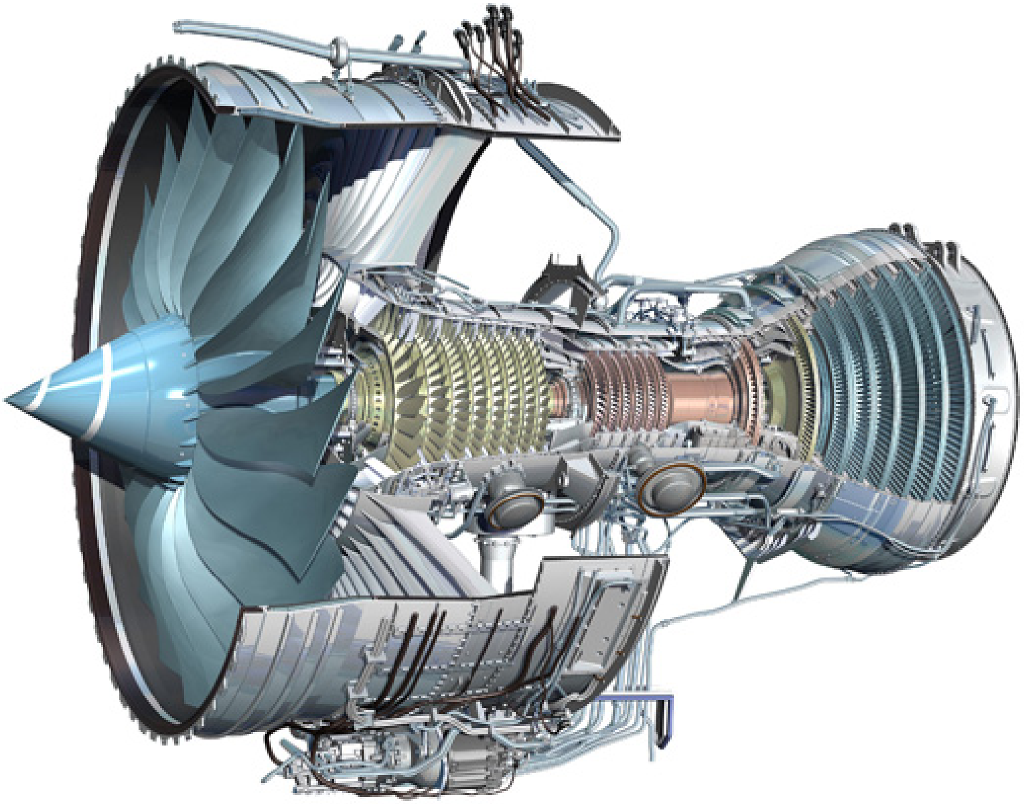
THE EVOLUTION OF TURBOMACHINERY DESIGN (METHODS)

Parsons 1895



Rolls-Royce 2008

Parsons 1895

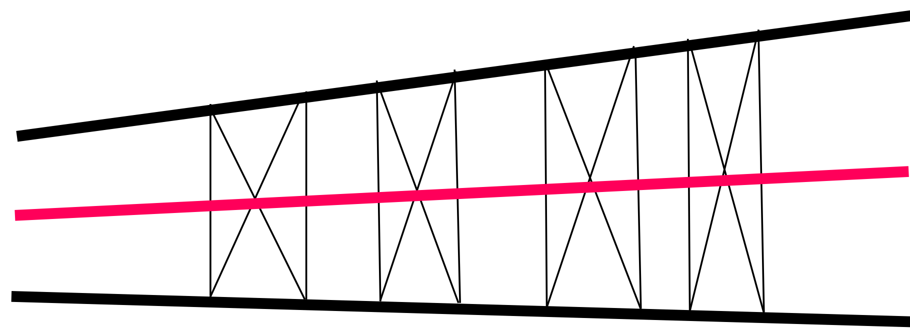
100KW Steam turbine

- Pitch/chord a bit too low.
- Tip thinning on suction side.
- Trailing edge FAR too thick.
- Surface roughness poor.

1900 - 1940. Mainly steam turbines.

Designs based on mean line velocity triangles with some cascade testing.

Free vortex design introduced in late 1920's but not generally accepted until Whittle in late 1930's.



Mean Line.

Mainly untwisted blading

1940-1950 Intensive development of the jet engine.

Much of the basic science came from NGTE, Pyestock.

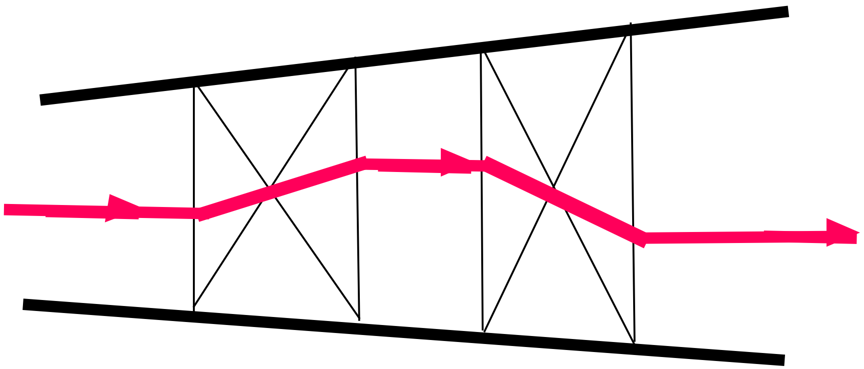
Cascade testing leads to correlations as the basis of design.

- Howell
- Carter
- Ainley & Mathieson

Some of these are still in use today.

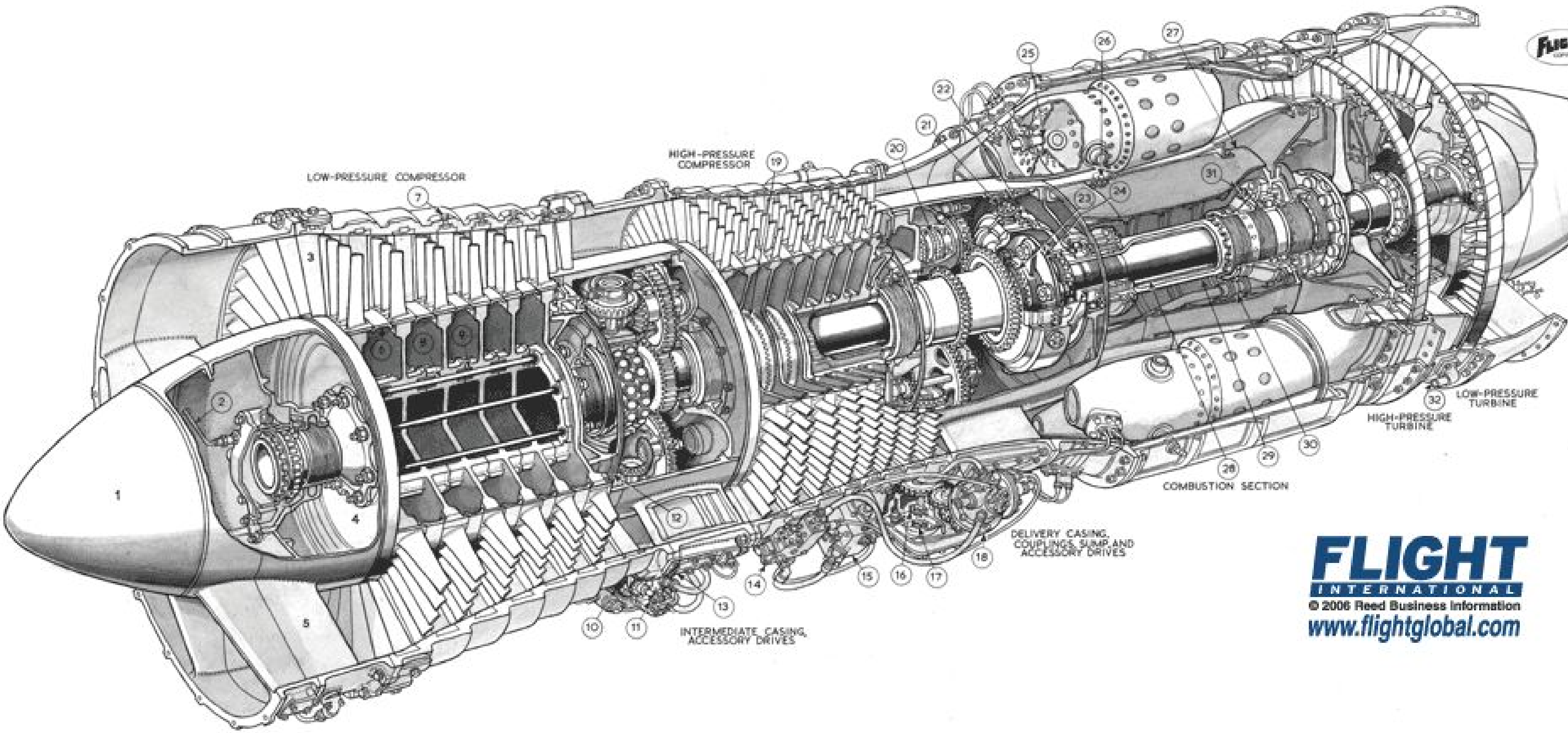
1950-1960. Radial Equilibrium used to predict the spanwise variation in velocity, etc . Assumes all the streamline shift occurs within the blade rows.

$$dP/dr = \rho V_\theta^2/r. \quad \text{-> Twisted blading.}$$

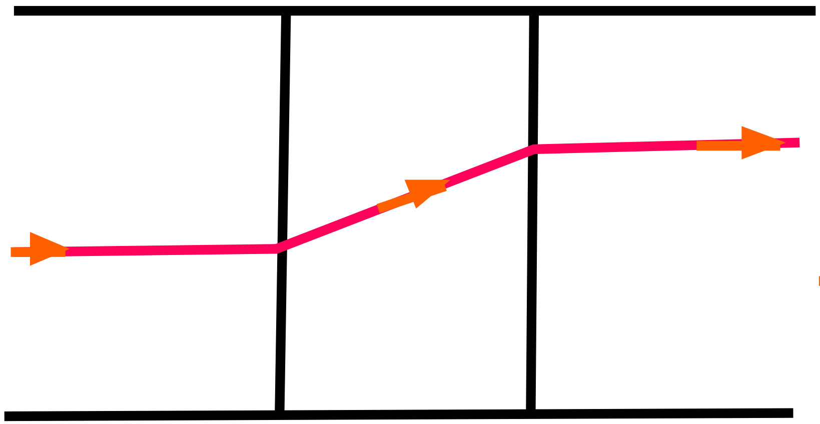


Using standard blade sections, C4, DCA, T6, etc.

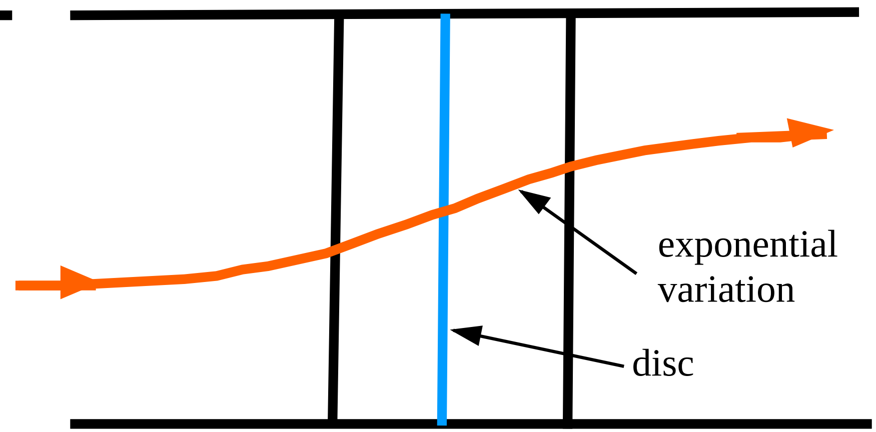
The Avon and Olympus engines were almost certainly designed in this way



Olympus Engine



Radial
Equilibrium



Actuator disc

Simple but neglects effects of streamline curvature

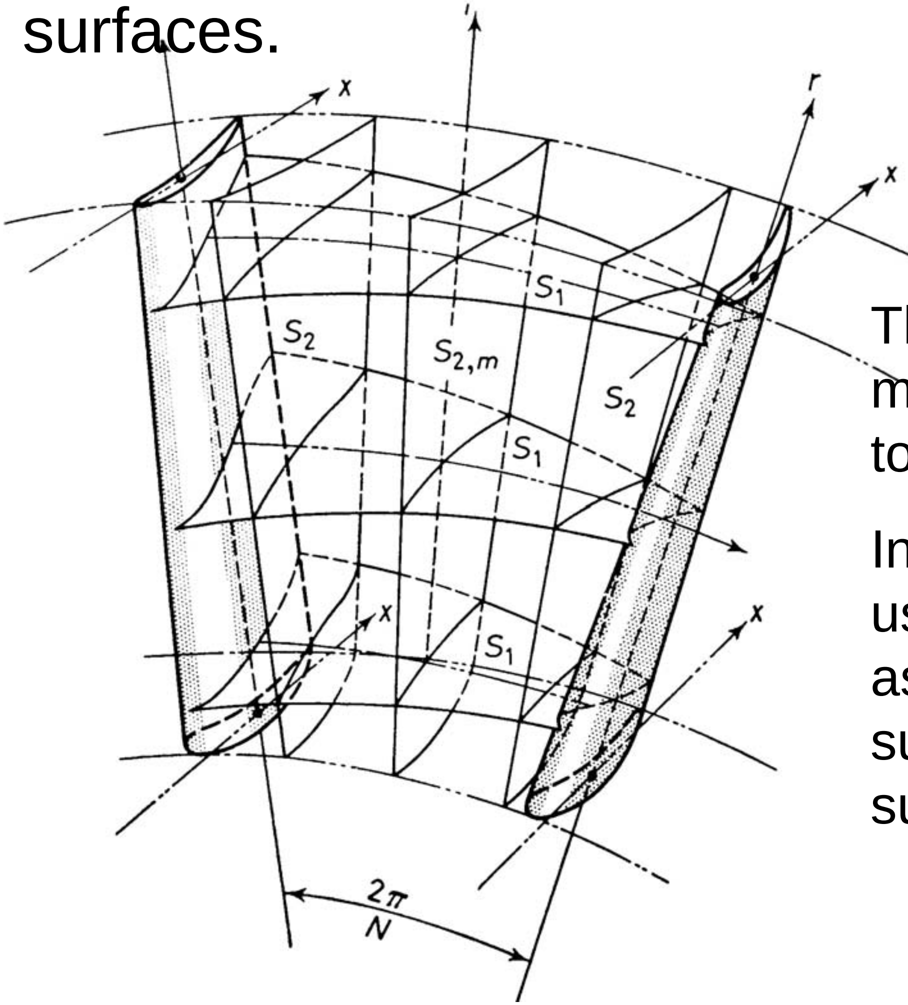
1960's

Mathematical theory

Involves Bessel functions

Where to place the disc ?

Early 1950's - Wu published his theory for predicting 3D flow by iterating between solutions on S2 (hub to tip) and S1 (blade to blade) stream surfaces.



This was far ahead of its time as no methods (or computers) were available to solve the resulting equations.

In fact the method has seldom been used in its full complexity. We usually assume a single axisymmetric S2 surface and several untwisted S1 surfaces.

The S2 (hub to tip or throughflow) solution has become the “backbone” of turbomachinery design.

Initially there was rivalry between the matrix-stream function method and the streamline curvature method of solving the equations.

Stream Function method

$$\left.\begin{aligned}\rho\, r\, Vx &= \frac{d\psi}{dr}\\ \rho\, r\, Vr &= -\frac{d\psi}{dx}\end{aligned}\right. \longrightarrow \nabla^2\psi = Fcn\left(\frac{dh}{dr}, T\frac{ds}{dr}, \frac{d(rV_\theta)}{dr}, \frac{d\psi}{dr}, \frac{d\psi}{dx}, etc\right)$$

Streamline curvature method

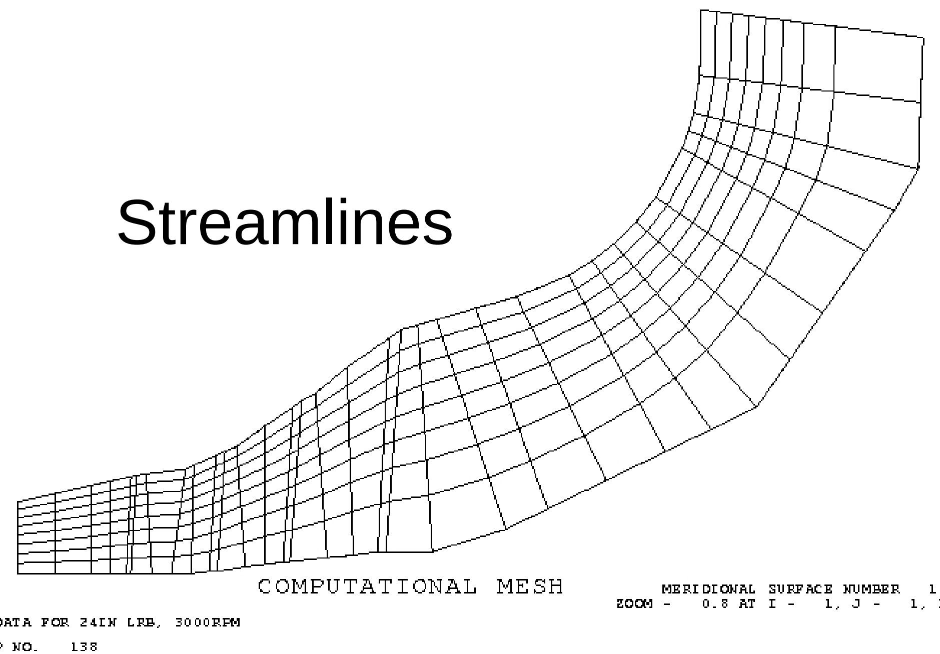
$$Vm\frac{dVm}{dr} = Fcn\left(\frac{dh}{dr}, T\frac{ds}{dr}, \frac{drV_\theta}{dr}, \frac{V_m^2}{r_c}, \frac{dV_m}{dm}, etc\right)$$

The streamline curvature method has become dominant mainly through its relative simplicity and its superior ability to deal with supersonic flows.

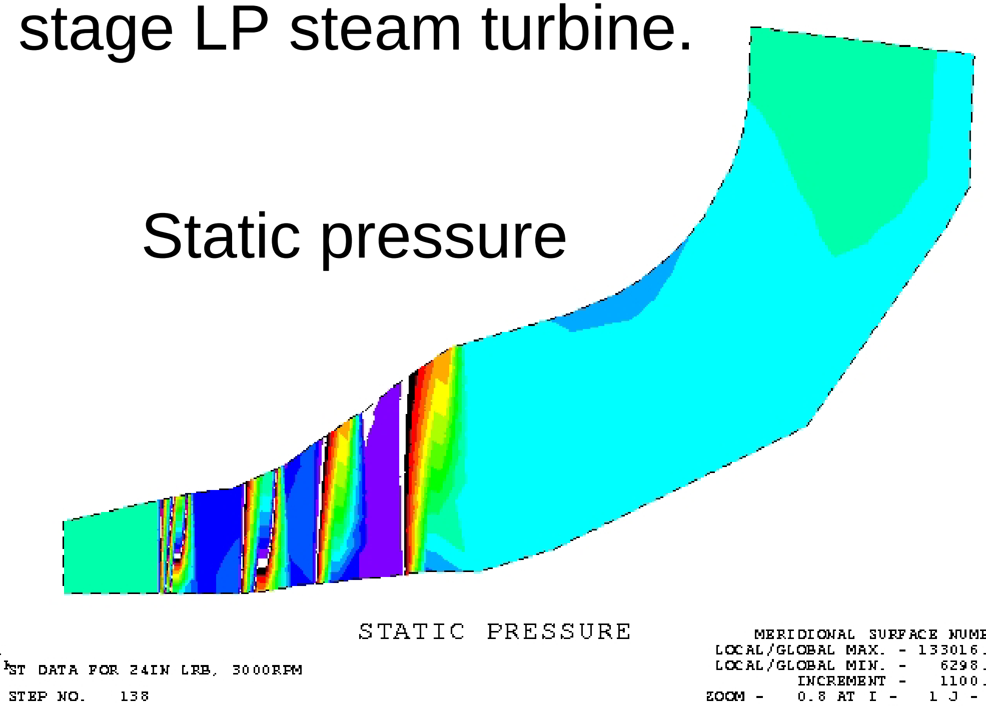
Extensions to deal with multiple choked turbines, as in LP steam turbines, were developed in the 1970’s. These brought about significant improvements in LP steam turbine performance.

4 stage LP steam turbine.

Streamlines



Static pressure



Loss and deviation correlations remain an essential part of any throughflow method.

In fact the method may be thought of as a means of applying the correlations to a non-uniform flow. The accuracy of the results is determined more by the accuracy of the correlations than by that of the numerical method.

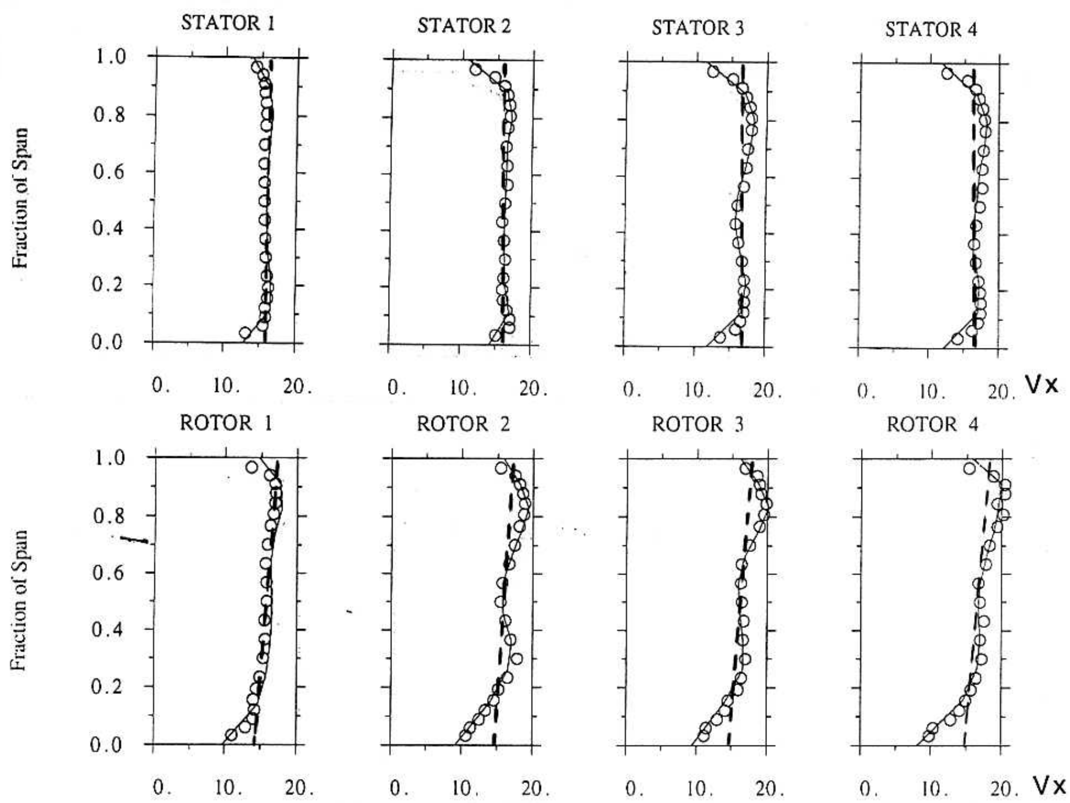


Fig 8. Measured and calculated axial velocity profiles through a 4 stage axial turbine.

Throughflow calculation for a 3 stage turbine using:

a) design

b) measured

blade exit flow angles.

In the 1970-s - 80's new correlations were developed by:

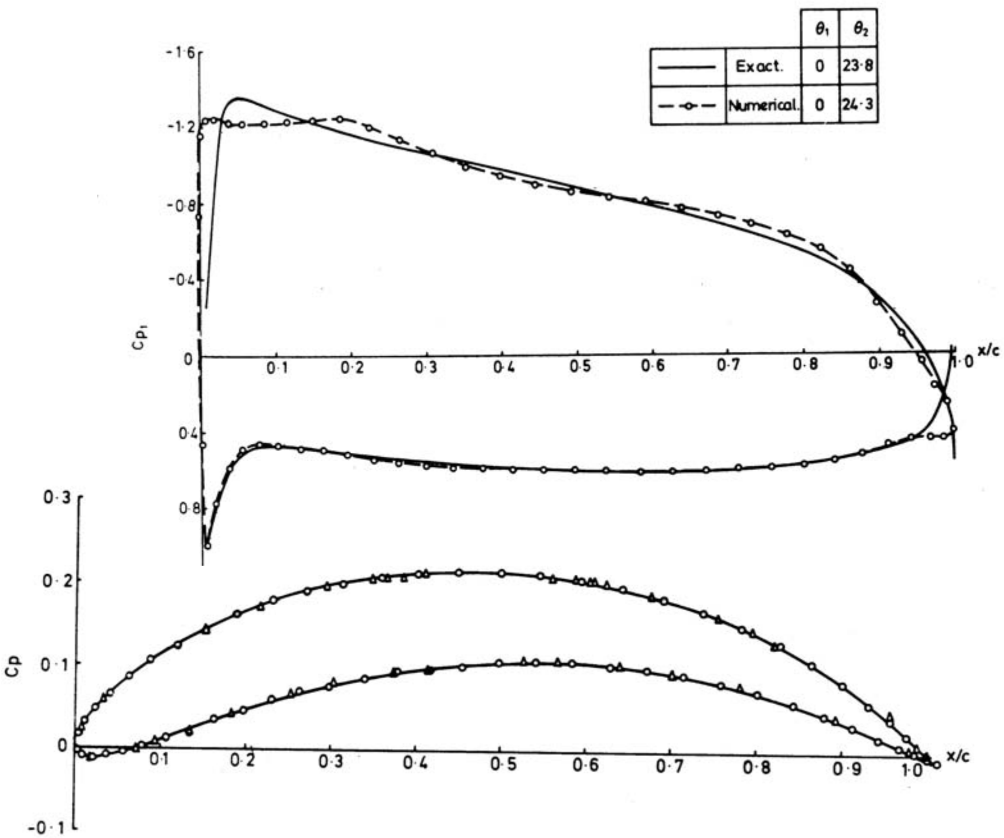
- Craig & Cox
- Dunham & Came
- Howell & Calvert

Despite these improvements correlations remain of very limited accuracy when applied to machines significantly different from those from which they were developed.

Preliminary design methods are still based on such correlations.

Blade to blade calculations on the S1 stream surface were developed in the 1960's , these were initially 2D and incompressible.

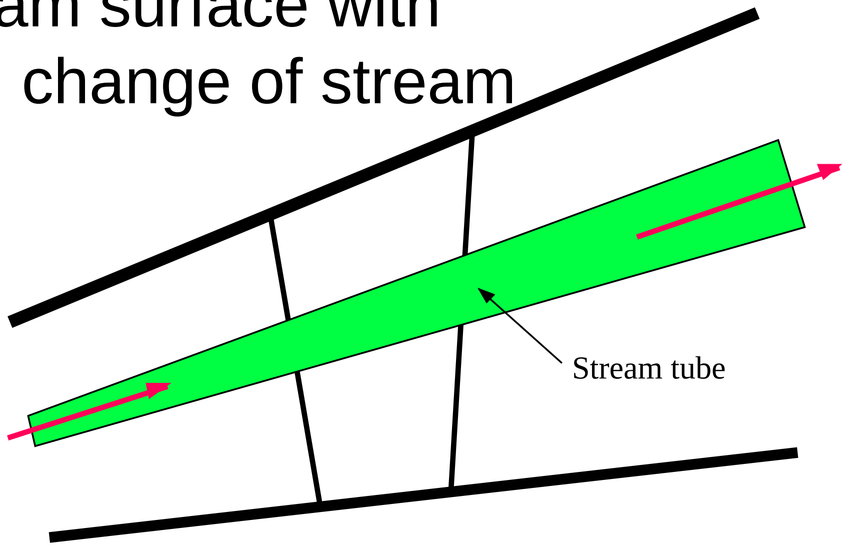
The surface singularity method (Martensen) was developed by Wilkinson and others into a very fast and accurate method. The major unknown was how to apply the Kutta condition at the trailing edge.



Despite their accuracy these methods were of limited use because the real flow is seldom either incompressible or two dimensional.

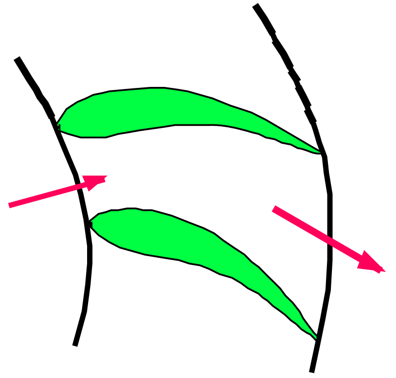
Blade to blade calculation methods for inviscid compressible flow were developed in the late 1960’s and 1970’s .

These solved for the flow on a stream surface with allowance for change in radius and change of stream surface thickness.



Methods were based on:

- Stream function
- Velocity potential
- Streamline curvature
- Time marching solution of the Euler equations.

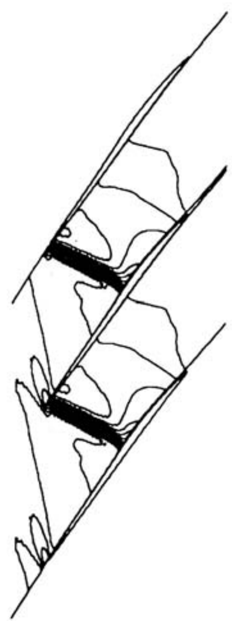


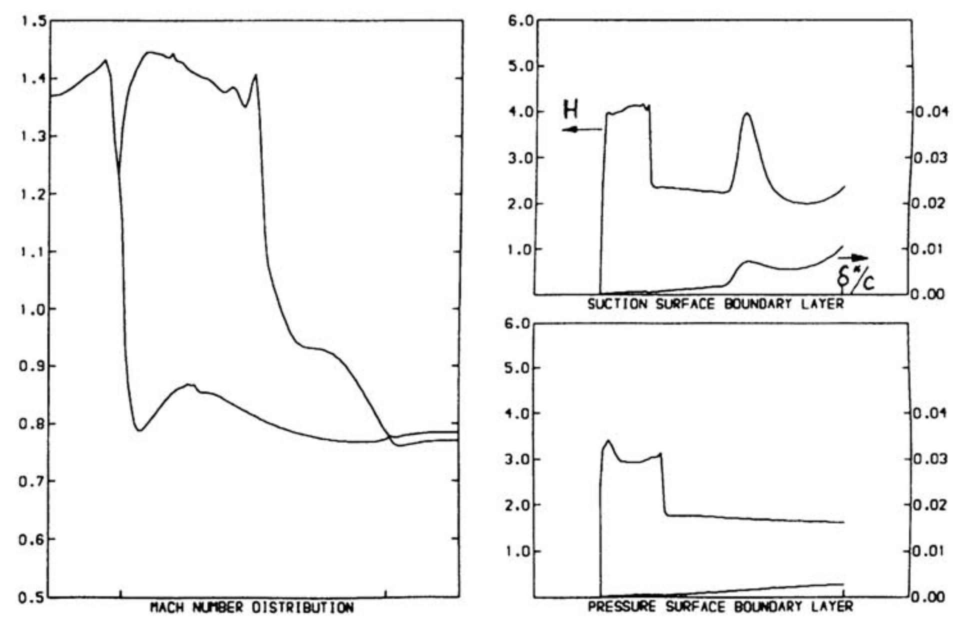
- Stream function and streamline curvature methods were fast but difficult to extend to transonic flow. They are no longer used.
- Velocity potential methods were fast and able to cope with small amounts of supersonic flow but shock waves were not well captured. They are still used.
- Time marching solutions were much slower but are able to cope with high Mach numbers and to capture shock waves. They are now the dominant method.

This type of method was used to develop “controlled diffusion” blading for axial compressors, giving significant improvements in performance.

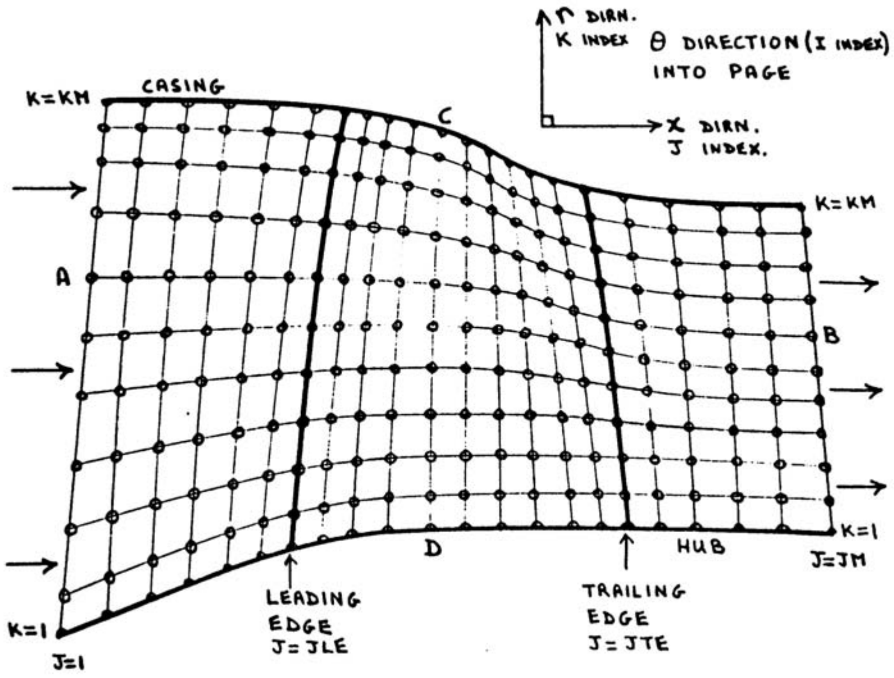
Although transonic compressors (fans) were initially developed without any flow calculation methods, the time marching methods allowed their design to be put on a much more sound footing.

A widely used method, including boundary layers, was developed by Calvert & Ginder at Pyestock.





The time marching method had the advantage of being readily extended to fully 3D flow. This was done in the mid 1970's .



A typical coarse grid for early 3D calculations.

Initially the available computers only allowed coarse grid solutions, typically 4000 (10x40x10) grid points. Although this seriously limited their accuracy the 3D methods soon lead to improved physical understanding of 3D effects such as blade sweep and blade lean.

In particular it was discovered that blade lean could have an extremely powerful effect on the flow. This had been neglected by previous methods.

When low aspect ratio blades are leaned the constant static pressure lines remain almost "frozen" .

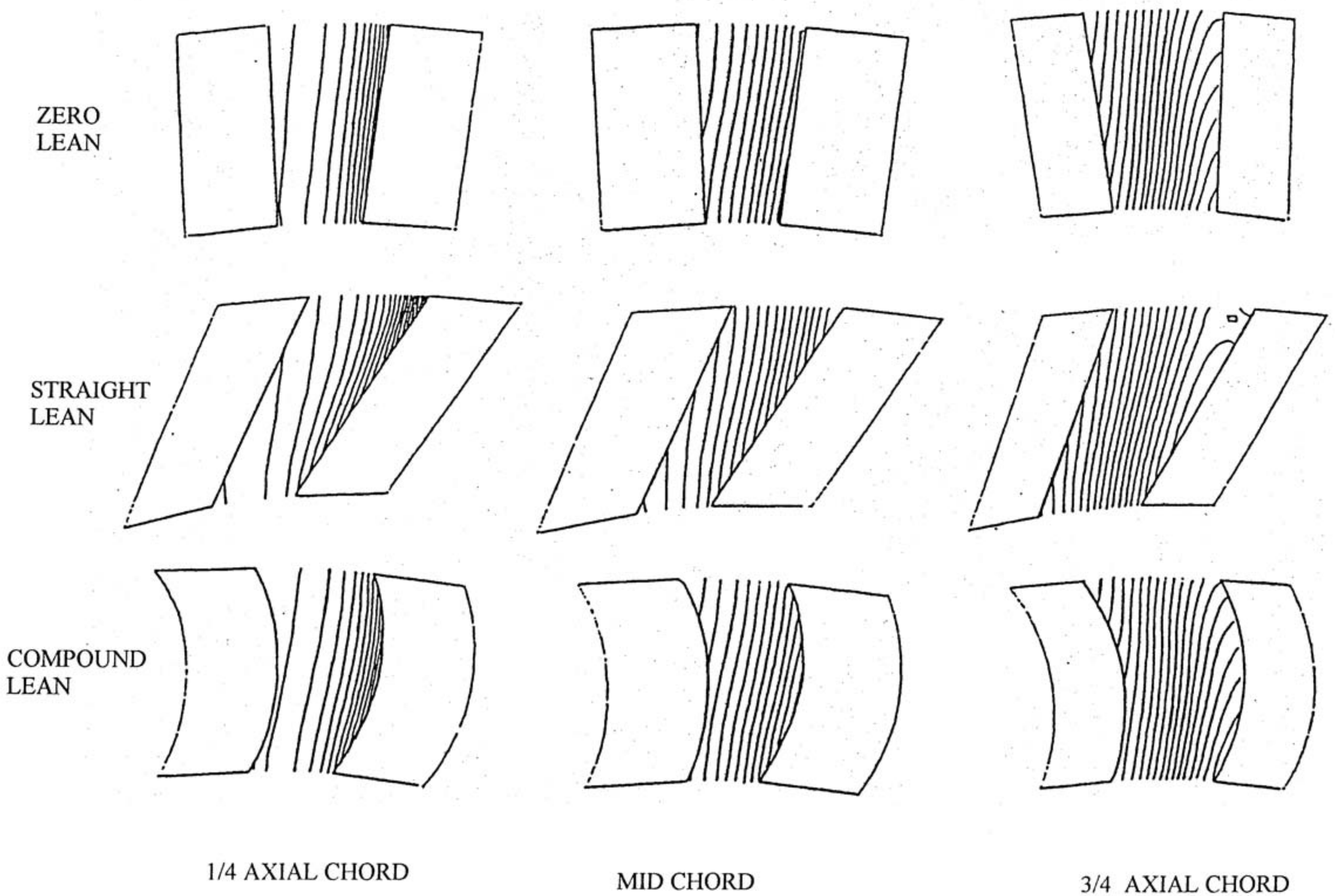
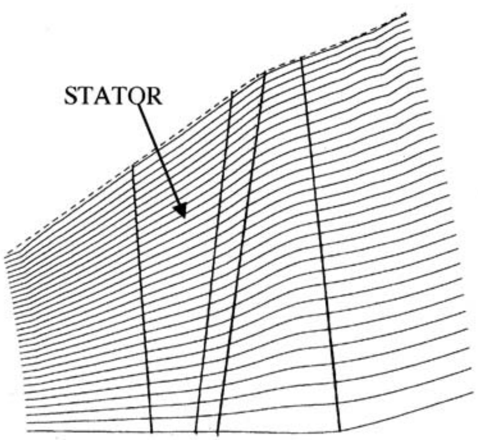


FIG 14. STATIC PRESSURE CONTOURS THROUGH A TURBINE STATOR WITH DIFFERENT STACKINGS. SUCTION SURFACE TO RIGHT OF PASSAGE.

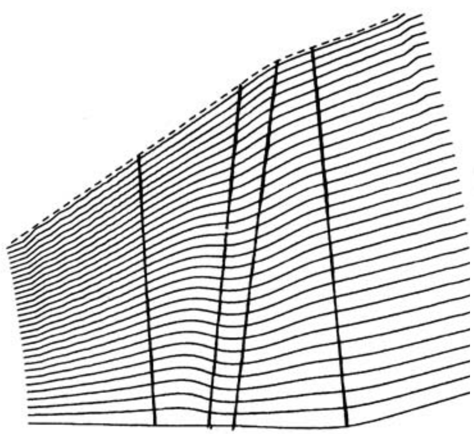
For high aspect ratio blades, leaning the stator, with the pressure surface inclined inwards, can be very beneficial in increasing the root reaction. This has been exploited in LP steam turbines where older designs often suffered from negative root reaction.

THE PRESSURE CAN BE INCREASED BY LEANING THE LAST STATOR BLADES AWAY FROM THE RADIAL DIRECTION SO THAT THEY EXERT A RADIALLY *INWARDS* FORCE ON THE STEAM PASSING THROUGH THEM.

THIS CAUSES THE STREAMLINES TO MOVE *OUTWARDS*.

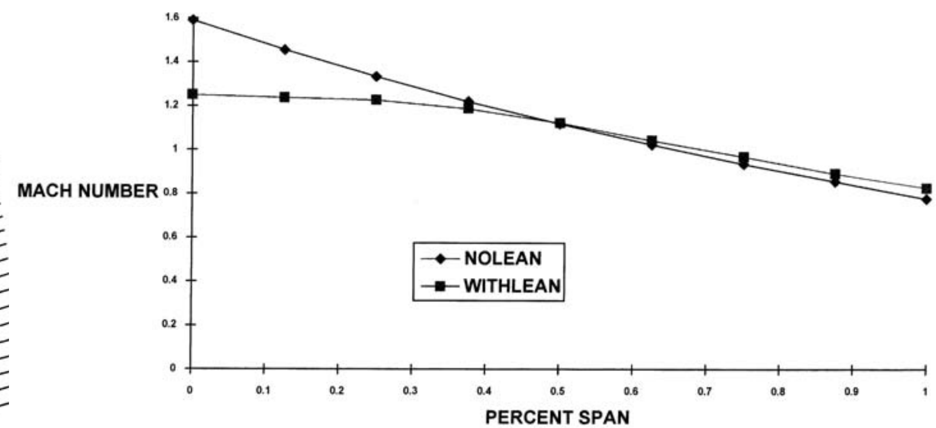


STREAMLINES WITH NO LEAN

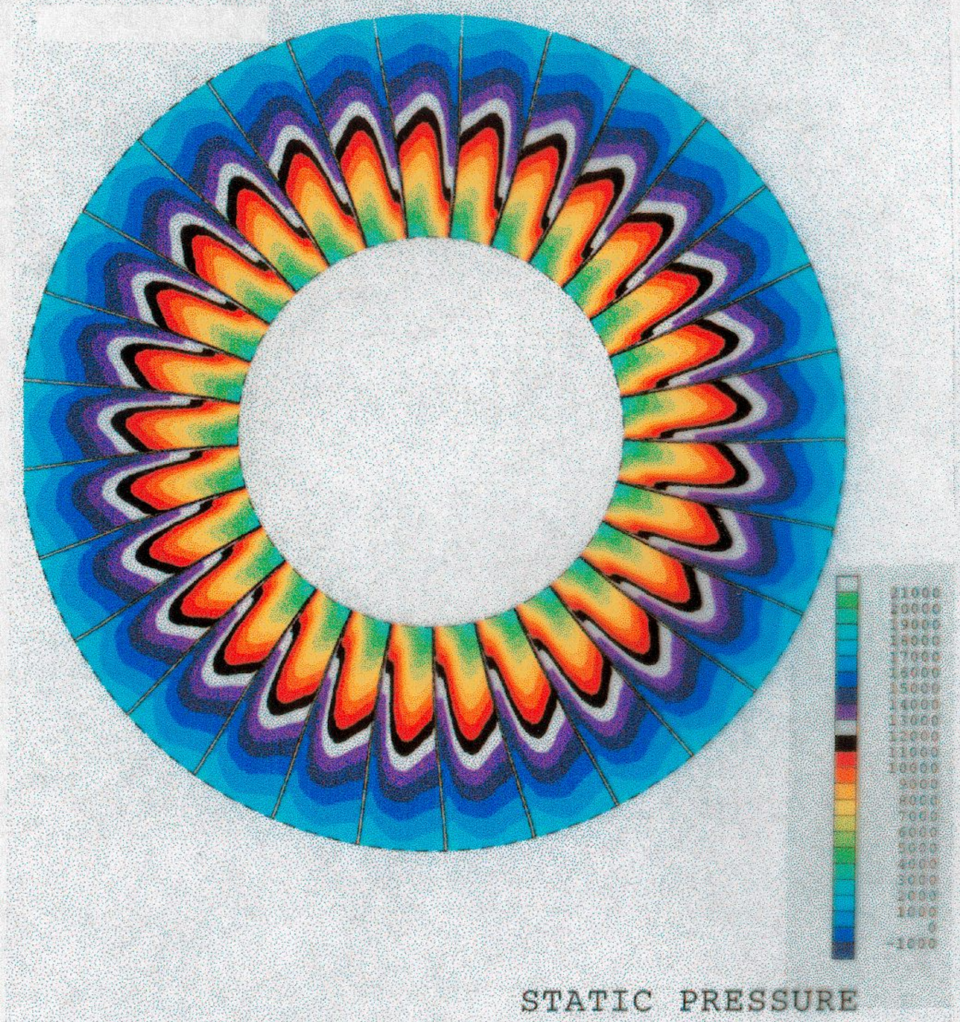


STREAMLINES WITH LEAN

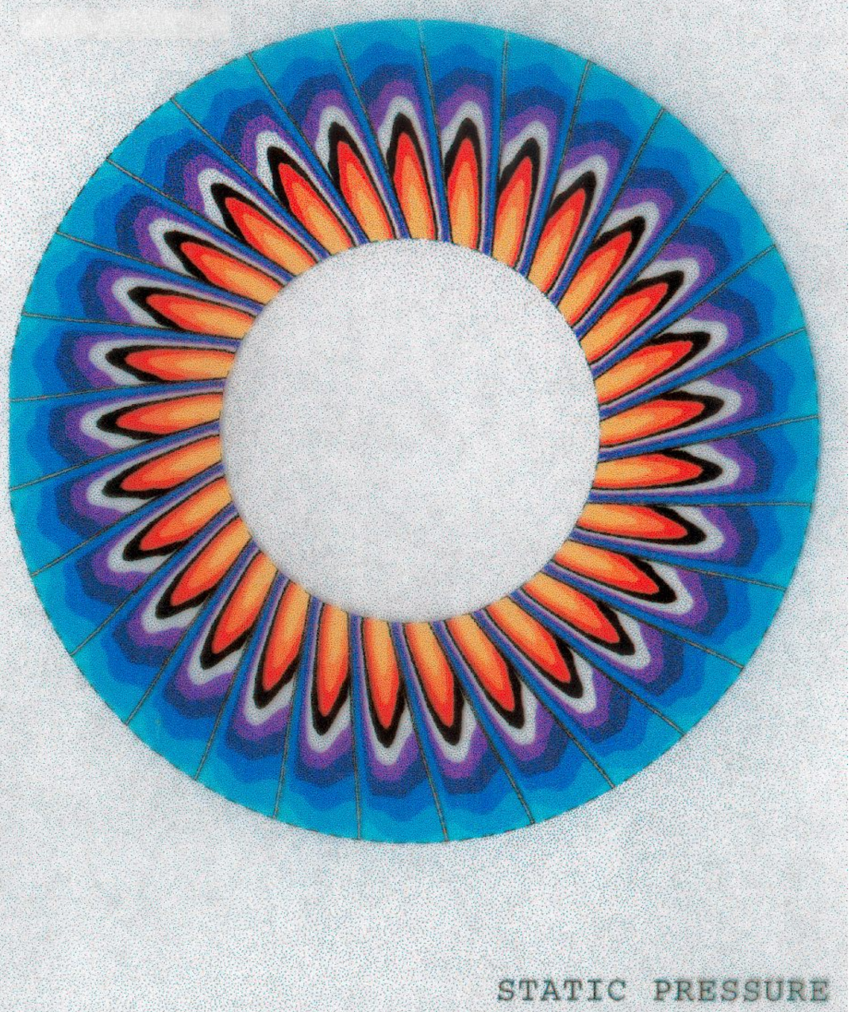
FIG 13. MACH NUMBER DISTRIBUTION AFTER THE LAST STATOR ROW OF A LARGE STEAM TURBINE



STATIC PRESSURES AT THE STATOR TRAILING EDGE



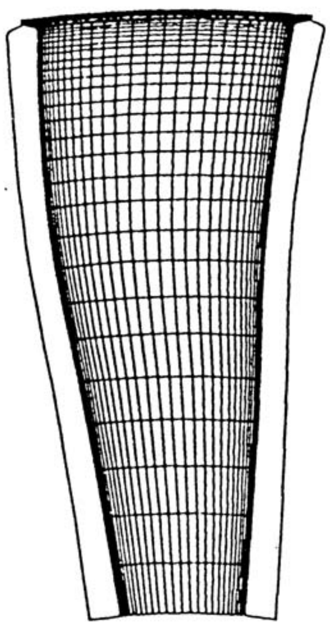
WITH NO LEAN ON STATORS

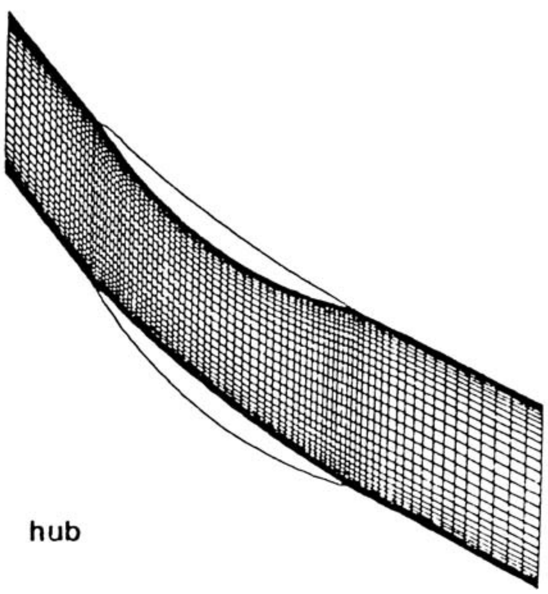


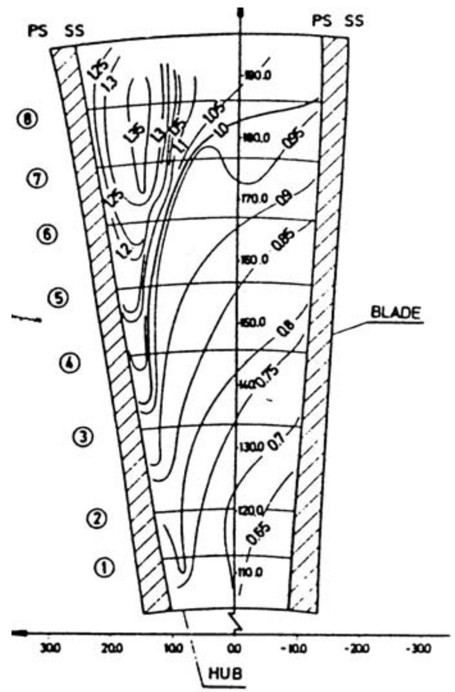
WITH LEAN ON STATORS

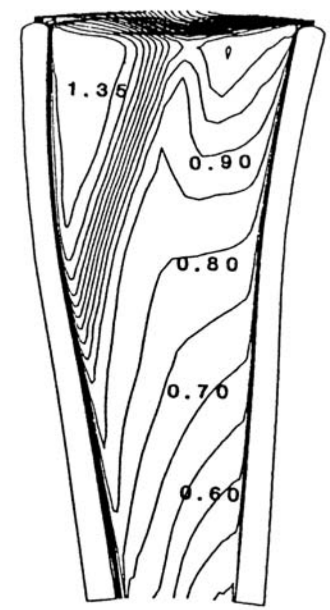
The move from Euler to Navier-Stokes solutions mainly depended on advances in computer power. This became available in the mid 1980’s. A widely used method was developed by Dawes.

Initially relatively coarse grids (33x60x33) were used with mixing length turbulence models and wall functions. Despite this useful results were obtained, especially for transonic fans.

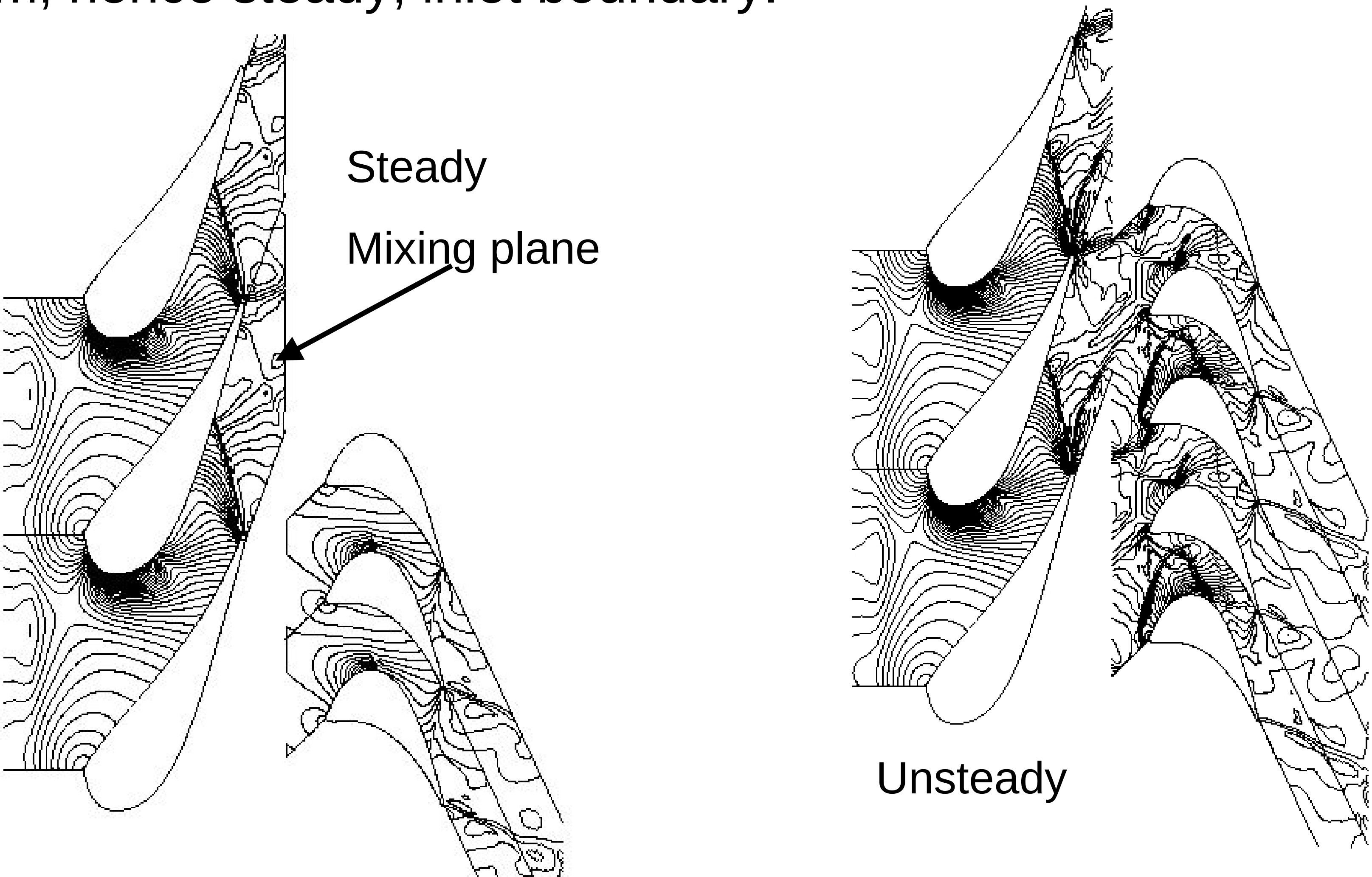








The next development, around 1990, was the ability to calculate multiple blade rows in a single steady calculation. This was achieved by the inclusion of mixing planes between blade rows so that each row "sees" a circumferentially uniform, hence steady, inlet boundary.

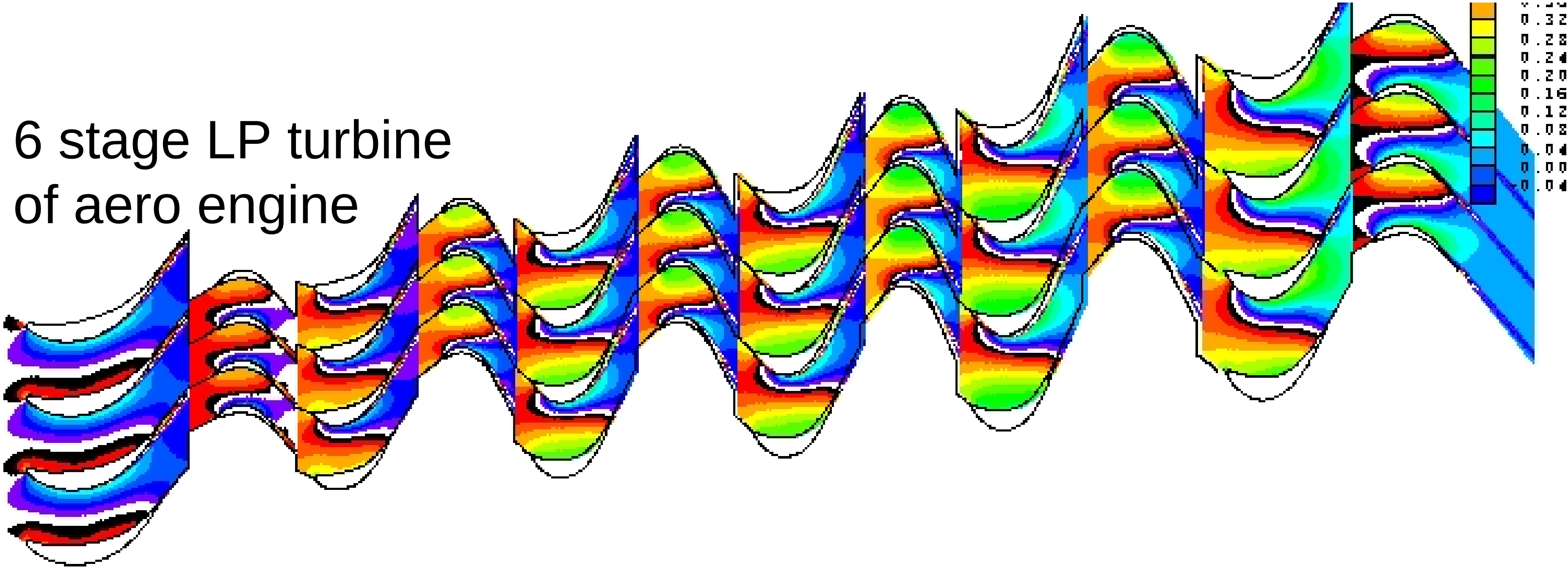


3D viscous calculations for multistage machines are now routine.

Formulation of a correct mixing plane model is one of the most difficult problems in CFD.

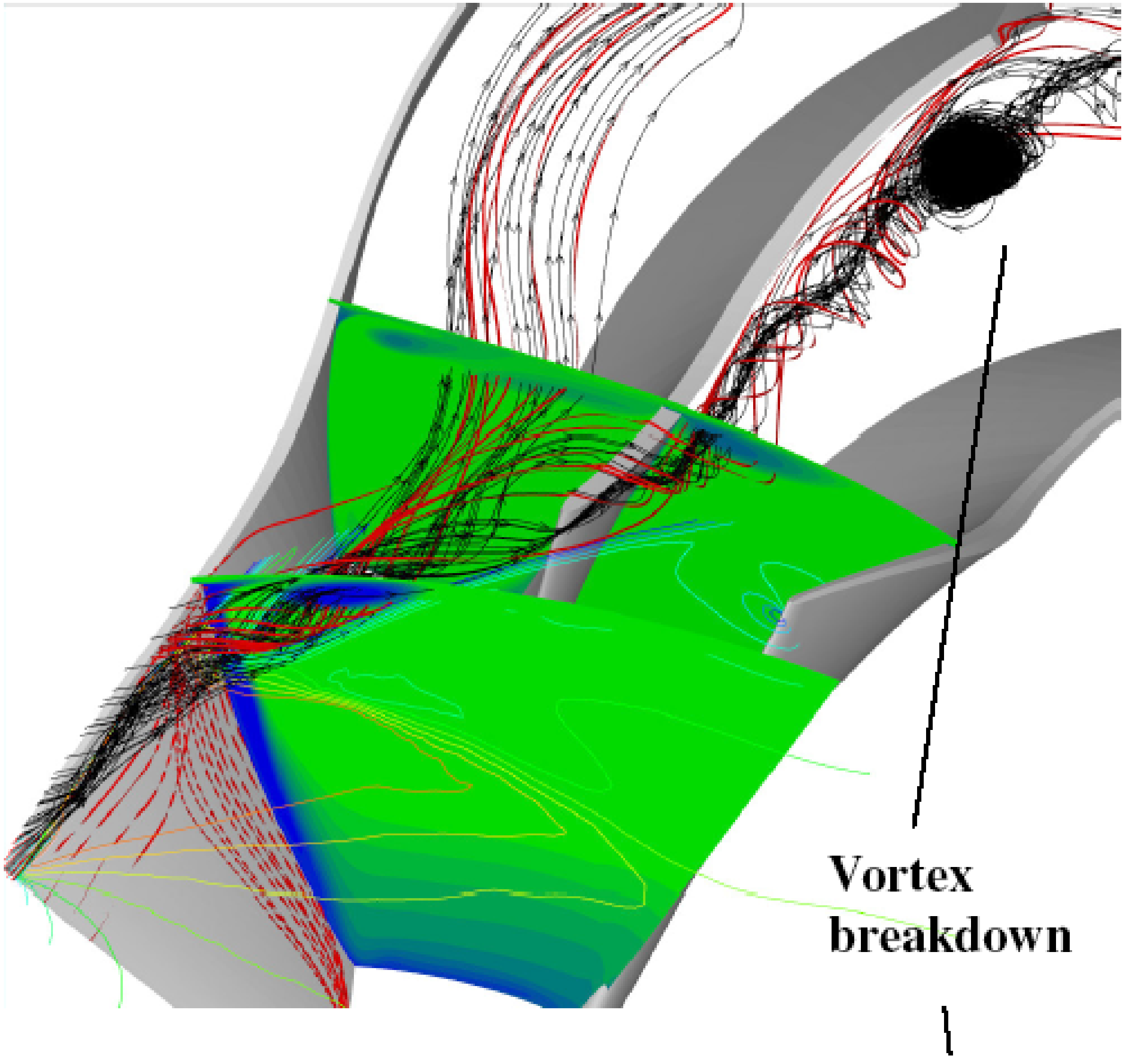
Adamcyzk has developed an alternative “average passage” model which claims to include some measure of the unsteady effects. This is slower and more complex but is widely used in the USA.

6 stage LP turbine
of aero engine

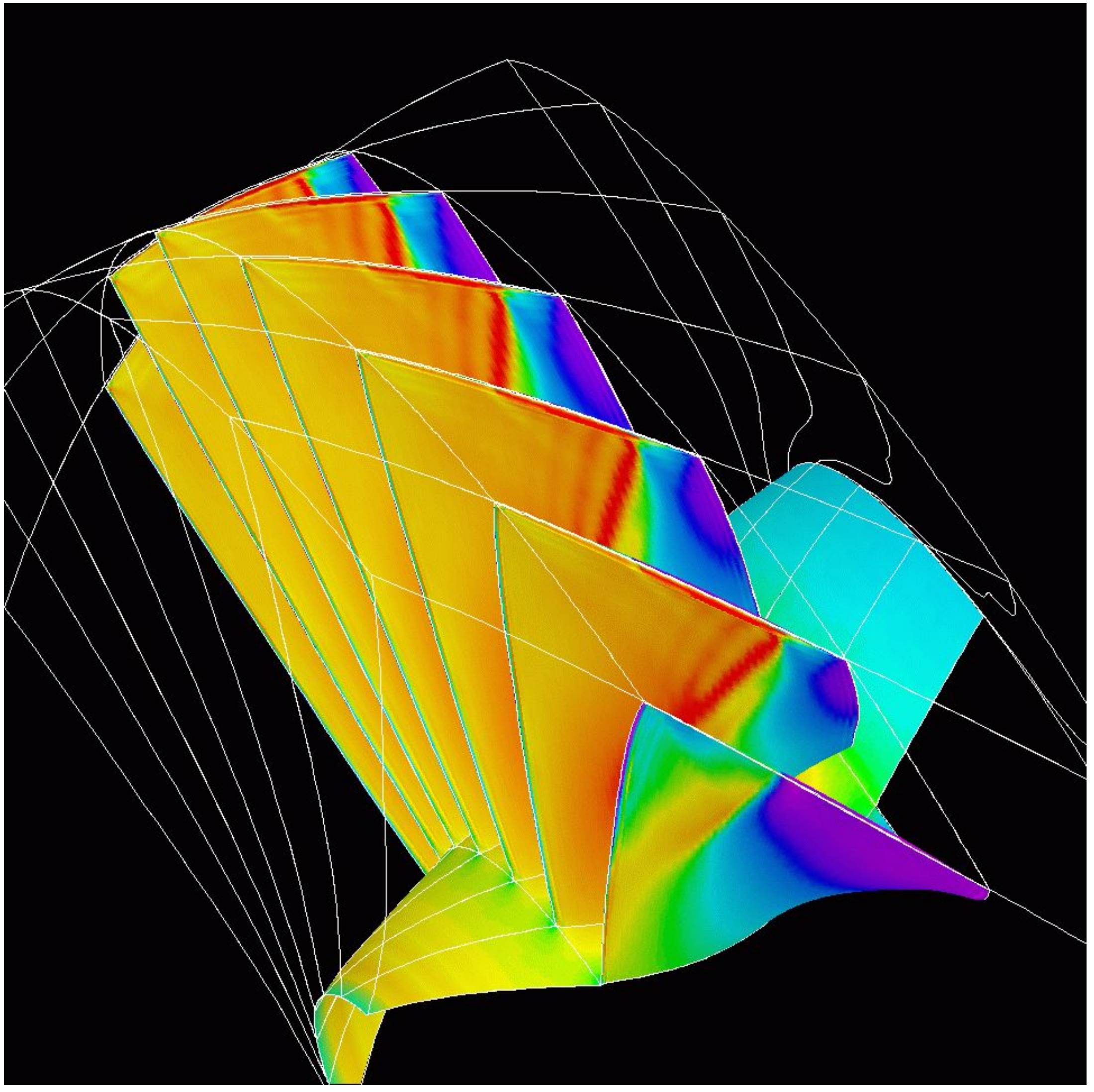


CFD is now an essential part of all turbomachinery design, including radial and mixed flow machines.

The flow in a centrifugal compressor is found to be dominated by tip leakage.



CFD can certainly generate some pretty pictures -- but does it always give the right answer ????



SOME LIMITATIONS OF CFD

It is very important to realise that CFD is not an exact science. As designers are more and more exposed to CFD results and less and less to experimental results it is very important that they understand what CFD results can be trusted and what can not.

This is particularly important when CFD is used in conjunction with optimisation software to produce an "optimum" design within certain constraints.

The optimiser will very likely exploit weaknesses in the CFD.

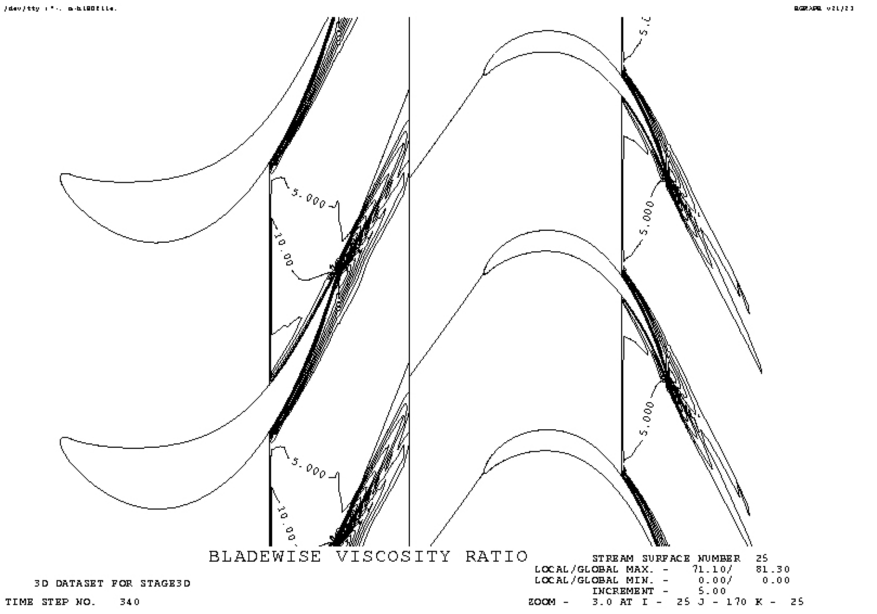
There are many things that we cannot predict accurately with CFD, these include:

- Boundary layer transition
- Turbulence modelling
- Endwall loss
- Leakage loss
- Compressor leading edge flow
- Turbine trailing edge flow
- Effects of small geometrical features
- Unsteady losses

Boundary layer transition

is influenced by:

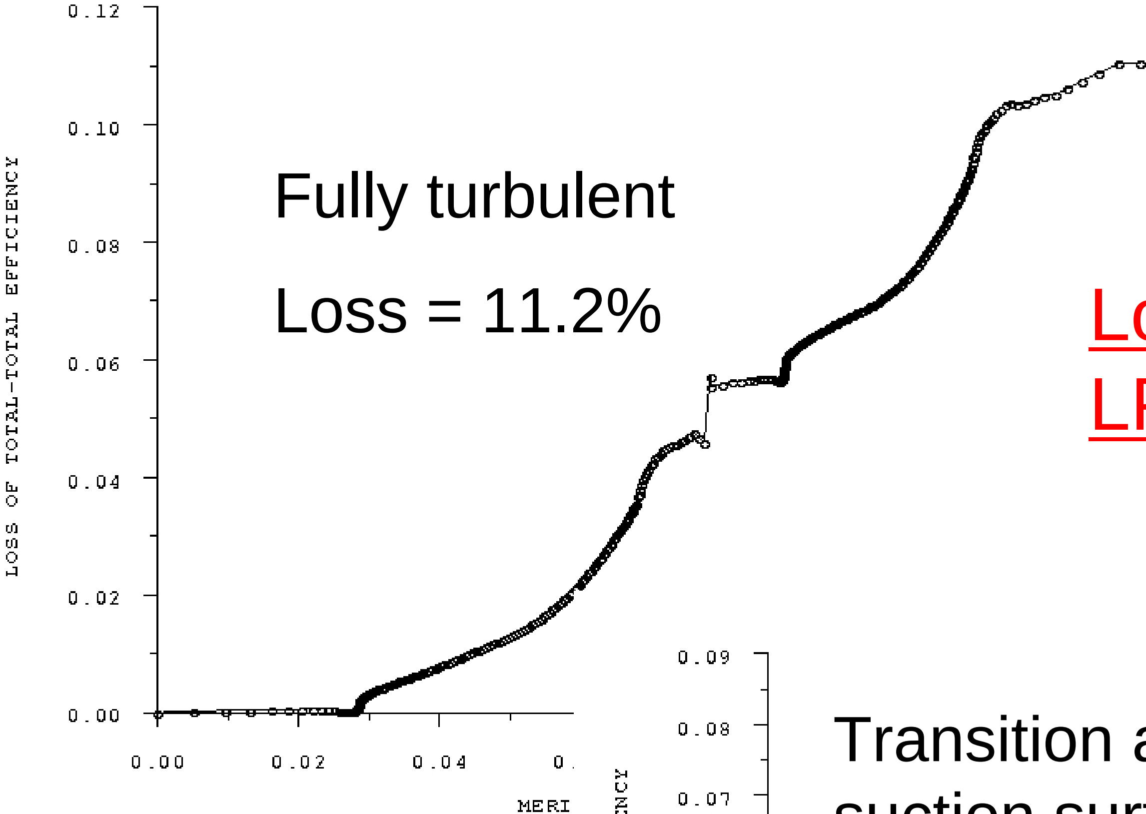
- Pressure gradient
- Reynolds number
- Turbulence level
- Surface curvature
- Surface roughness
- 3D flow



Contours of turbulent viscosity

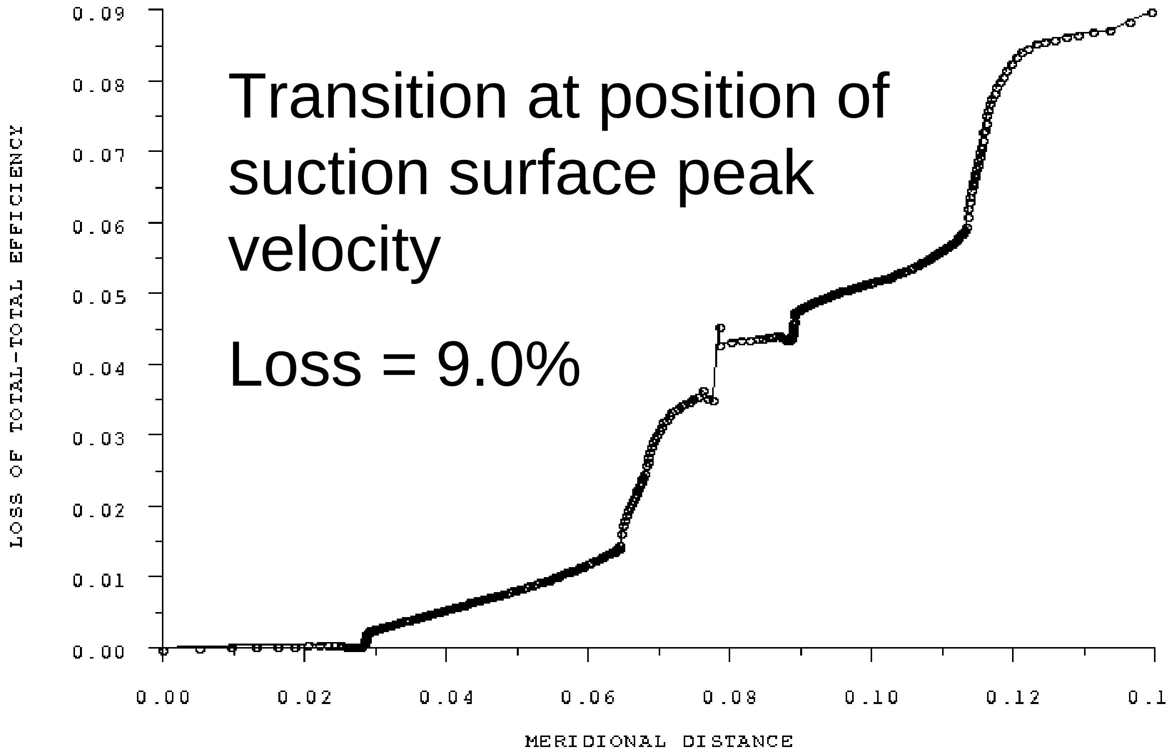
We cannot predict it accurately except under very idealised conditions. It can have a large influence on the efficiency at low Reynolds numbers (< 5×10^5).

Lost efficiency of a LP turbine stage



Fully turbulent

Loss = 11.2%



Transition at position of suction surface peak velocity

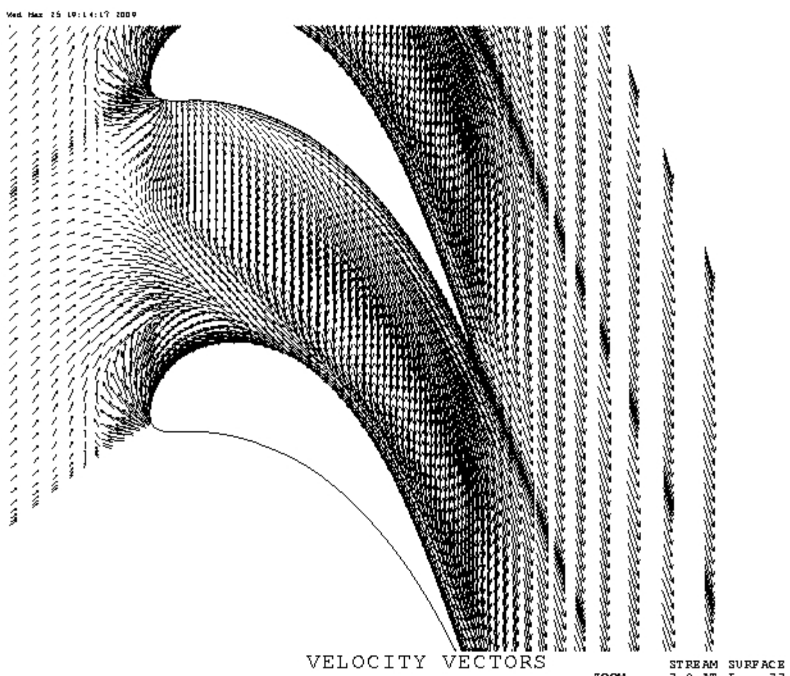
Loss = 9.0%

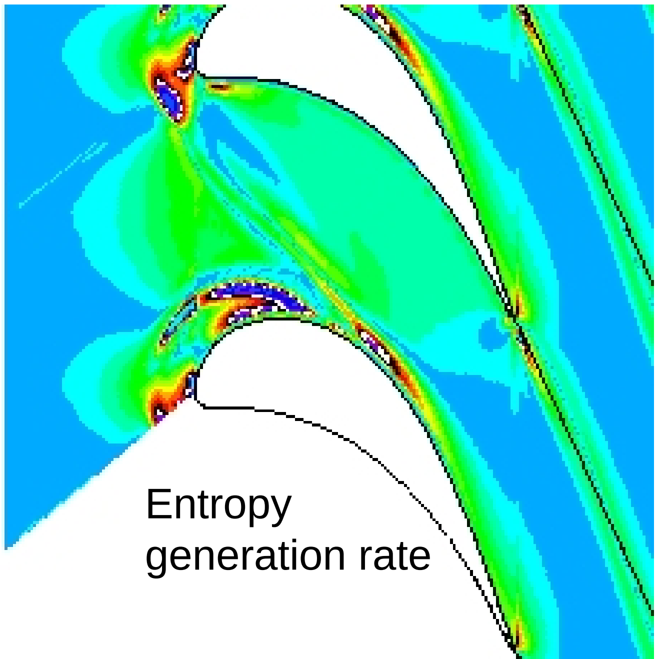
ENDWALL (or secondary) LOSS

Secondary flow and endwall loss in both turbines and compressors is mainly determined by the thickness and skew of the annulus boundary layers.

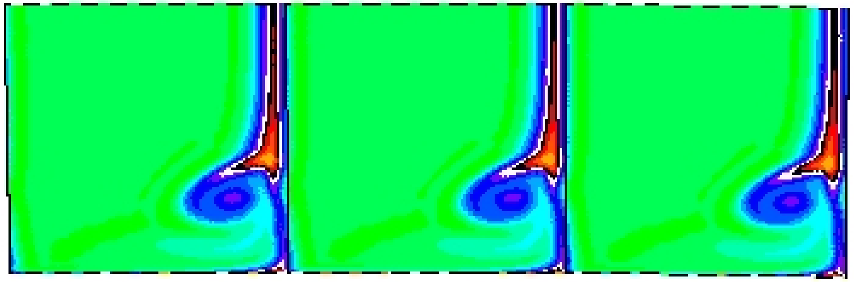
In a real machine we do not know either the thickness or the skew of these boundary layers. They are largely determined by leakage flows, cavities and steps in the upstream hub or casing.

In addition part of the new endwall boundary layer after the separation line is likely to be laminar. We cannot predict this.



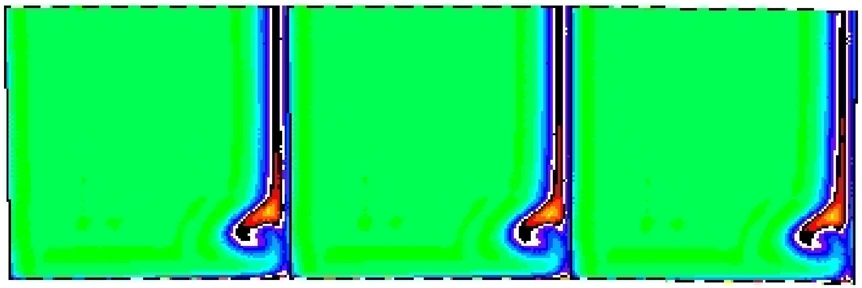


PREDICTED LOSS OF A TURBINE CASCADE WITH DIFFERENT INLET ENDWALL BOUNDARY LAYERS



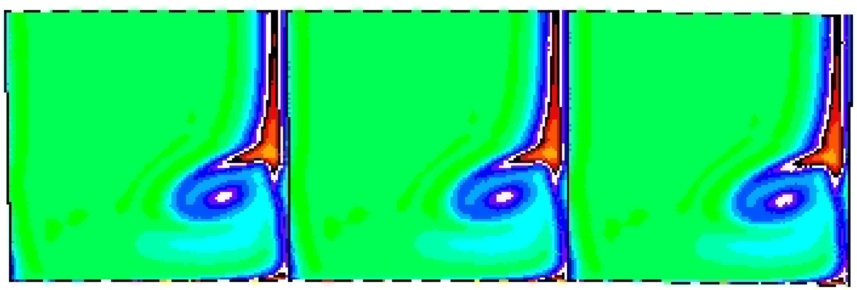
Datum inlet BL

$Y_{sec,net} = 2.6\%$



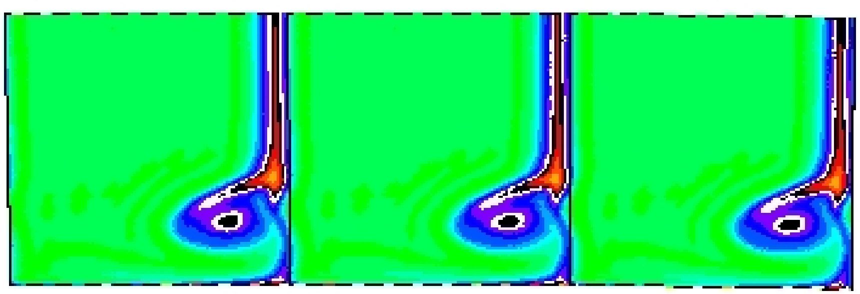
Thin inlet BL

$Y_{sec,net} = 2.05\%$



Thick inlet BL

$Y_{sec,net} = 2.8\%$



Positive skewed inlet BL,

$Y_{sec,net} = 3.3\%$

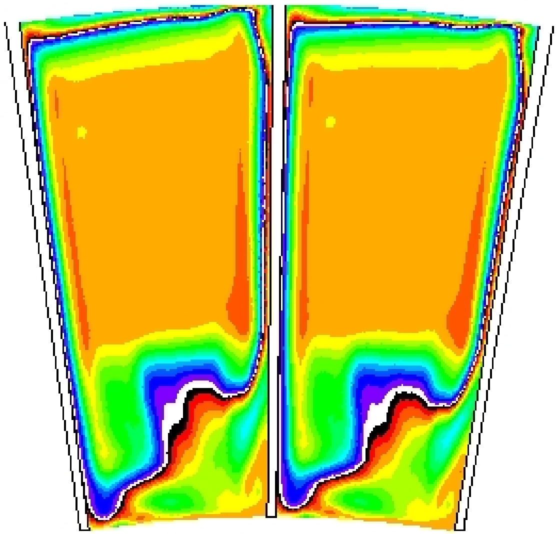
TURBULENCE MODELLING

Different turbulence models can give very different results, as can different choices of the constants in any one model.

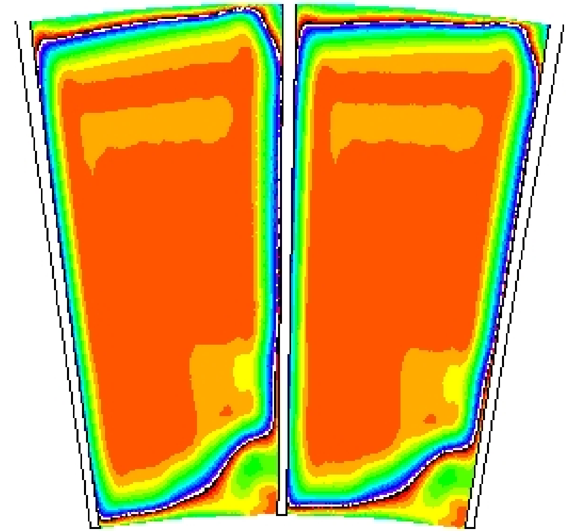
The difference occurs mainly when there are separations present. The separation point is usually reasonably well predicted but the size of the separation is very dependent on the model.

This is especially important in compressors and makes prediction of the stall point particularly difficult.

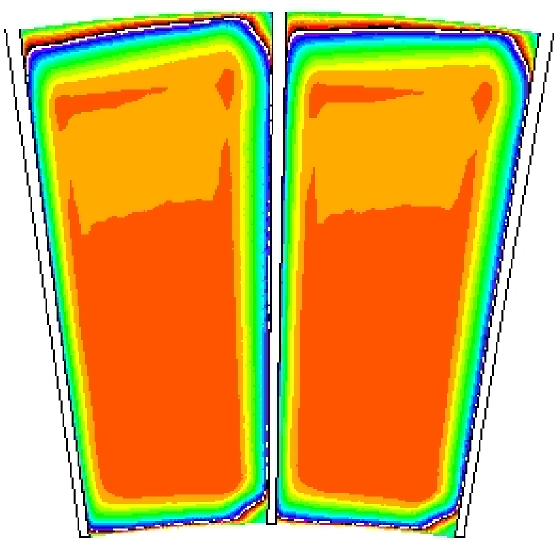
It seems unlikely that any significant improvement in turbulence models will be possible until DNS becomes a routine tool - if ever.



0.01 - stalls

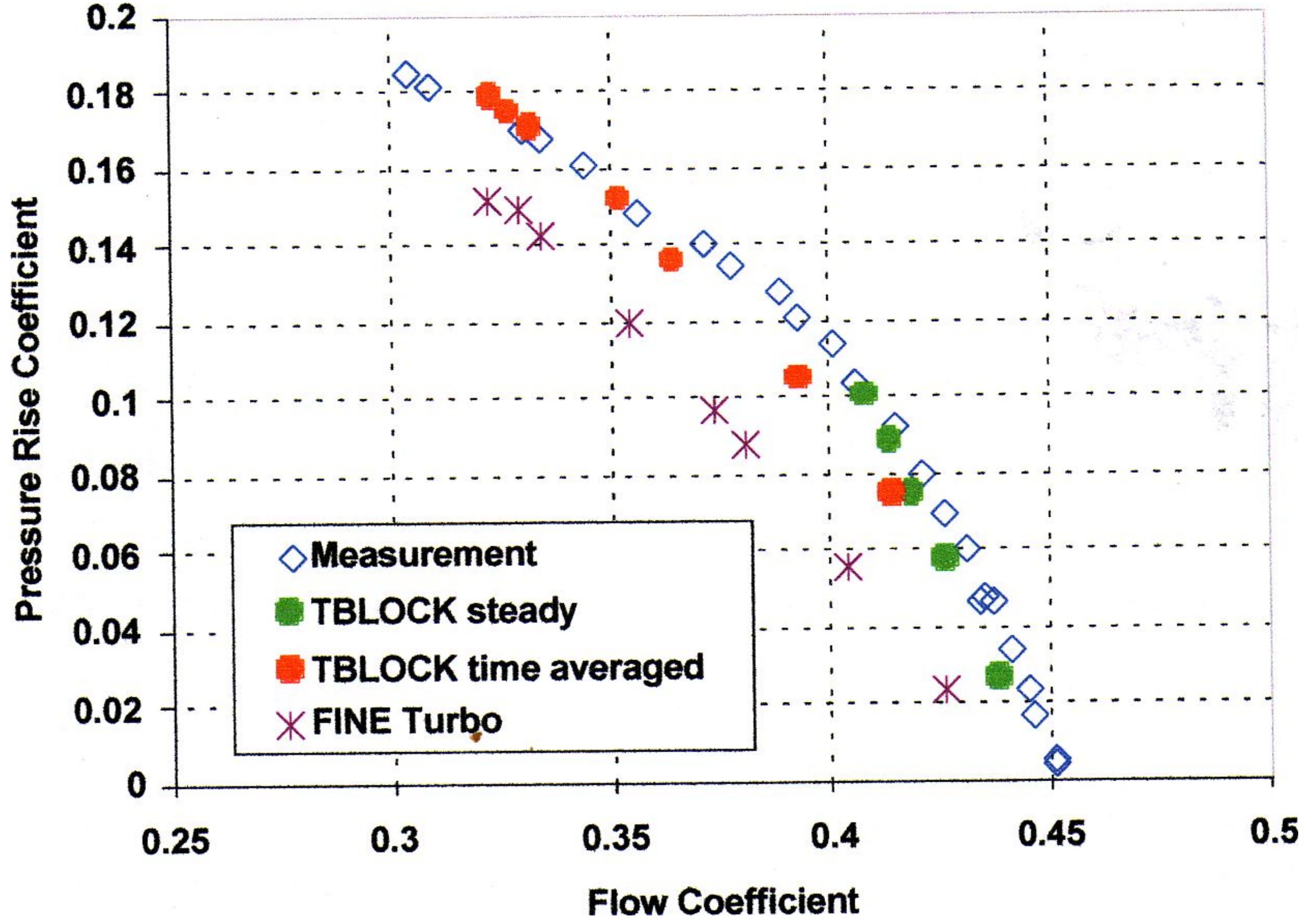


0.02



0.03

Hub separations in the stator of the MHI compressor with different mixing length limits.

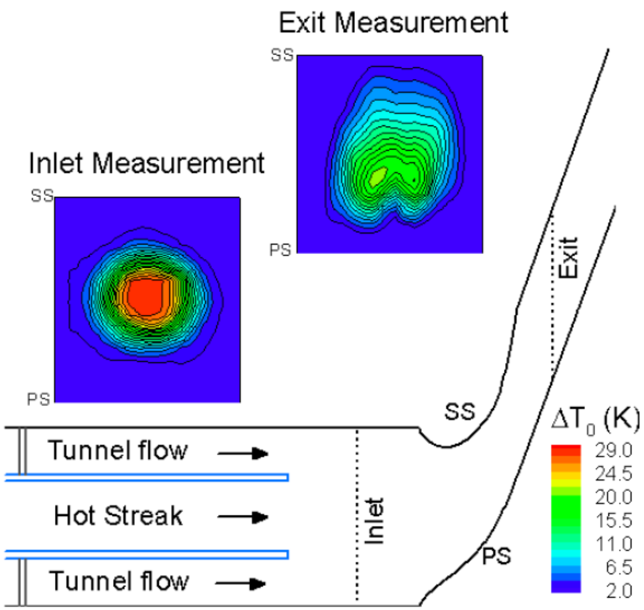


Prediction of compressor stall point is often based on steady calculations. This is not reliable. Unsteady calculations give better predictions but are still not good. Whole annulus unsteady calculations are needed.

FREE STREAM TURBULENCE

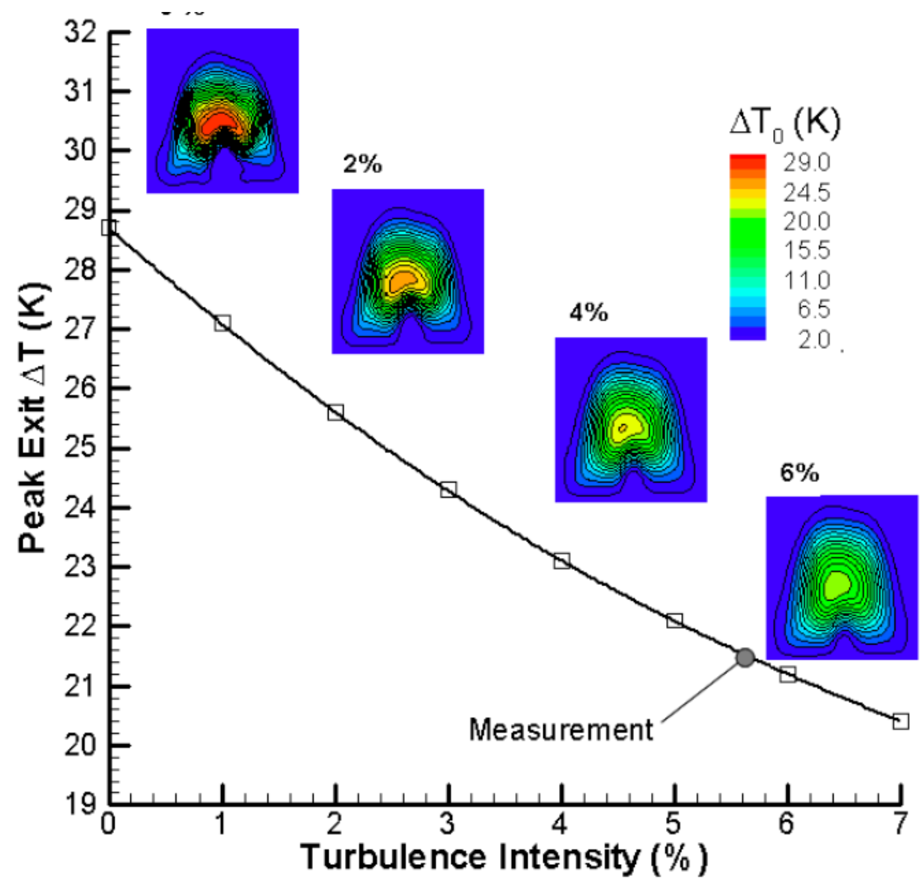
- has a major effect on the diffusion of temperature and entropy.

In a real machine the turbulence intensity and length scales are scarcely ever known.



ig. 4 Hot streak injection on the duct and measured

J Ong’s experiment



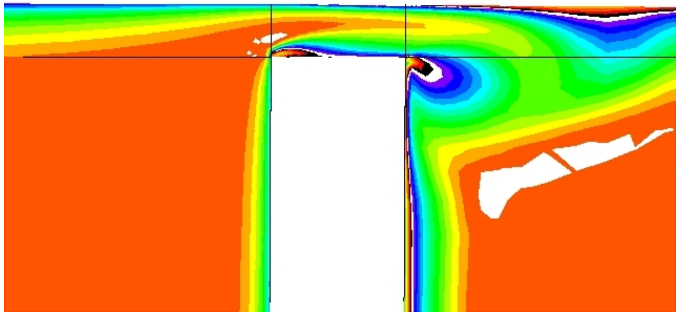
ı. 5 Effect of free stream turbulence on temperature tributions.

TIP LEAKAGE FLOWS

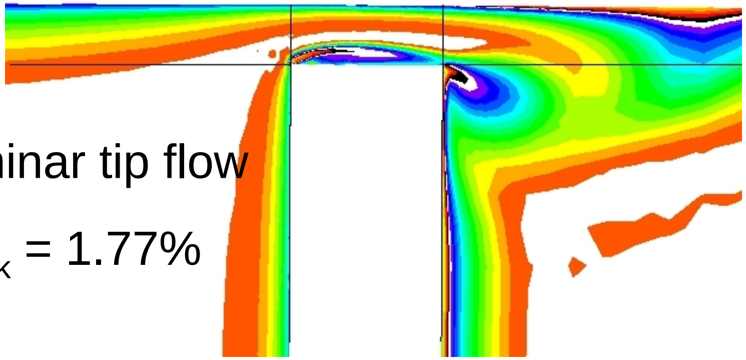
Tip leakage flows are often surprisingly well predicted by CFD - but :-

In practice we seldom know the tip gap accurately. The exact shape of the pressure surface to tip gap corner is also important.

The contraction of the leakage jet is very dependent on the viscous modeling in a region where the flow is changing extremely rapidly. This directly affects the leakage flow rate and loss.

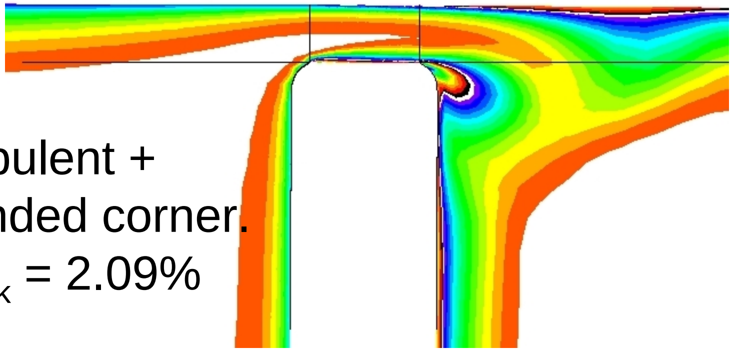


Turbulent tip flow.
M_{leak} = 1.81%



Laminar tip flow

M_{leak} = 1.77%



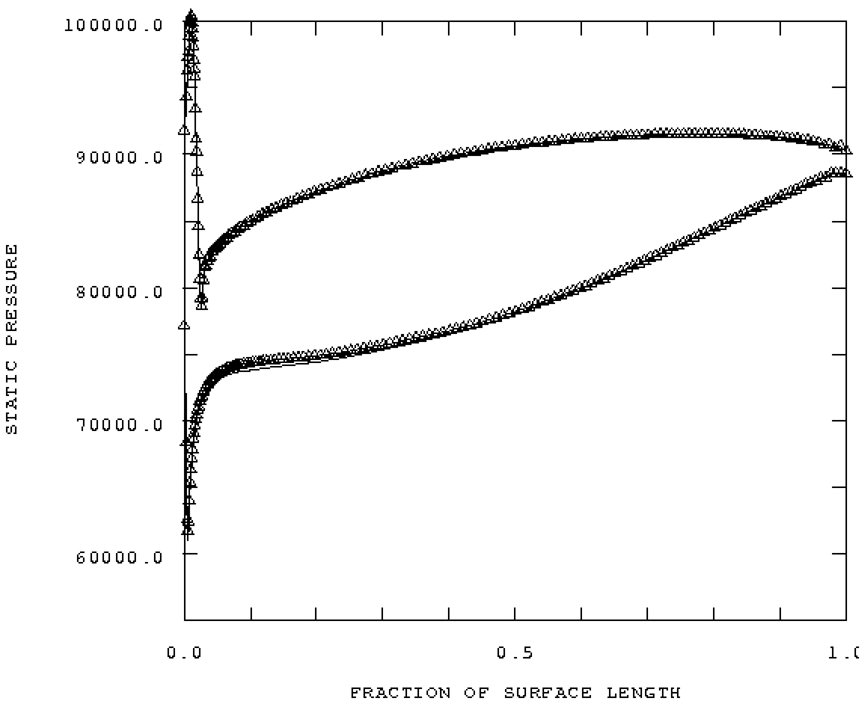
Turbulent + rounded corner.
M_{leak} = 2.09%

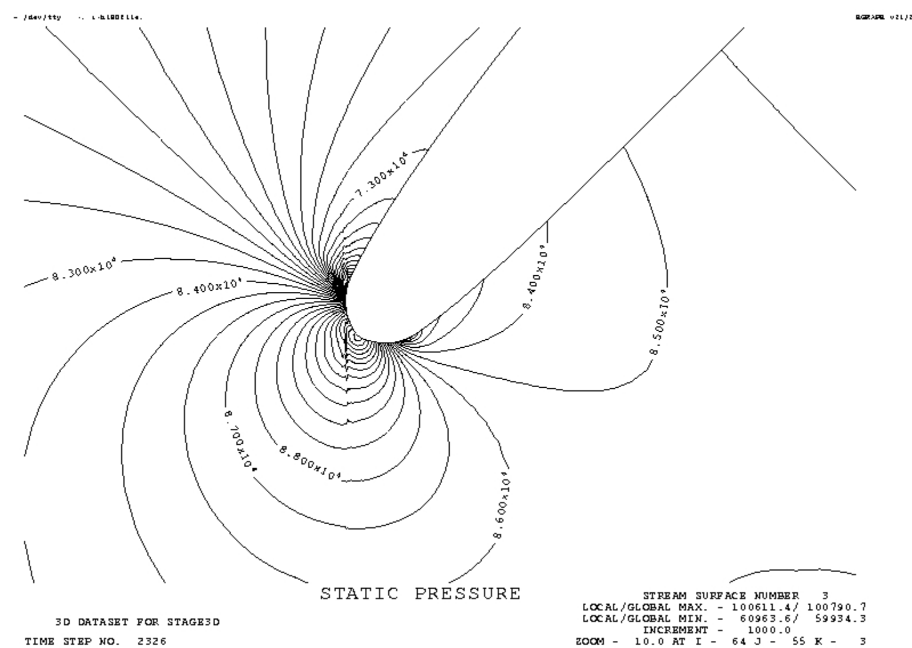
COMPRESSOR LEADING EDGES

Compressor leading edge flow is very difficult to predict accurately.

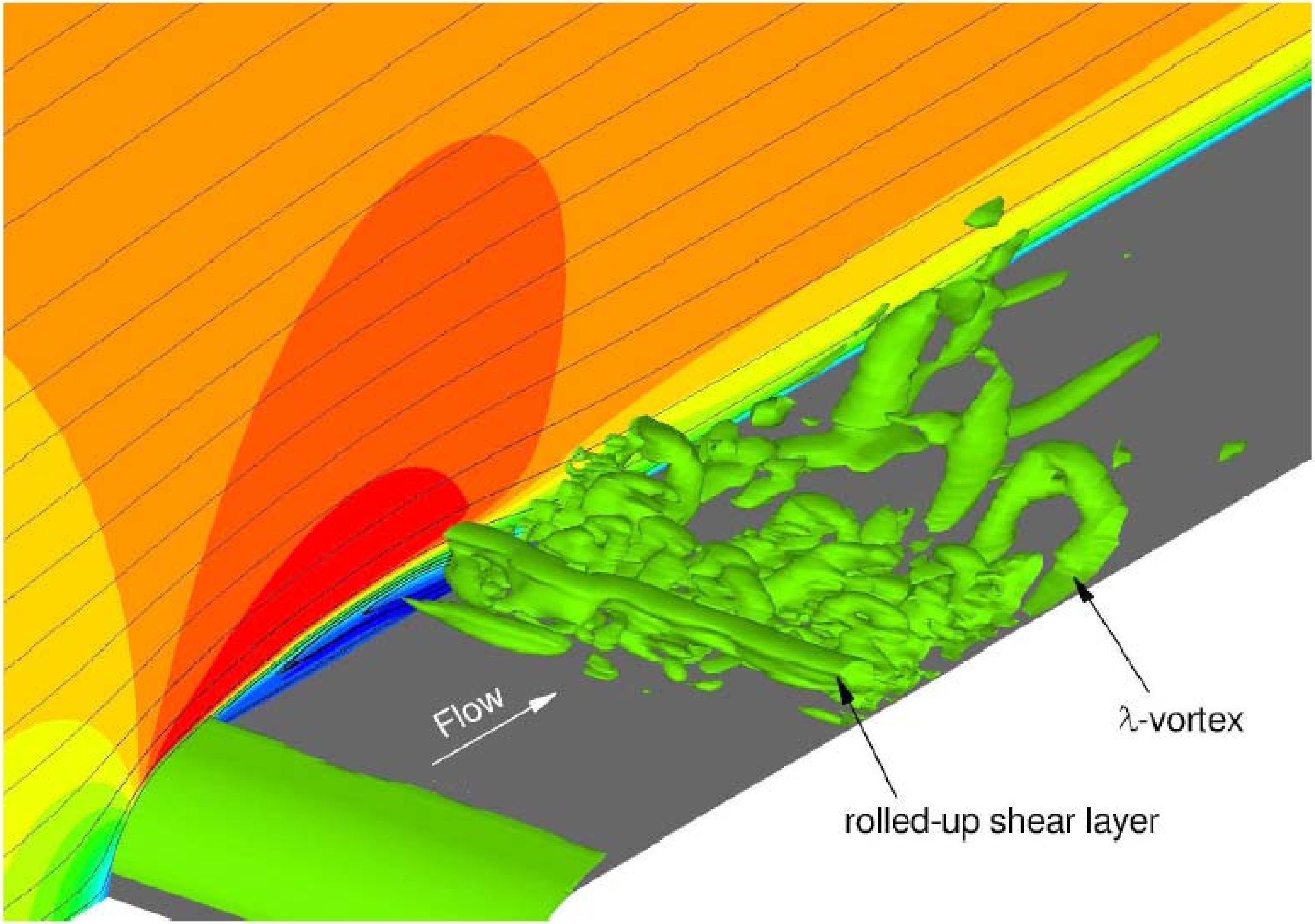
The velocity can change from zero to supersonic in a distance of 0.5mm. Calculations usually predict a “spike” in velocity immediately downstream of the LE.

This can have a major effect on the development of the downstream boundary layer and hence on overall loss.





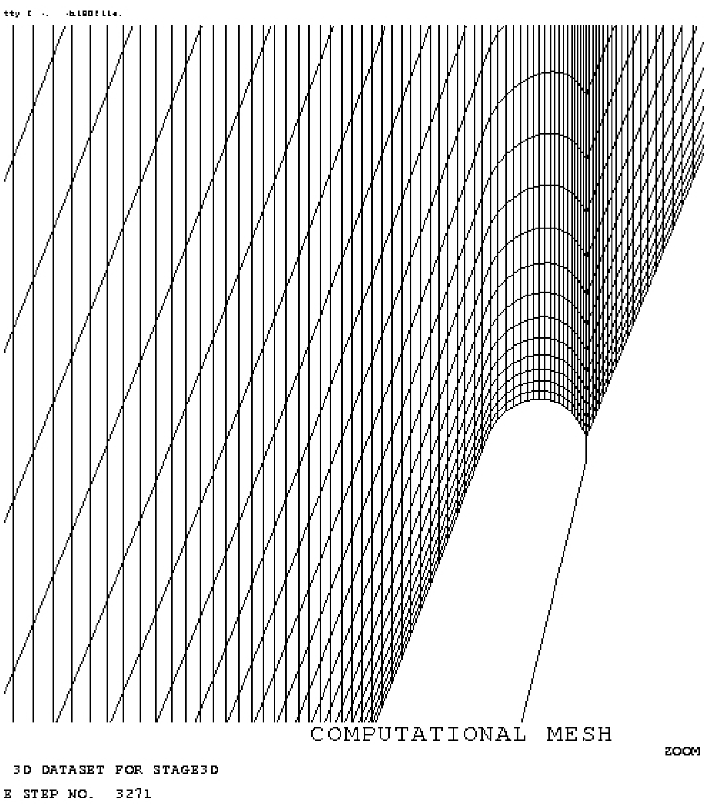
This is what the Leading edge flow is really like.

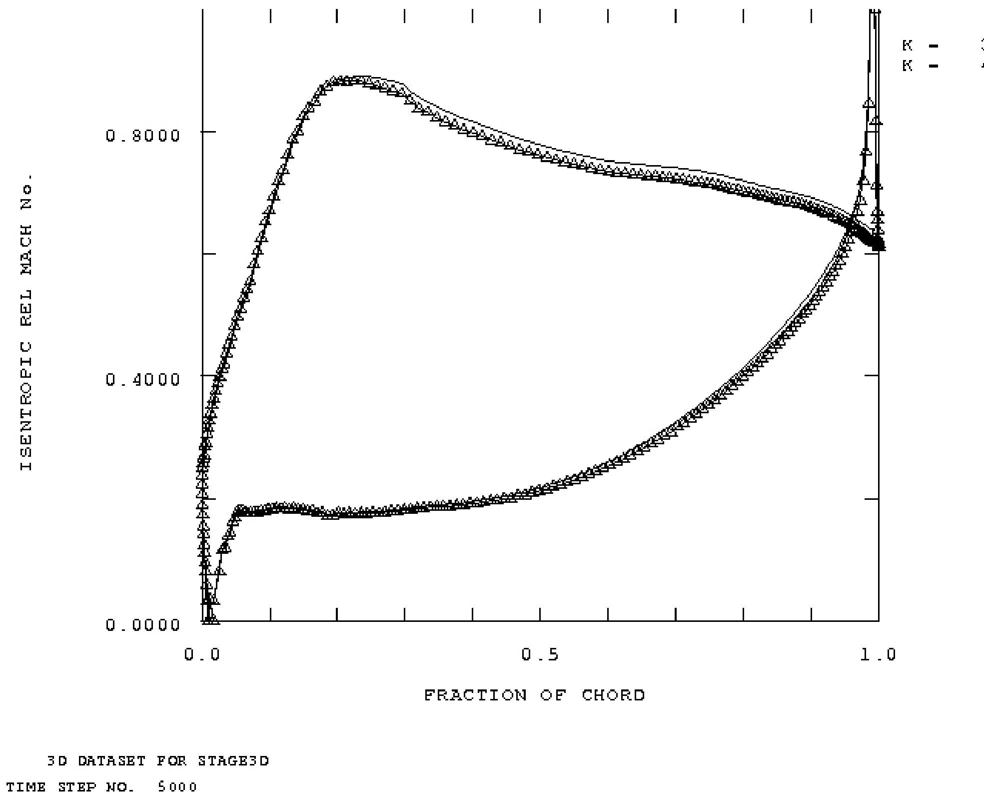


TRAILING EDGE FLOWS

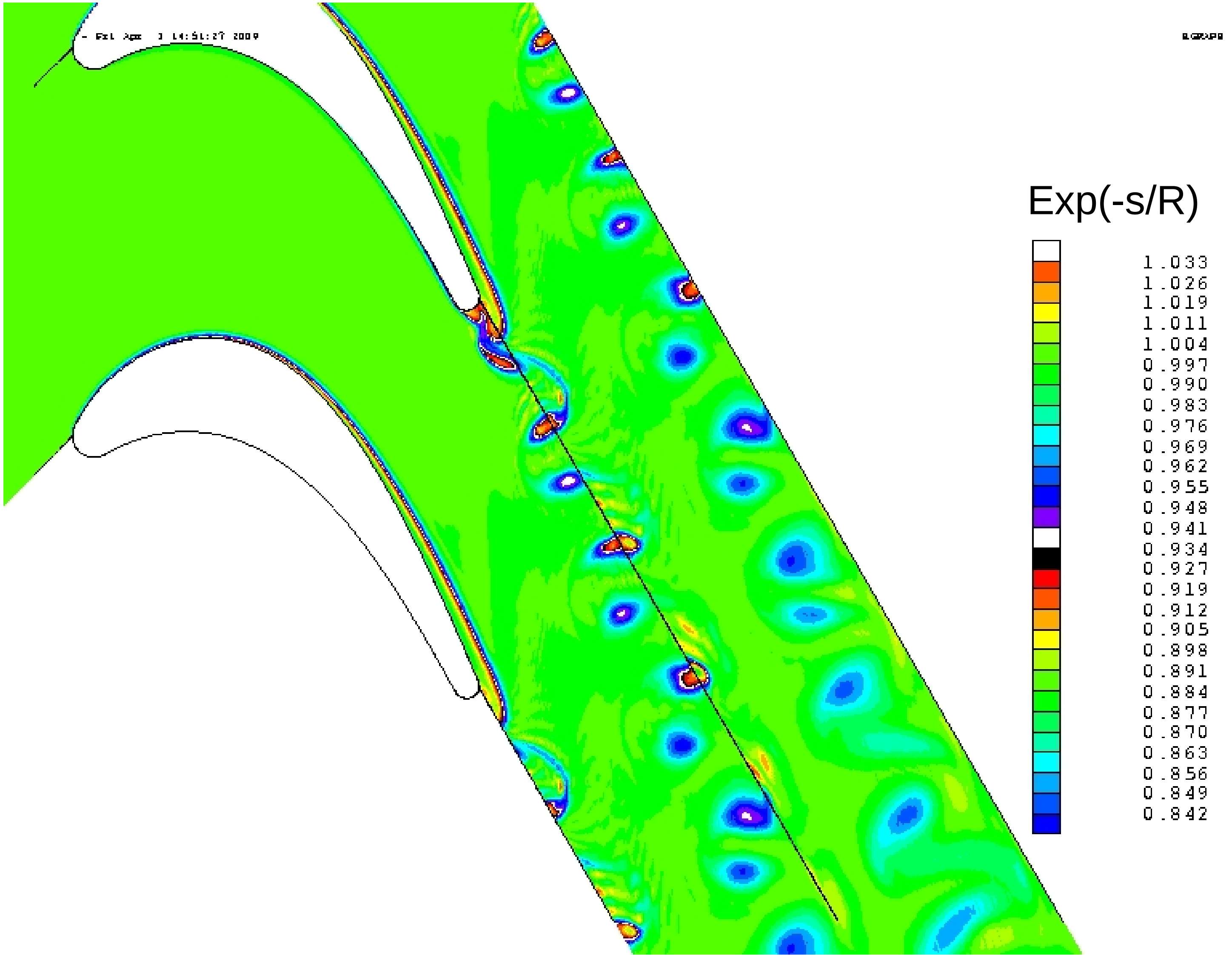
For blades with a thick trailing edge it is very common for CFD to predict negative loading at the trailing edge.

This is never found in experiments. It causes underturning, low base pressure and increased loss





The real flow is usually unsteady with vortex shedding but the time-average flow usually shows zero loading at the TE - in subsonic flow.

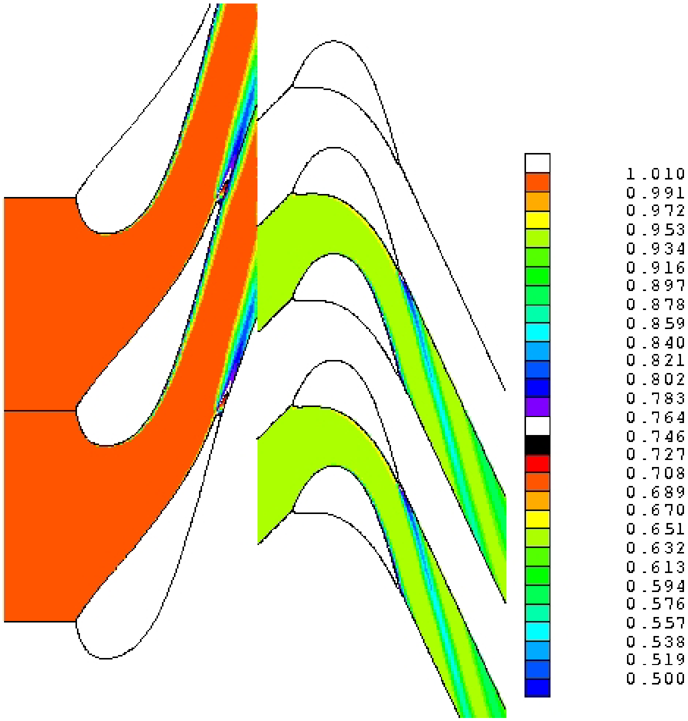


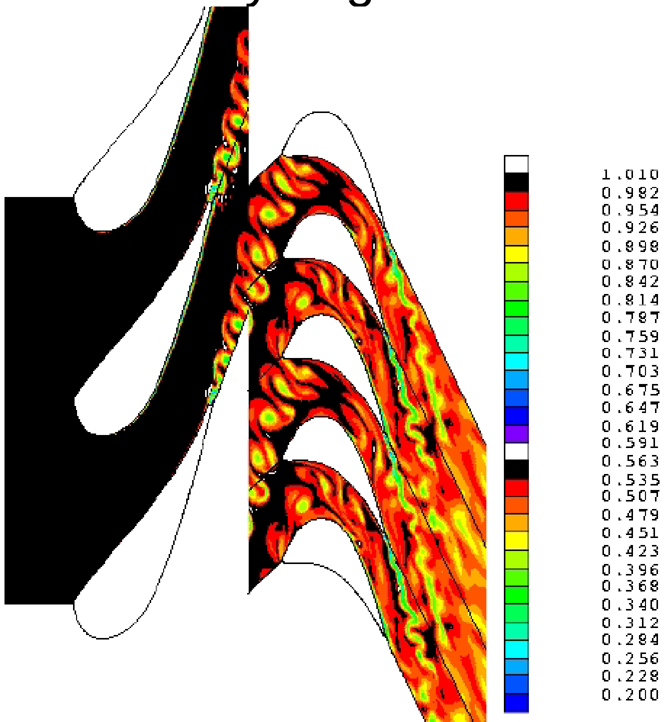
MIXING PLANES

Mixing plane models assume that the flow mixes instantaneously to a pitchwise uniform flow at the mixing plane. This generates a mixing loss.

In reality the mixing continues through the next blade row end the mixing takes place as an unsteady process. It is unlikely that the loss generated will be exactly the same.

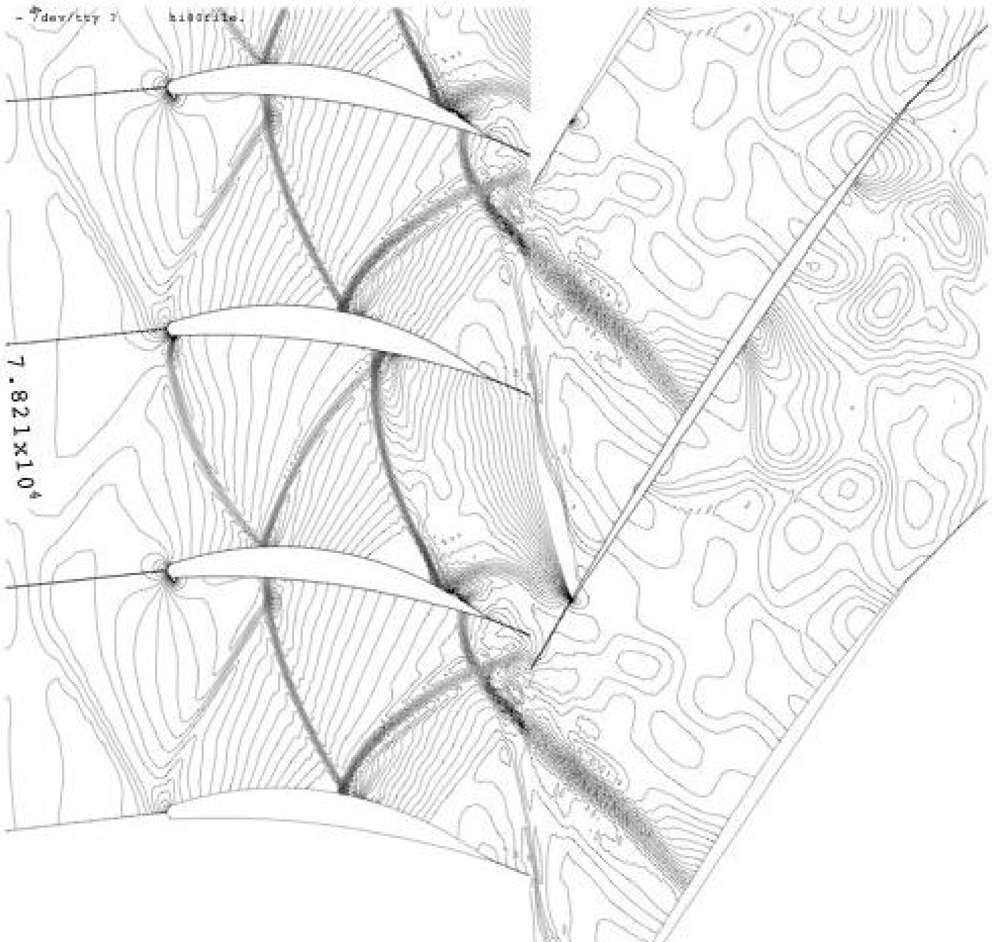
Some authors claim that there is a significant difference between the loss predicted by unsteady calculations and by mixing plane calculations but the author has never predicted any large difference.



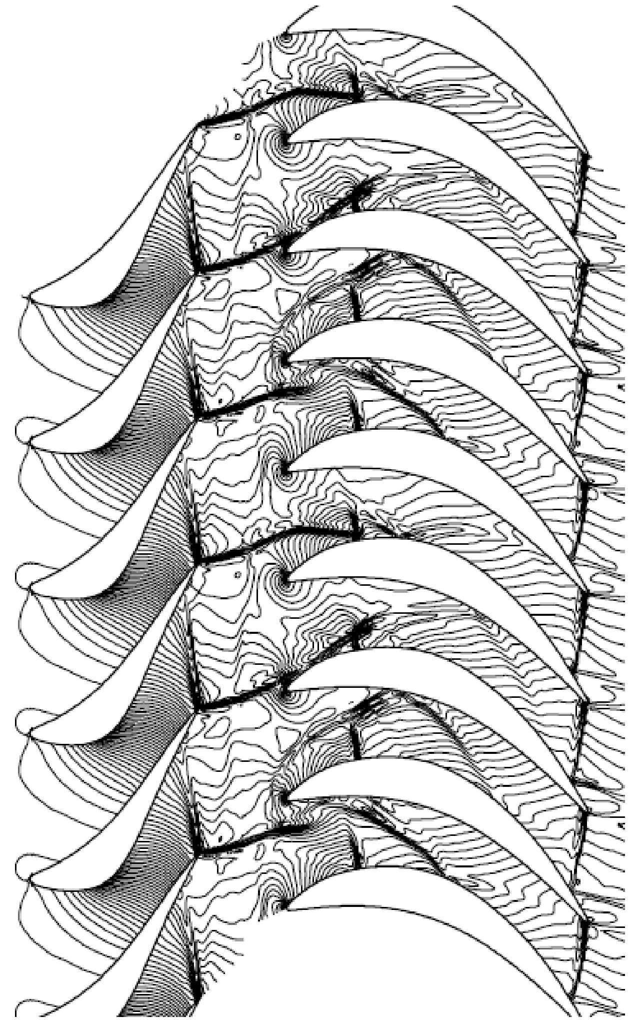


UNSTEADY FLOW

Should we ever believe a steady calculation ??



Compressor first rotor + IGV



Near hub of last stage steam turbine

So, errors in CFD may be due to:

Modelling errors	Turbulence, transition, mixing planes
Unknown boundary conditions	Endwall boundary layers, Free stream turbulence, inlet profiles, cooling and leakage flows
Unknown geometry	Tip gaps, leading edge shape, sharpness of corners, blade deflection and deformation.

If any of these are present (and they almost always will be) then CFD predictions should be treated with some reservation.

- They should usually be used on a comparative basis rather than as an absolute prediction of performance.
- Inspect the results from computer optimisation very carefully to check that they are realistic.
- Always study the details of the CFD solution to try to understand the basic Physics. One can often decide on good or bad features of the flow even when their effects cannot be quantified.

Overall Conclusions

- There are only limited possibilities for further improvements in design methods.
- Improvements in machine performance will come from attention to small details such as:
    - tip and seal clearances,
    - leading and trailing edge shapes and thicknesses,
    - reduced size of hub and casing steps and cavities,
    - better use of cooling flows.
- There is still need for experimental testing - but use it to calibrate CFD.
- Use the experiments and CFD to understand the flow Physics - and then think how the flow can be improved.

CONGRATULATIONS TO

PCA ENGINEERS

ON 20 YEARS OF SUCCESS AND

BEST WISHES FOR CONTINUED

SUCCESS IN THE FUTURE