bell Prize 1999** for algorithmic quality and performance
- ”Keep it simple” – world’s most powerful computers have very weak operating systems

- Much effort on coarse-grid solvers (AMG and XX^T), OpenACC, adaptive meshes, new MPI *etc.*
- CRESTA Project, SeRC Application Experts

- Good scaling up to **1,000,000** ranks on Mira (10PFlops BG/Q)



FLOW
LINNÉ FLOW CENTRE

Beskow – the new Cray XC-40

- Pilot access from mid December 2014…

- …we prepared a big case (3.4 million elements, about 5 billion grid points)
- But then… The code stops immediately with:

`ABORT: MPI_TAG_UB too small!`

MPI_TAG_UB

- Inter-node communication via pure MPI
- Messages identified by destination process and tag=element number
- Upper limit for tags is MPI_TAG_UB
  - MPI Standard – 16 bits – up to **32,768 values**
  - Most MPI libraries – 32 bits – up to **2,147,483,647 values**
- Justifies use of global element number in 1999 (number of elements was around 10000)
- Cray XC-40 – **22 bits – up to 2,097,152 values**, but now we have 3,400,000 elements!

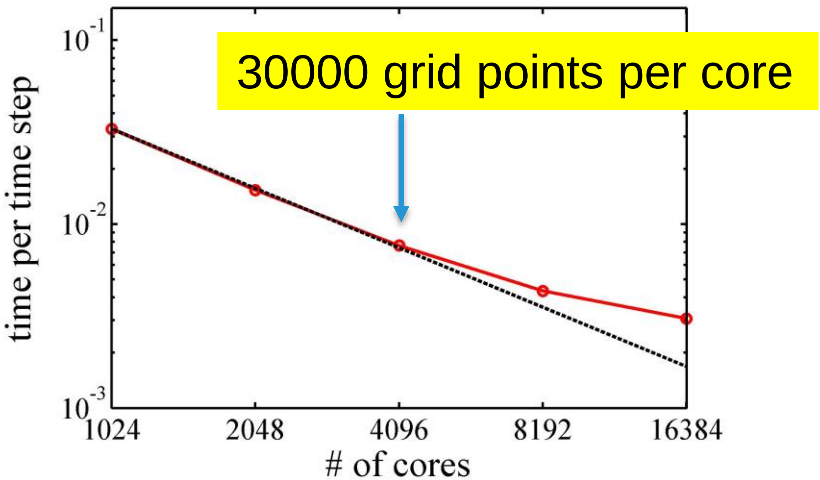
MPI_TAG_UB

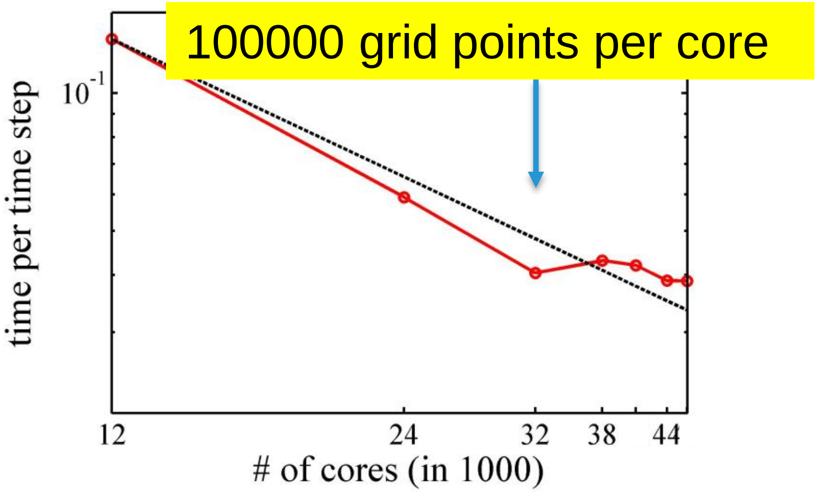
- Simple fix for Beskow, but **deeeeeep** in the code…
- Unique global element number **`eg`** has to be replaced with the unique pair:
  - destination process number **`mid`**
  - local element number at the destination process **`e`**

```
      do eg=1,nelgt                 ! sync NOT needed here

         mid = gllnid(eg)
         e   = gllel (eg)
!     tag for sending and receiving changed from global (eg) to local (e) element number
!     to avoind problems with MPI_TAG_UB on CRAY
#ifdef DEBUG
         if (nio.eq.0.and.mod(eg,niop).eq.0) write(6,*) eg,' mesh read'
#endif
         if (mid.ne.nid.and.ni[illegible].eq.0) then              ! read & send

            if(ierr.eq.0) [illegible]
              call byte_re[illegible] (buf,nwds,[illegible])
              call csend(e,ierr,len1,mid[illegible]
              if(ierr.eq.0) call csend(e,buf,len,mid,0)
            else
              call csend(e,ierr,len1,mid,0)
            endif
```

Parallel Scaling on Beskow

- Pilot user phase (December 2014)



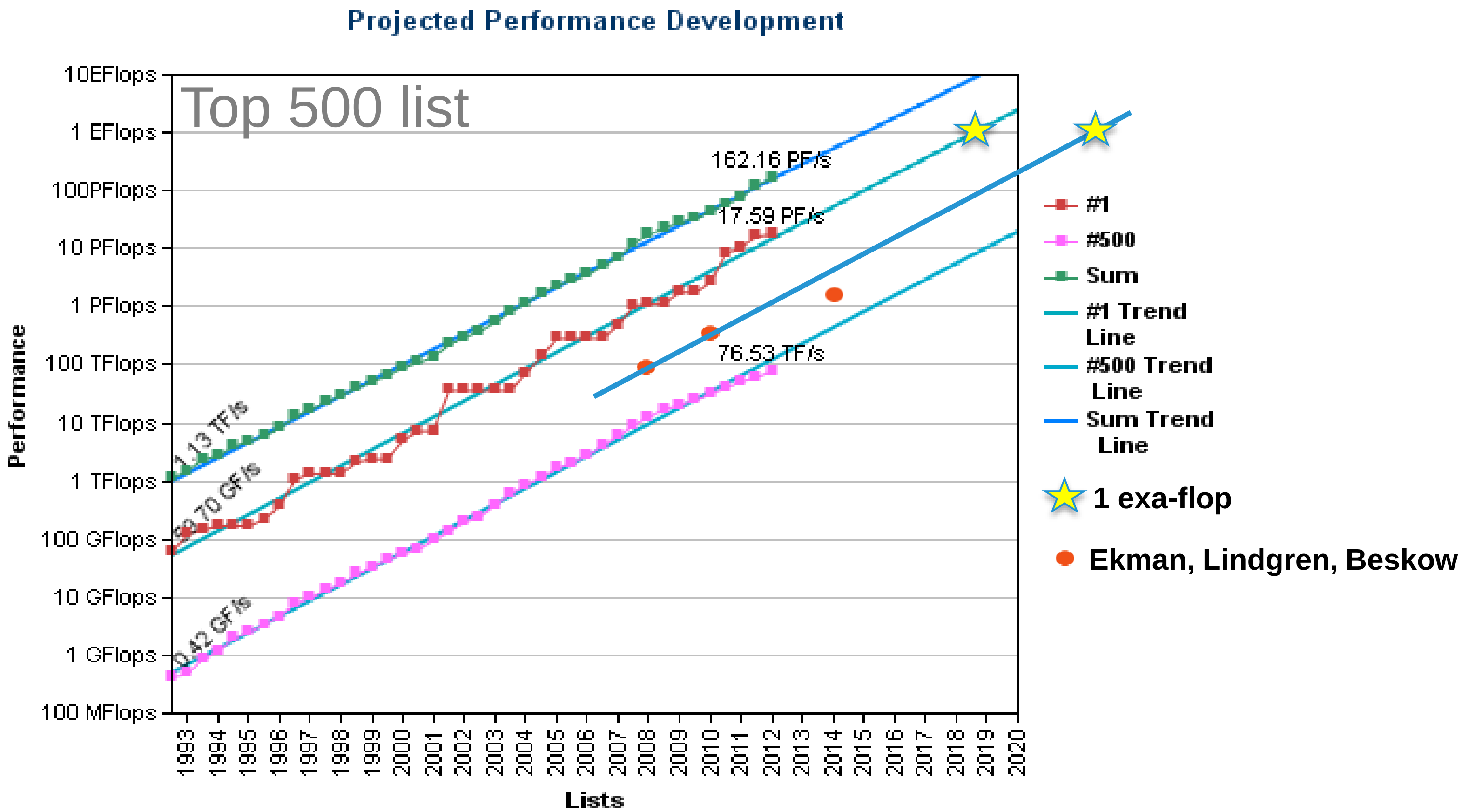


Strong scaling: small case (120 million GP)   large case (2.3 billion GP)

- Comparison to Lindgren: 3-4x faster per core
- Comparison to Triolith: 1.4x faster per core

Future possibilities with increasing computer speed, exa-flop 2018/19?

Slope faster than hardware development!



“Numerical wind tunnel”

EU-project RECEPT, KTH Mechanics

laminar Flow Control Experiment:
$Re = 1*15/1.5*10^{-5} = 1x10^{6}$

turbulent boundary layer:
$Re= 5*30/1.5*10^{-5} = 10x10^{6}$

- DNS of typical wind tunnel experiment
  - ~ 10 billion grid points
  - ~ 100 million core hours
  - ~ 1 peta byte of data
  - ~ **100 days on 32000 cores**

    (peta-scale sufficient, Beskow)

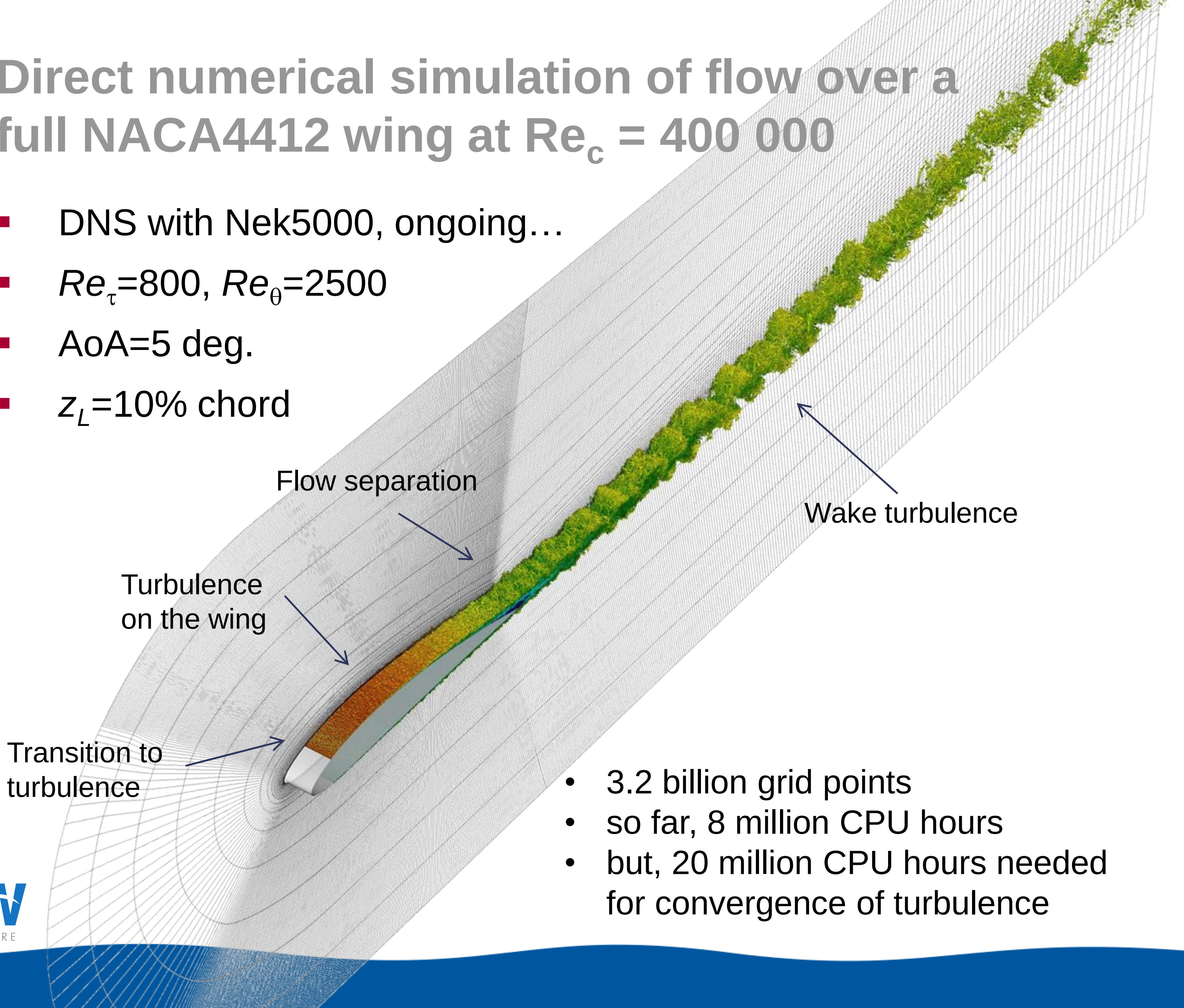
- DNS of Saab 2000 wing section
  - ~ 1000 times larger computation

    (exa-scale needed, >10 years)

FLOW
LINNÉ FLOW CENTRE

Direct numerical simulation of flow over a full NACA4412 wing at Re_c = 400 000

- DNS with Nek5000, ongoing…
- Re_τ=800, Re_θ=2500
- AoA=5 deg.
- z_L=10% chord

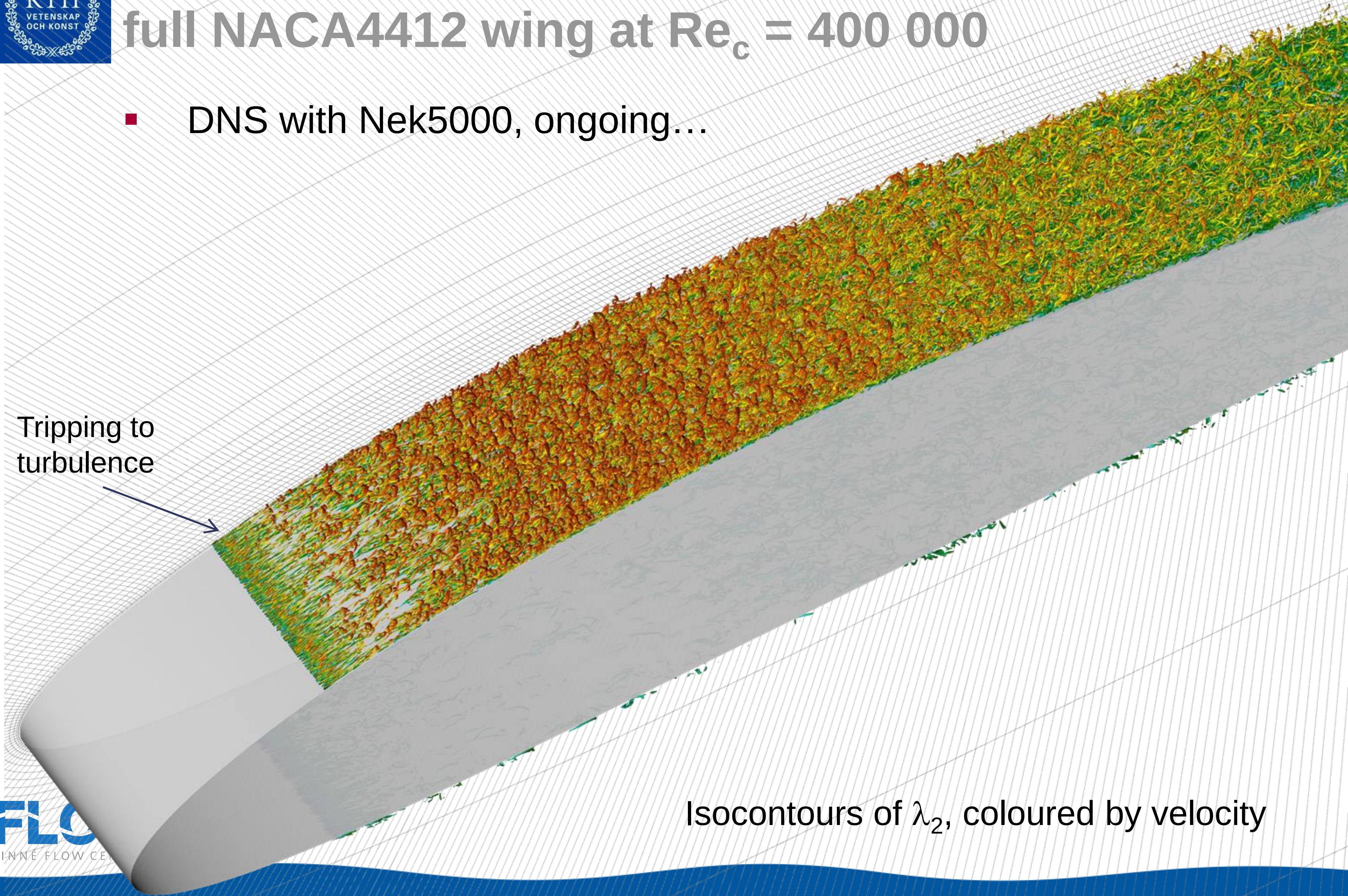


- 3.2 billion grid points
- so far, 8 million CPU hours
- but, 20 million CPU hours needed for convergence of turbulence

FLOW
LINNÉ FLOW CENTRE

Direct numerical simulation of flow over a full NACA4412 wing at Re_c = 400 000

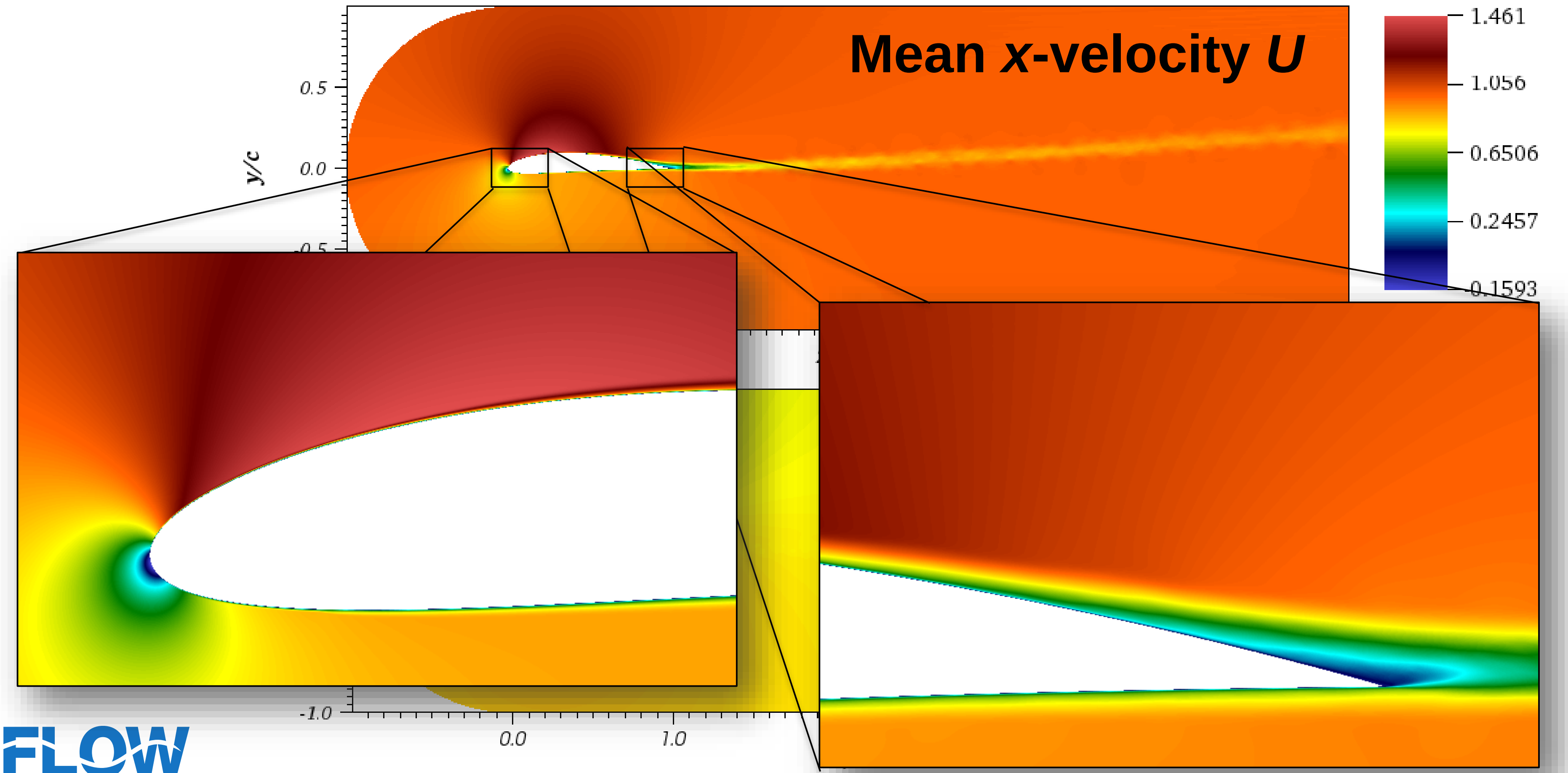
- DNS with Nek5000, ongoing…



Isocontours of λ_2, coloured by velocity

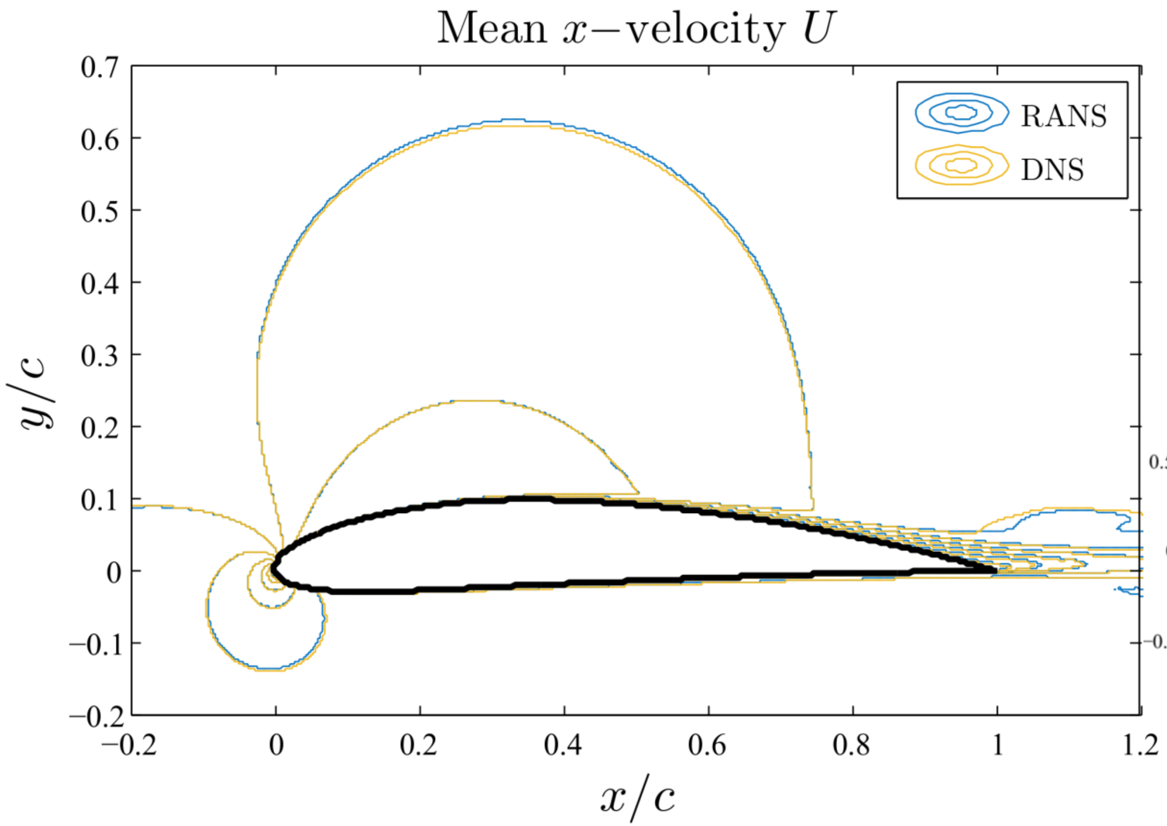
Direct numerical simulation of flow over a full NACA4412 wing at Re_c = 400 000

- Flow statistics: Averages of Turbulence

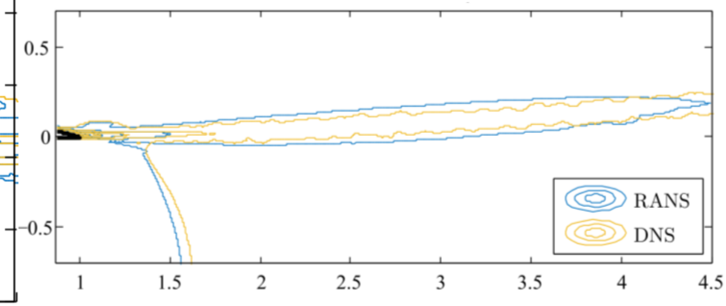


Direct numerical simulation of flow over a full NACA4412 wing at Re_c = 400 000

- Same geometry simulated with state-of-the-art RANS to design the mesh and boundary conditions
- Excellent agreement between RANS and DNS!

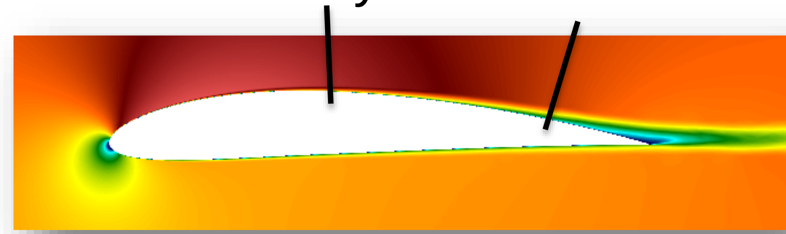


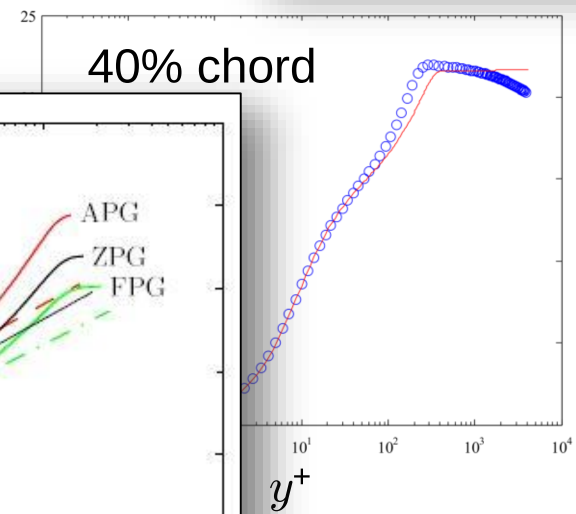
Wake region

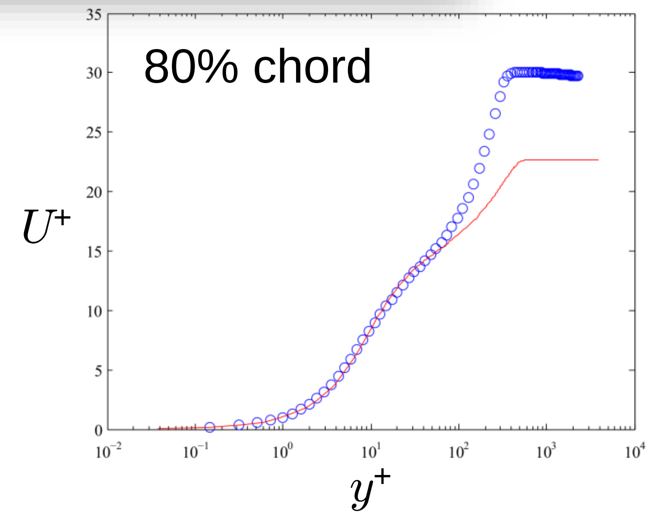


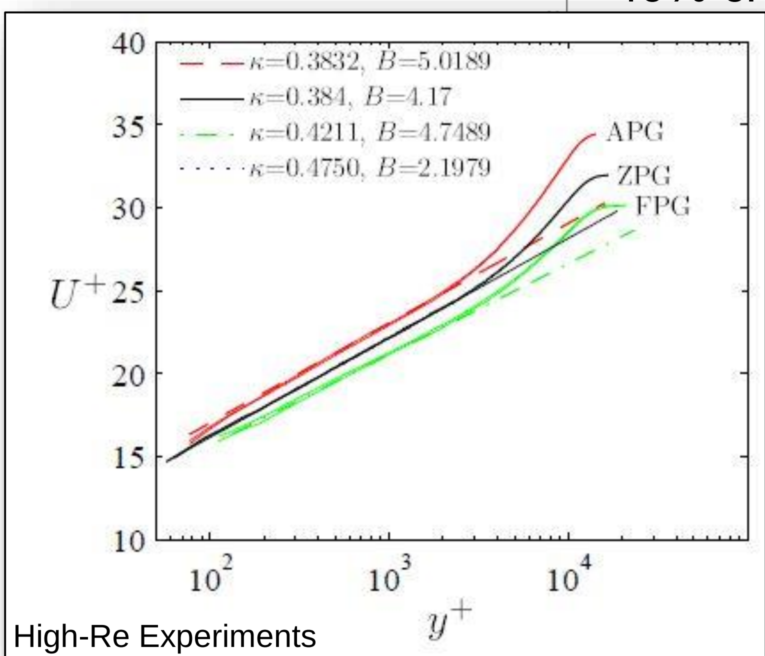
Direct numerical simulation of flow over a full NACA4412 wing at Re_c = 400 000

- Turbulence Statistics: Mean velocity





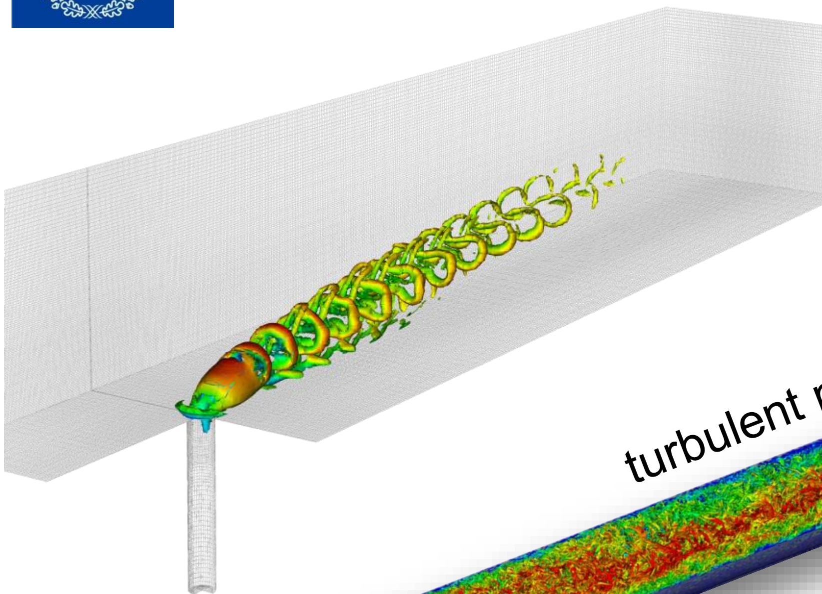




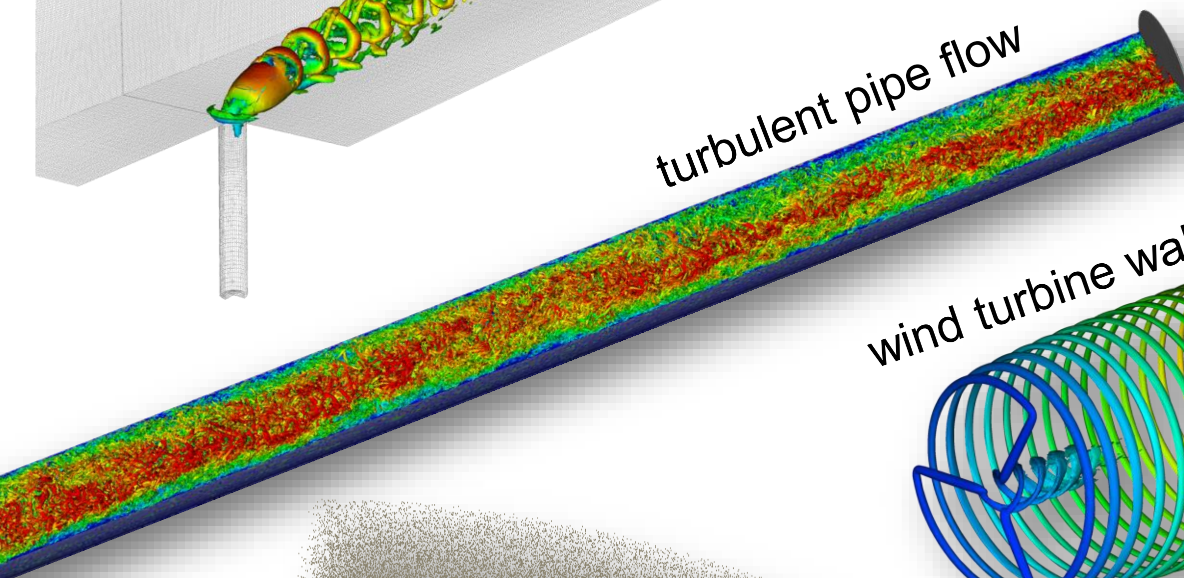
Canonical boundary layer (Schlatter 2010)

Present simulations on wings

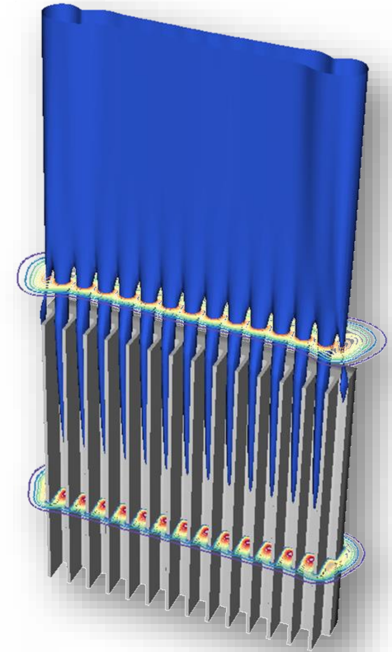
Some other Nek5000 projects



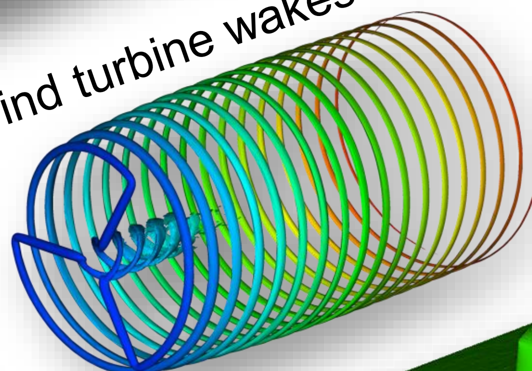
stability tools for jet in crossflow



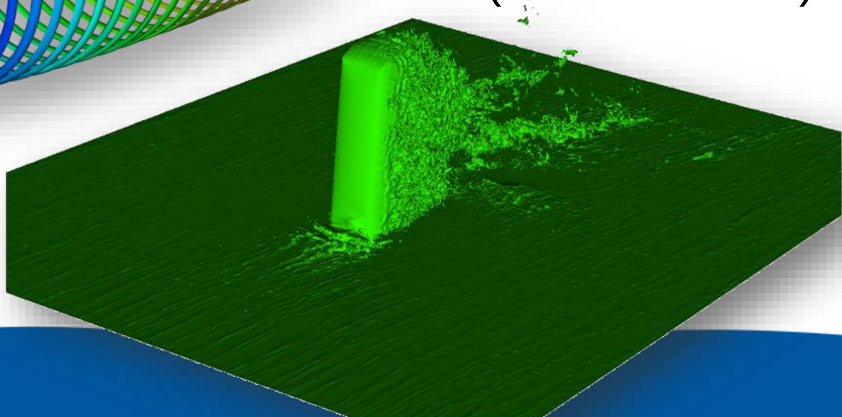
turbulent pipe flow



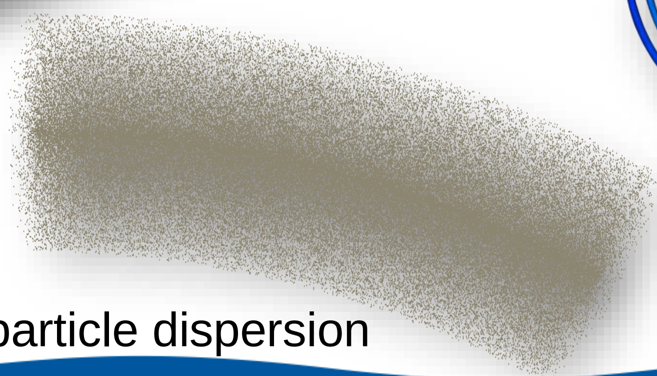
optimisation of heat sinks



wind turbine wakes



”skyscraper” (with U Ottawa)



particle dispersion

Acknowdledgments:

KTH VETENSKAP OCH KONST

VisIt

ParaView

KTH VETENSKAP OCH KONST
ROYAL INSTITUTE OF TECHNOLOGY

SeRC
Swedish e-Science Research Centre

FLOW
LINNÉ FLOW CENTRE

PDC HPC

NSC
National Supercomputer Centre at Linköpings universitet

PRACE

Knut och Alice Wallenbergs Stiftelse

Vetenskapsrådet

SNIC
Swedish National Infrastructure for Computing

Thank You!