Computational methods for design and optimisation

Design approaches

The idea is to combine computational predictions of performance of a component or even a whole configuration with automated procedures to generate new trial designs, and search for a better, or the best possible (optimal), design.

The two design approaches are *direct* and *inverse*.

Direct

1. Start with some initial configuration (e.g. an airfoil profile)
2. Analyse how well the initial design performs in terms of some objective(s), perhaps while satisfying some constraint (best cruise L/D, given some minimum t/c)
3. Perform several/many cycles involving modification to the design and re-analyses of performance so as to find the best possible solution, given some limits on the total computational effort

Inverse

1. Start with some prescribed behaviour we wish to achieve (e.g. a specified pressure distribution)
2. Try to find a configuration that produces the prescribed behaviour.

Which of these approaches is best depends on the particular problem, the available computational methods and resources. Direct methods are probably the most often used in general design, but inverse methods are common in wing design.

See Keane & Nair, *Computational Approaches for Aerospace Design*, Wiley 2005, for a comprehensive review of direct/optimization methods.

Example of subsonic wing aero design trade-off decisions

Airfoils, Planforms, and Twist

These are the three things we can adjust

Mark Drela

From June 2004 issue RC Soaring Digest.

Chris Adams writes:
I am trying to understand the reasons for changing camber as one progresses from the root to the tip on small wings. Can anyone tell me more about the lower camber at the tip and whether this induces effective washout on the wing?

Answer:

There are numerous airfoil/planform/twist conflicts between the following requirements:

Goes against washout and camber variation.

1) Good penetration L/D or good handlaunch height. This wants all spanwise locations to go to zero Cl at the same time as the aircraft's AoA is reduced, and also for each location to remain within its airfoil's drag bucket.

Recall that adding camber moves the drag bucket to higher Cl values, while thinning it reduces the Cl range of the bucket, but lowers Cd _min.

This could say *or* with more stall-resistant airfoils.

2) Tip stall resistance in tight circling maneuvers. This wants smaller Cl towards the tip, preferably with more stall-resistant tip airfoils. This is complicated by the lower tip Reynolds numbers due to taper.

Alternatively, we could use a larger chord and add some washout to keep c*Cl the same.

3) Minimum induced drag. Assuming the span is fixed, this ideally wants the c*Cl distribution to be elliptical at slow thermal speeds. Two extreme possibilities are :

Want low induced drag always but if compromises will be made, most need it at low speed.

i) a constant chord c, with elliptical Cl via washout --- great for 2), awful for 1)
ii) an elliptical planform c, constant Cl via flat wing and constant airfoil --- OK for 1), bad for 2)

Can tune low induced drag at low speed while achieving other things too.

The simplest safe baseline compromise solution is:

a) A constant airfoil, zero twist, and a planform with a considerably wider tip than elliptical. This is nearly ideal for 1), OK for 2), and least favorable for 3).

Example of subsonic wing design trade-off decisions

The following fine-tuning mods can be done:

b) The tip airfoils are thinned, while maintaining their camber and keeping the zero twist. This benefits 2) the most, since it compensates for the lower tip Re and usually gives a larger local Clmax. But this thinning narrows the tip airfoil's bucket somewhat, which may penalize 1). The c*Cl stays the same, and so 3) is unaffected by this modification. Note: The smaller thickness makes the tip airfoils appear more undercambered, even though their camber has not really been changed.

c) The tip chords are narrowed slightly from the "simple" wider-tip solution, and some washout is added. This mod can make the loading nearly elliptical, and benefits 3) the most. On the other hand, 2) is more or less unaffected, but 1) will suffer if the washout is done to excess.

d) The tip chords are narrowed slightly as in c), but the tip chords are decambered the correct amount in lieu of washout. This benefits 1) at some cost to 2). The benefit to 3) is same as with c).

My HLGs use a blend of mods b) and d). My current 2-meter RES project uses a blend of b),c),d) in suitable proportions.

I don't know what's best for F3B.

The best combination of mods b), c), d) depends on which performance consideration is most important. If 1) is most important, like in a windy-day HLG, the simple flat wing solution a) may be best. If you want a calm-day floater for small and weak thermals, then making mods b) and c) is most appropriate. Mod d) is useful in lieu of c) to keep the wing flat for easier construction perhaps.

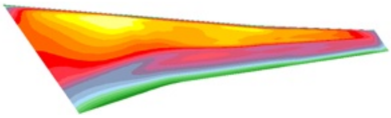
- Mark Drela

Transonic wing design methodology

This is strongly influenced by trying to get straight upper surface pressure contours while maintaining adequate lift and handling.

Mod. no	Modification	Reason
1	Increase the thickness of the forward part of the root section. Decrease the thickness of the rear part of the root section.	To obtain similar chordwise upper-surface velocity distributions due to thickness along the span.
2	Increase the thickness-chord ratio of the root section.	To obtain identical chordwise upper-surface velocity distributions due to thickness along the span.
3	Decrease the positive camber or apply negative camber on the root section.	To adapt the pattern of the chordwise upper-surface velocity distribution due to lift to that of the basic airfoil section.
4	Increase the incidence of the root section.	To obtain identical chordwise upper-surface velocity distributions along the span.
These four modifications together should lead to straight swept isobars over most of the wing upper surface in the design condition.		
5	Modify the wing lower surface along the span (mostly on the inner wing).	To obtain the desired spanwise distribution of the local lift coefficient.
6	Modify the lower surface velocity distribution on the root section regarding front and rear loading.	To minimise the wing pitching moment.
7	Modify the leading-edge region on the outer wing	To obtain satisfactory stalling characteristics

Table 22.1 - Summary of modifications to be performed on a plane tapered swept wing with constant airfoil sections in order to obtain a wing with satisfactory high- and low- speed characteristics



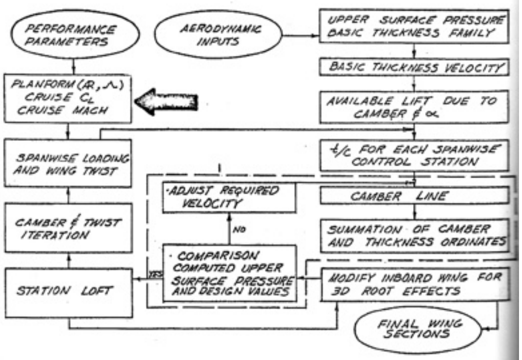


Figure 23.1 - The wing design process. Source: AGARD LS-37, paper no. 6

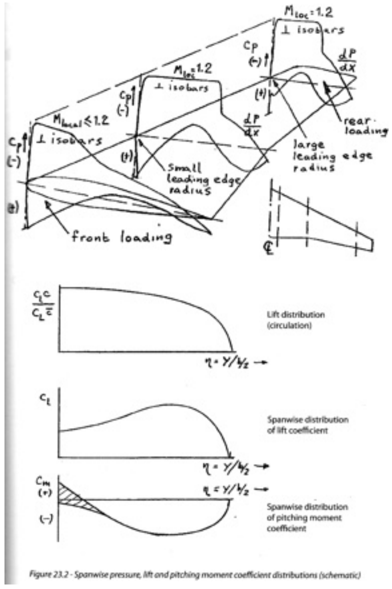


Figure 23.2 - Spanwise pressure, lift and pitching moment coefficient distributions (schematic)

ESDU 97017 (transonic) Wing design process – 1

TABLE 1 WING DESIGN: TWO-DIMENSIONAL DESIGN PROCESS

1. Select Prime Design Point (PDP), *i.e.* $(M_\infty, C_L)_{des,3d}$ values
2. Convert to equivalent, two-dimensional values, $(M_\infty, C_L)_{des,2d}$, by simple sweep rules (Reference 3) and select initial section.
3. Derive/modify upper surface curvature.
4. Derive $z \sim x$, check t/c and z at $x = 1$.

   Satisfactory — Not Satisfactory → change $z(x)$ →

5. Run CFD Code at $(M_\infty, \alpha)_{des,2d}$

   Check C_L

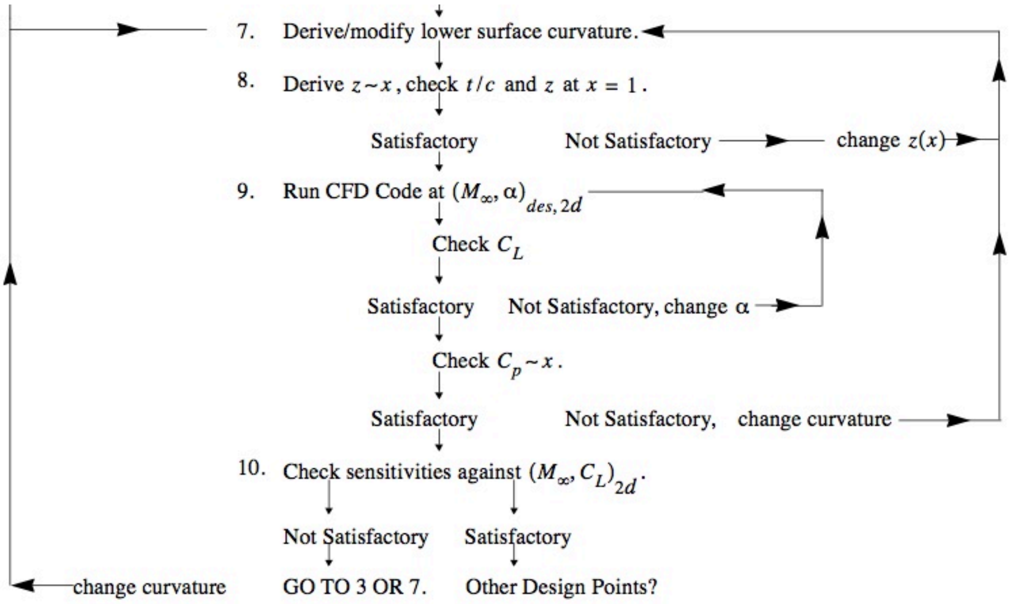
   Satisfactory — Not Satisfactory, change α →

   Check $C_p \sim x$

   Satisfactory — Not Satisfactory, change curvature →

6. Check spar depths.
7. Derive/modify lower surface curvature.

ESDU 97017 (transonic) Wing design process – 2



THIS IS ONLY A STARTING POINT FOR THREE DIMENSIONAL DESIGN, SEE TABLE 2

ESDU 97017 (transonic) Wing design process – 3

TABLE 2 WING DESIGN: THREE-DIMENSIONAL DESIGN PROCESS

PHASE 1

This is characterised by the use of multiple control stations (at each computational grid station) and ignoring the effects of structural deflections under load. For the upper surface it produces the "Ideal Flying Shape" but only a brief exploration of the lower surface is included.

Preliminaries

P1 Fix $(M_\infty, C_L)_{des,3d}$; corresponding to equivalent two-dimensional design or taken from performance requirements.

P2 Decide $t/c \sim \eta$ variation (if any).

P3 Convert two-dimensional section to three dimensions using simple sweep theory (Reference 18 or 14).

P4 Adjust sections for t/c if necessary at grid stations. Extrapolate and interpolate as needed for complete wing definition. Check that the sections are "sensible".

P5 Construct target upper surface pressure distribution (see Section 4). Hence lay out upper surface isobar pattern.

Depending on the CFD methods available, it may be possible to undertake the process from step I1 onwards with a representative body included. If this is so, it may be beneficial to include "wing-alone" runs in the **Preliminaries**. This will provide early estimates of twist distribution, wing flow-field data (to aid in body shaping/non-interfering components) and a means of studying effects of changes in wing planform.

Initial Runs

I1 Run the three-dimensional CFD code at $(M_\infty, C_L)_{des,3d}$. The incidence must be estimated or taken from the two-dimensional exercise *i.e.* $\sin\alpha_{3d} = \sin\alpha_{2d}\cos\Lambda$. (See Reference 18 for choice of Λ.)

I2 Compare the following against the target/design values:

- Overall lift coefficient,
- Pressure distributions at grid stations,
- Upper surface isobars.

Also examine the spanwise lift distribution.

I3 Depending on the above, change the incidence to $(\alpha \pm \Delta\alpha)$ and return to the CFD code at $(M_\infty)_{des,3d}$. If it is possible, make the first estimate of twist distribution at this stage.

I4 Check as at step I2, plus the variation with α of:

- Overall lift coefficient,
- Local (sectional) lift coefficients.

This allows a first approximation of the required values of wing incidence and twist distribution and of body setting angle. It may be that a run at a third incidence is needed to establish a sensible starting point for the Design Iterations.

ESDU 97017 (transonic) Wing design process – 4

Design Iterations

If previous runs have been carried out with a representative body included, a revised body integration and twist could be incorporated prior to proceeding with steps D1 onwards.

D1 Run the CFD code at $(M_\infty)_{des,\,3d}$ and the incidence and twist evaluated at step I4.

D2 Check the values of:

Overall lift coefficient,
Overall pitching moment,
Sectional lift coefficients,
Sectional pressure distributions,
Upper surface isobars.

D3 Based on these results, start redesigning the upper surface using the curvature distributions (see Section 5), continually applying the checks at step D2. The following points should be borne in mind:

The outer wing is affected very much by the inner wing flow, so start designing the inner wing first – moving outwards as success is achieved.

The inner wing *may* impose flow conditions on the outer wing that make it impossible to achieve the desired outer wing pressures/isobars. So, do not wait until the inner wing is perfect before trying to design the outer. Treat the design process as a series of sweeps across the wing span, each sweep consisting of a series of iterations from inner to outer wing.

D4 During step D3, α will have varied in a small range; now check the sensitivity of the upper surface design by systematic variations of α and M_∞.

D5 At this point it is appropriate to examine the fuselage shaping in the wing intersection region and the installation of any tip body (see Sections 10 and 12).

D6 If the lower surface design point is the same as the upper surface design point, then repeat the above process for that surface. The lower surface must accommodate the following:

Required t/c and chordwise position,
Required thickness for rear spar,
Required thickness for front spar/slat.

If the design points for the two surfaces are different, then only a brief exercise is required in this phase to show that the above three conditions are likely to be met. If the design is to utilise variable-camber devices, for example on a combat aircraft, to alleviate the need to compromise between sustained turn rate and cruise/dash conditions, then flap deflection becomes part of the lower-surface design.

D7 If significant changes have been made, return to the upper surface design point to examine any effect there.

ESDU 97017 (transonic) Wing design process – 5

PHASE 2

At this point the number and positions of the control stations for production are finalised and used to define the wing from hereon together with the appropriate rules of interpolation. Structural distortion under load is also included from distributions of *EI* and *GJ* or alternative data; the most important effect is the twist due to bending; the bending itself is less significant. Distortions to camber can also be left to Phase 3 when Finite Element Methods will be used.

This Phase produces the first estimate of the Jig Shape and the Real Flying Shapes at the primary upper and lower surface design points.

D8 Take the final Phase 1 wing and remove the loading associated with the upper surface design point to yield a zero '*g*' shape. Position the required number of control stations to best represent the wing using the interpolation/extrapolation rules. This is the first iteration of the Jig Shape.

D9 Apply the upper surface design point loading to the Jig Shape to produce a Flying Shape and run the CFD code at the design condition.

D10 Compare with the design targets as before (step D2). If changes are required, return to step D8 to make them, *i.e.* an iteration between steps D8, D9 and D10.

D11 Reassess the fuselage shaping and tip body installation.

D12 Apply the loading appropriate to the lower surface design point to the Jig Shape wing to produce the Flying Shape for this condition.

D13 Run the CFD code at the lower surface design point and compare as for the upper surface but looking particularly for any excess wave drag and ensuring that the requirements at step D6 are met. If variable camber was considered at step D6, it must be reconsidered here.

D14 If redesign is needed, modify the Jig Shape, reapply the loading and run the CFD code on the new Flying Shape. This iteration is repeated as necessary.

D15 Now examine the lower fuselage shape, *i.e.* that part below the wing plane.

D16 Assess the impact of pylons and stores/engines by comparing pylon-on results with pylon-off (see Section 11).

D17 Check for any effects on the upper surface by running the upper surface Flying Shape at the relevant design point.

PHASE 3

It is assumed that by this stage, the structural definition of the wing is adequate for the use of Finite Element Methods to calculate the wing distortion under load. This distortion will include changes to both twist and camber.

D18 Starting with the Phase 2 wing, repeat steps D8 to D17 using Finite Element Methods to define the wing distortion. While pitching-moment effects will have been considered at the concept layout stage (and at step D2), it is necessary to check these effects at the end of the wing design and adjust the configuration and/or return to step D8.

Final Checks

Depending on the availability of CFD methods, final checks might be undertaken with fully-integrated powered nacelles with any significant nozzle effects represented.

F1 Design the wing pylons with suitable allowance for the engine/store within the geometric constraints. This assumes that the pylons are thin and the checks at step D16 did not show a dominant effect; if this is not the case, pylon design must start at step D16.

F2 All the possible combinations of tip missile, launcher and adapter must be checked, for combat aircraft.

F3 The wing must now be exercised throughout the entire ranges of C_L, M_∞ and Reynolds number (concentrating on the important performance points and areas of difficulty including high lift at low speeds). Any problems will require a return to step D18.

Choice of design variables — 1

The general principle is to choose as few design variables as possible, partly because the computational design process (direct or inverse) is iterative. If there are N design variables then the design space is N-dimensional.

Example: defining an airfoil surface

We need a large number of grid points to accurately compute the flow field. So in the design setting one choice is to use the airfoil locations at all the mesh points as variables.

However, this is a bad idea! If each grid point location is taken as an independent design variable then there will be a very large number of variables.

It is generally best to decouple the specification of the design from its physical realization, i.e. we look for an efficient way to parameterize the surface shape.

Some possibilities:

1. Use some analytic expression, or several expressions defined over intervals, to provide a continuous analytic description of the surface.
2. Define a smaller set of airfoil coordinates and interpolate directly between these.
3. Take an existing basic airfoil shape and define a series of shape modification functions (e.g. the Hicks-Henne functions) that modify the contour.
4. Use a *B*-spline control-polygon to define the surface.

Options 3 & 4 are the most commonly used.

Choice of design variables — 2

Hicks-Henne shape modification functions

1. Say we start with a NACA0012 airfoil. Then a completely different airfoil can be generated by adding a set of 'localized' functions to the baseline shape.

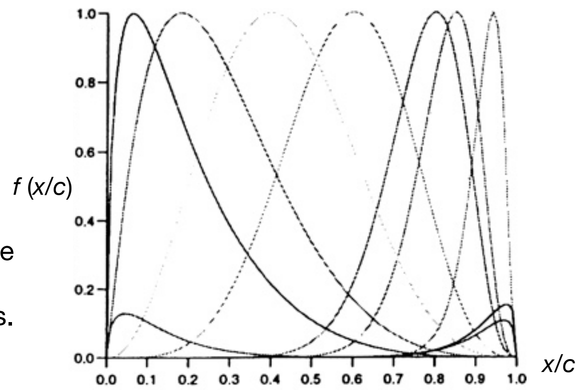
$$y_{t,\text{mod}} = y_{t,\text{base}} + \sum_{i=1}^{20} A_i f_i(x) \quad \text{Top surface}$$

$$y_{b,\text{mod}} = y_{b,\text{base}} + \sum_{i=11}^{20} A_i f_i(x) \quad \text{Bottom surface}$$

The new airfoil is defined in terms of the coefficients A_i as well as shape functions $f_i(x)$.

2. A commonly used family of shape functions are from Hicks & Henne (1987) *J Aircraft* **15**(7). These are continuous, and continuously differentiable.

$$f = \left[\sin\left(\pi x^{\log 5/\log t_1}\right)\right]^{t_2}$$

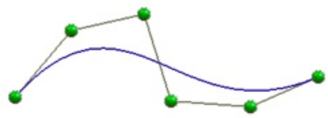


3. Parameters t_1, t_2 control the characteristic shape of the curves, altering their localisation and spread. These would be fixed along with the number of shape functions.

4. The design variables are the coefficients A_i.

5. These functions are useful for making local modifications to airfoils, but for direct design, *B*-spline representations are generally preferred owing to their greater flexibility.

Choice of design variables — 3



<u>*B*-spline polynomials (a.k.a. NURBS)</u>

1. Let $p(t)$ be the position vector along a curve, specified by the parameter t.

2. A *B*-spline is defined by

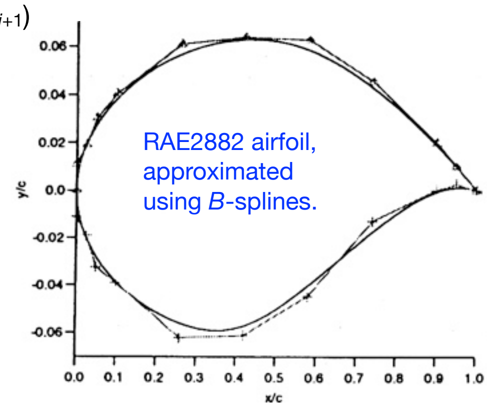
$$p(t) = \sum_{i=1}^{N+1} b_i N_i^k(t), \quad t_{\min} \leq t \leq t_{\max}, \quad 2 \leq k \leq N+1$$

where the b_i are position vectors of the N+1 defining polygon vertices and the N_i^k are normalized *B*-spline basis functions.

3. The *i*th normalized *B*-spline basis function of order k (polynomial degree k-1) is defined by the recursion formulae

$$N_i^1 = \begin{cases} 1 & \text{if } x_i \leq t \leq x_{i+1} \\ 0 & \text{otherwise} \end{cases} \qquad \text{and} \qquad N_i^k(t) = \frac{(t - x_i)N_i^{k-1}(t)}{x_{i+k-1} - x_i} + \frac{(x_{i+k} - t)N_{i+1}^{k-1}(t)}{x_{i+k} - x_{i+1}}$$

where the x_i are the elements of an ordered 1D knot vector ($x_i \leq x_{i+1}$)



4. Each side of an airfoil is then defined by assigning a starting base polygon and allowing each vertex to move within a defined range. One would check that the airfoil surfaces do not cross and perhaps that a minimum thickness was achieved at spar locations.

5. The design variables are given as the array of locations of the vertex points $X(b_i)$.

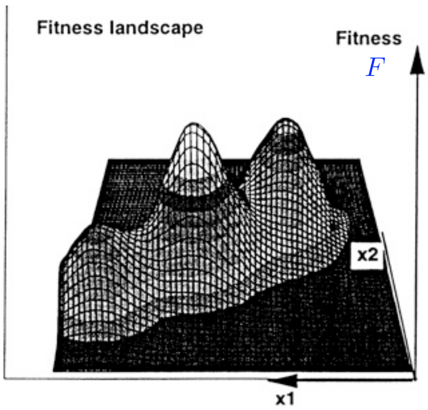
Direct design — 1

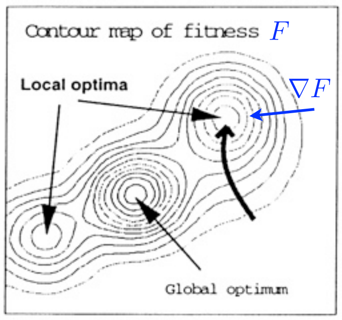
1. A number of methods exist for direct design optimization. We will briefly consider <u>unconstrained gradient-based optimization</u>.

2. Optimization consists of minimizing/maximizing the value of the *objective function* which is a scalar function of a vector of design variables.

3. In unconstrained optimization the objective function needs to be chosen with care. For example, designing an airfoil to minimize C_d produces an airfoil of zero size. A better choice might be to maximize L/D.

4. A contour plot of the objective function against the design variables is often called a <u>fitness landscape</u>. This might be difficult to visualize if there are more than two design variables.





∇F is a vector that points in the direction of most rapid increase in F at any location in parameter space.

5. Optimization methods seek maxima/minima in the fitness landscape, hopefully global ones.

Direct design — 2

Gradient search

1. If the form or shape of the design depends continuously on the design variables we can use calculus-based methods to follow the gradient to a turning point. Typically the gradient cannot be found analytically but we can approximate it by perturbing each variable in turn.

2. Say the objective function (e.g. *L/D*) is $F = F(\boldsymbol{u}, \boldsymbol{x})$ where $\boldsymbol{u}$ represents the flow field and $\boldsymbol{x}$ represents the geometry. The geometry $\boldsymbol{x}$ (and hence $\boldsymbol{u}$) depends on our *N* design variables, which could be our HH function coefficients/weights A_i or the *B*-spline knot locations b_i. Say for A_i, we want to maximize *F* (i.e. mimimize -*F)*:

    i. Perturb each A_i separately;

    ii. Determine the (vector) gradient $\partial F/\partial A_i$ — <u>the negative of this is the direction of steepest descent in value of *F*</u>;

    iii. Perturb all the A_i in this direction;

    iv. Recompute -*F*;

    v. If not a minimum, return to step i.

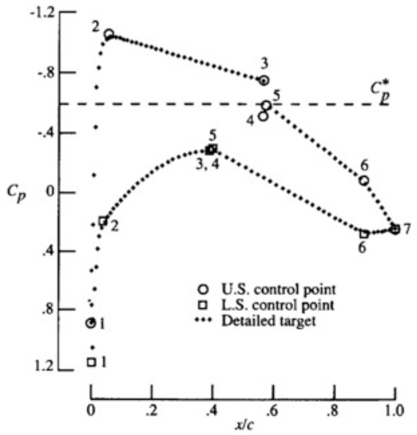
3. For the case shown, this would only get us to a *local* minimum, whereas we'd like the global minimum. A work-around might be to use a randomized set of initial design variables, but that's inefficient. We need a large number of function evaluations and that could be expensive/lengthly.

4. Newer techniques (*adjoint methods*) enable gradients to be found (with some restrictions) for a single function evaluation.

5. All the methods that rely on a continuous variation in the objective function with variations in design variables break down if that is untrue. Then we face brute-force combinatorial optimisation.

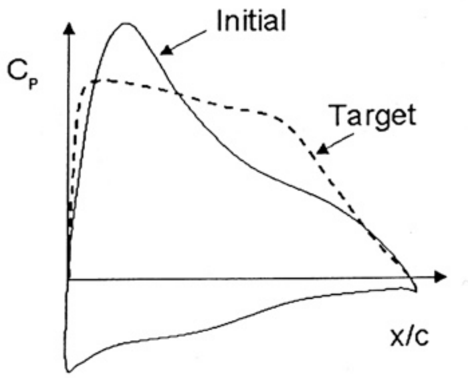
Inverse design — 1

Inverse design by streamline curvature, method of Smith & Campbell, e.g. NASA TP-3045 and 3260 (1990, 1992).



For inverse design we indirectly prescribe the flowfield, usually by nominating the desired C_p distribution (NB using a small set of control points) and attempt to design/change the geometry to achieve this.

(Note that there is some art in devising and parameterizing an appropriate distribution.)



The method is based on relating changes in geometry to changes in pressure, based on physics.

We will give an outline of the method based in inviscid theory, which is not exact but typically sufficient for high-*Re* design. Related methods have been developed for RANS-based solution.

The relationship between geometry and flow depends on whether the flow is compressible/high speed or incompressible/low speed.

Inverse design — 2

For <u>compressible</u> flow, for small changes in airfoil slope *y'*, supersonic thin airfoil theory gives

$$C_p = 2y'/\sqrt{M_\infty^2 - 1} \equiv y'/\lambda \qquad \text{so that} \qquad \Delta y' = \lambda \Delta C_p$$

then

$$\Delta y'' = \lambda \frac{\mathrm{d}\Delta C_p}{\mathrm{d}x} \tag{1}$$

For <u>incompressible</u> flow, the situation is more complicated. Smith & Campbell combine theory and some empiricism to obtain

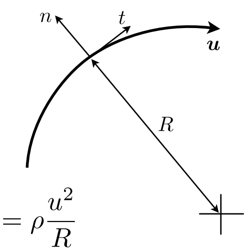
$$\Delta y'' = \Delta C_p \omega A \left[1 + C_o^2\right]^B \left[1 + y'^2\right]^{3/2} \tag{2}$$

where ω is a relaxation factor (typically < 1), *A* = +/–1 on upper/lower surface, *B* is an empirically chosen constant and $C_o = y''/[1+y'^2]^{3/2}$ is the airfoil surface curvature.

Exponent *B* would be ½ according to incompressible flow theory, but in the generalization *B* is taken in the range (0, ½).

Equation (2) can be used up to Mach numbers slightly greater than unity, but beyond this it is necessary to use (1). We will look at the derivation of (2) below.

Inverse design — 3



<u>Derivation of incompressible flow relationship between slope & C_p.</u>

Consider a flow with speed *u* along a curved streamline, radius *R*.

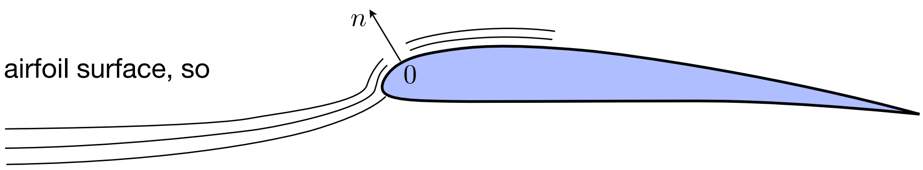
Pressure gradient normal to the streamline for inviscid incompressible flow: $\dfrac{\mathrm{d}p}{\mathrm{d}n} = \rho \dfrac{u^2}{R}$

From Bernoulli's equation $p + \frac{1}{2}\rho u^2 = \text{const}$ obtain $\dfrac{\mathrm{d}p}{\mathrm{d}n} = -\frac{1}{2}\rho \dfrac{\mathrm{d}u^2}{\mathrm{d}n} = -\rho u \dfrac{\mathrm{d}u}{\mathrm{d}n}$

Combine: $\rho u \dfrac{\mathrm{d}u}{\mathrm{d}n} = -\rho \dfrac{u^2}{R} \;\Rightarrow\; \dfrac{\mathrm{d}u}{u} = -\dfrac{\mathrm{d}n}{R}$

In general the radius of curvature on different local streamlines will be a function of coordinate *n*, or alternatively the curvature $C(n) = \dfrac{1}{R}$ hence $\dfrac{\mathrm{d}u}{u} = -C(n)\,\mathrm{d}n$

Let 0 denote a point on the airfoil surface, so



$$\int_\infty^0 \frac{\mathrm{d}u}{u} = -\int_\infty^0 C(n)\,\mathrm{d}n$$

Now it is assumed <u>empirically</u> that $C(n) = C_0 \mathrm{e}^{-kn}$ with the surface (*n* = 0) curvature $C = C_0$ and as *n*→∞, *C*→0, which is physically correct. Now integrate:

$$\ln\left(\frac{u_0}{U_\infty}\right) = \frac{C_0}{k}\mathrm{e}^{-kn} \quad \text{and} \quad C_0 = k\ln(u_0/U_\infty) \qquad \text{For small variations,} \quad \Delta C_0 = k\Delta u_0/U_\infty$$

Inverse design — 4

Also (quasi-empirical) assume $k \propto C_0$, leading to $\Delta C_0 = A_1 C_0 \, \Delta u_0 / U_\infty$ linking Δcurvature to Δvelocity.

Now $C_p = 1 - (u/U_\infty)^2$ so that for small variations $\Delta C_p = -2\Delta u / U_\infty$ hence $\Delta C = AC \, \Delta C_p$ (where $A = -0.5A_1$) so that changes in C_p are now related to changes in surface curvature.

However as $C \to 0$, a small change in radius will lead to a large change in C_p, so the above relationship is modified empirically to produce

$$\Delta C = \Delta C_p \omega A [1 + C^2]^B$$

where ω is a relaxation (damping) factor, $0 < B < 0.5$ (typically 0.2), $A = +/-1$ for upper/lower surface.

The curvature $C = y'' / \left(1 + y'^2\right)^{3/2}$ so assuming the curvature changes faster than the slope,

$$\Delta C = \Delta y'' / \left[1 + y'^2\right]^{3/2}$$

Finally

$$\Delta y'' = \Delta C_p \omega A \left[1 + C^2\right]^B \left[1 + y'^2\right]^{3/2}$$

Inverse design — 5

Applying the method (subsonic)

$$\Delta y'' = \Delta C_p \omega A \left[1 + C^2\right]^B \left[1 + y'^2\right]^{3/2}$$

1. We need a target pressure distribution. Also we have to be able to compute a new pressure distribution given the shape of the airfoil.
2. We use finite differences to compute slope and curvature terms in the above equation. Starting with say the upper surface of the airfoil we start at the leading edge and divide the surface into a large number of segments, the locations of the surface denoted by (x_i, y_i).
3. For a general location x_i we can compute y_{i+1} and we store the difference between it and its old location as Δy_{i+1}. We compute all the Δy_is to the TE.
4. Sum up all the vertical displacements to give the new location of the entire airfoil surface.
5. If the TE no longer closes, we simply rotate the new surface about the LE to force closure.

Example (with A=1, ω=1 and B=0.5)

1. Let the locations of three points on the airfoil surface be labelled $P, Q, R,$ corresponding to x_{i-1}, x_i, x_{i+1} be $P = (0.1,0.1)$, $Q = (0.14,0.12)$ and $R = (0.18,0.13)$. $\Delta x = \text{const} = 0.04$.
2. Let the difference $\Delta C_p = C_{p(\text{target})} - C_{p(\text{actual})} = 0.2$.
3. Recall central difference estimates

$$y'' = \frac{\mathrm{d}^2 y}{\mathrm{d}x^2} \approx \frac{y_{i+1} - 2y_i + y_{i-1}}{\Delta x^2}, \qquad y' = \frac{\mathrm{d}y}{\mathrm{d}x} \approx \frac{y_{i+1} - y_{i-1}}{2\Delta x}$$

Inverse design — 6

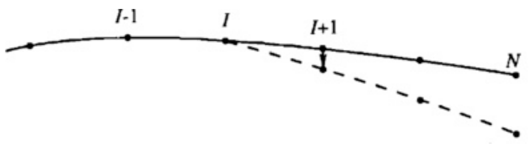
$$y'' \approx \frac{0.13 - 2 \times 0.12 + 0.1}{0.04^2} = -6.25, \qquad y' \approx \frac{0.13 - 0.1}{2 \times 0.04} = 0.375$$

$$C = \frac{y''}{(1 + y'^2)^{3/2}} \approx \frac{-6.25}{(1 + 0.375^2)^{3/2}} = -5.480$$

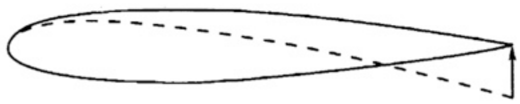
$$\Delta y'' = \Delta C_p \omega A \left[1 + C^2\right]^B \left[1 + y'^2\right]^{3/2} \approx 0.2 \times 1 \times 1 \times \left[1 + 5.480^2\right]^{1/2} \left[1 + 0.475^2\right]^{3/2} = 1.512$$

$$\Delta y'' \approx \frac{y_{i+1}^{\text{new}} - 2y_i + y_{i-1}}{\Delta x^2} - y''$$

$$y_{i+1}^{\text{new}} = \Delta y'' \Delta x^2 + 2y_i - y_{i-1} + y'' \Delta x^2 = 1.512 \times 0.04^2 + 2 \times 0.12 - 0.1 - 6.25 \times 0.04^2 = 0.132$$



(a) Shear surface segment to change curvature at a point.

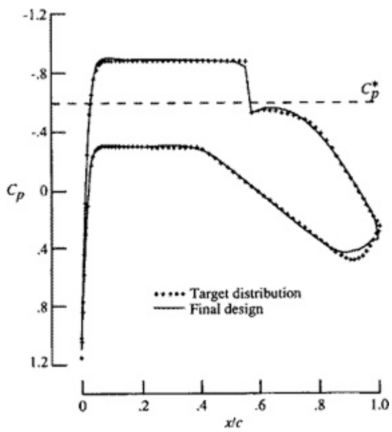


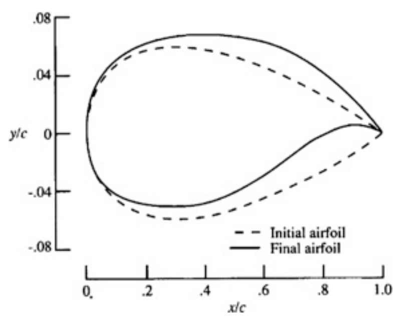
(b) Rotate entire surface to recover original trailing-edge points.

Inverse design — 7

Example results

Inviscid/Transonic/2D





RANS/Transonic/2D

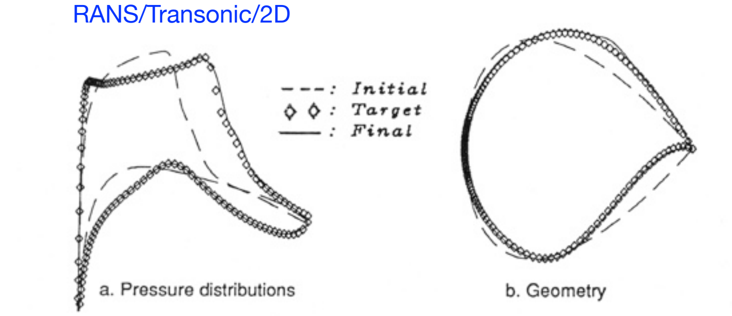
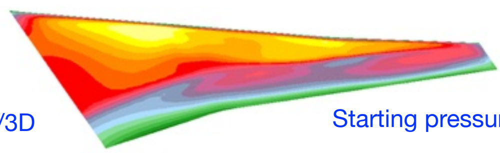
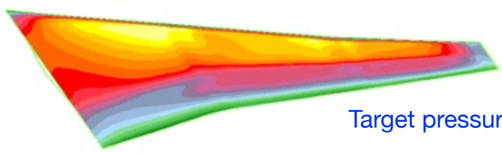


Fig. 4. Airfoil design–RAE2822 at M=0.75, α=2.81°, Re=6.2 million/C

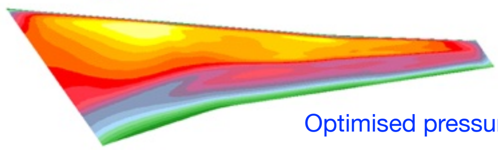
RANS/Transonic/3D



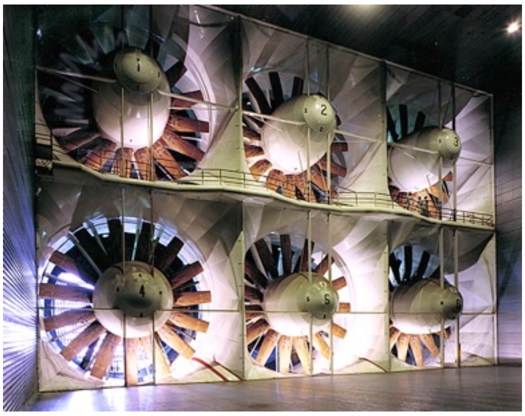
Starting pressures



Target pressures



Optimised pressures



Wind Tunnel Testing

Types of wind tunnel — 1

Steady flow

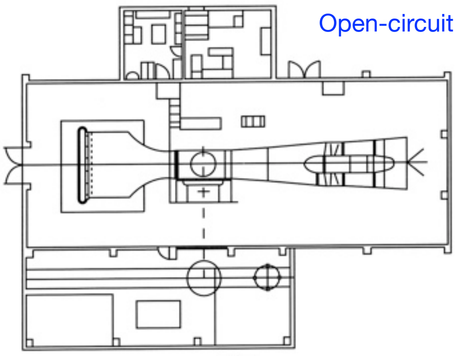
1. Most subsonic and transonic tunnels are of this type.
2. May be open or (more typically for larger facilities) closed circuit.
3. Specialized types:
    a. Boundary layer (BL research, or model atmospheric BL/wind engineering)
    b. Car test (moving floor, dynamometer)
    c. Spin/free flight
    d. Cryogenic
    e. Anechoic
    f. V/STOL
    g. Propulsion

Transient flow

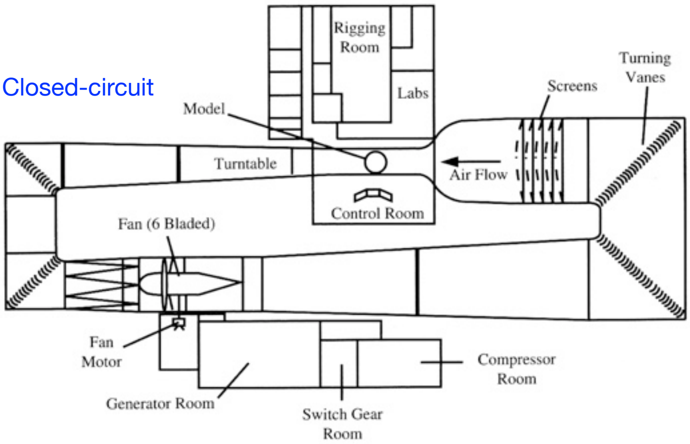
1. Most supersonic/hypersonic tunnels are of this type. Variants:
    a. Blowdown (from high-pressure reservoir) or indraft (to low-pressure reservoir)
    b. Shock tube (burst a diaphragm)
    c. Gun (shoot a high-speed/lightweight piston)

Types of wind tunnel — 2

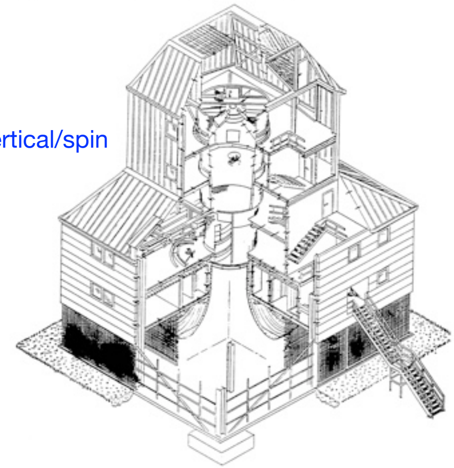
Subsonic



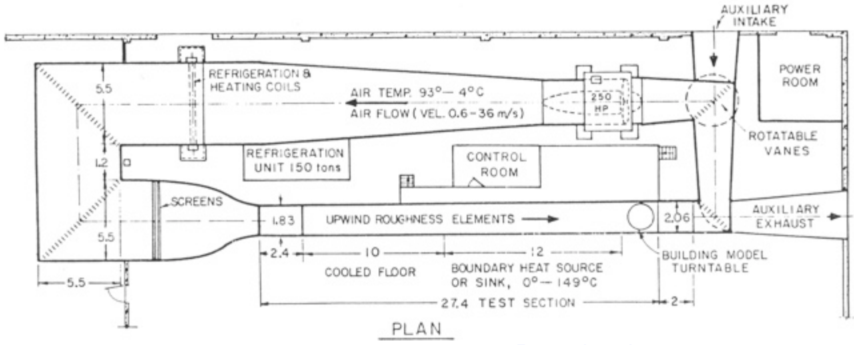
Open-circuit



Closed-circuit



Vertical/spin

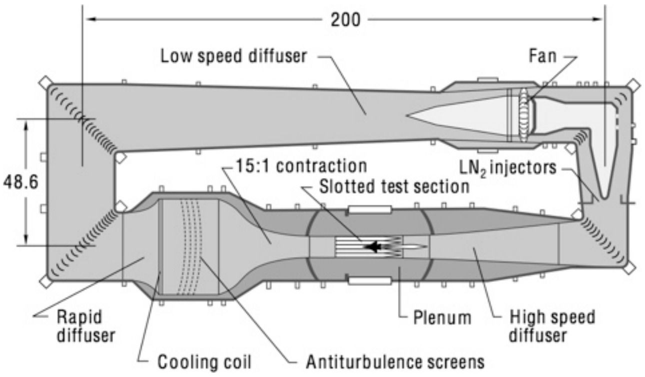


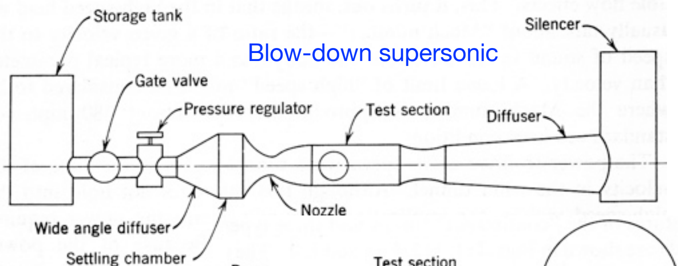
Boundary layer type

Types of wind tunnel — 3

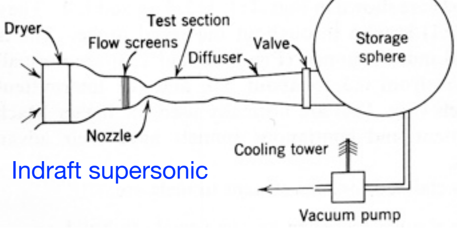
Super/hypersonic

Closed-circuit transonic

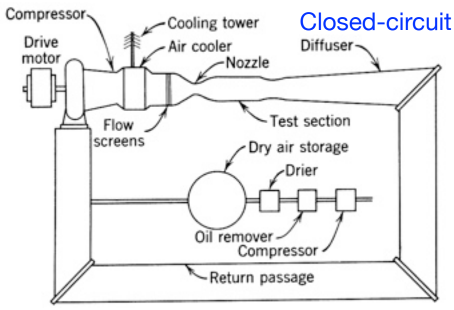




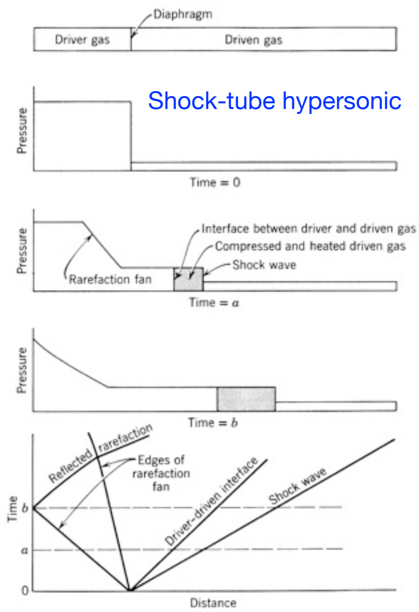
Blow-down supersonic



Indraft supersonic

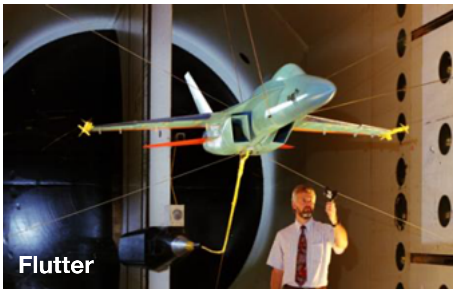


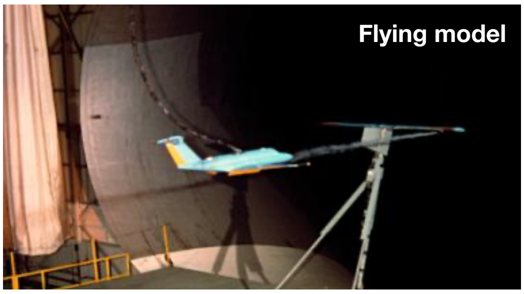
Closed-circuit supersonic

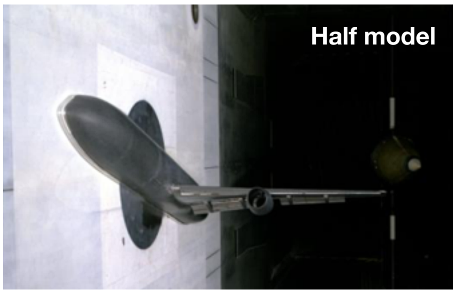


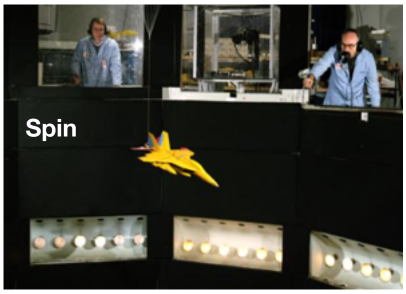
Shock-tube hypersonic

Types of model/support/tunnel

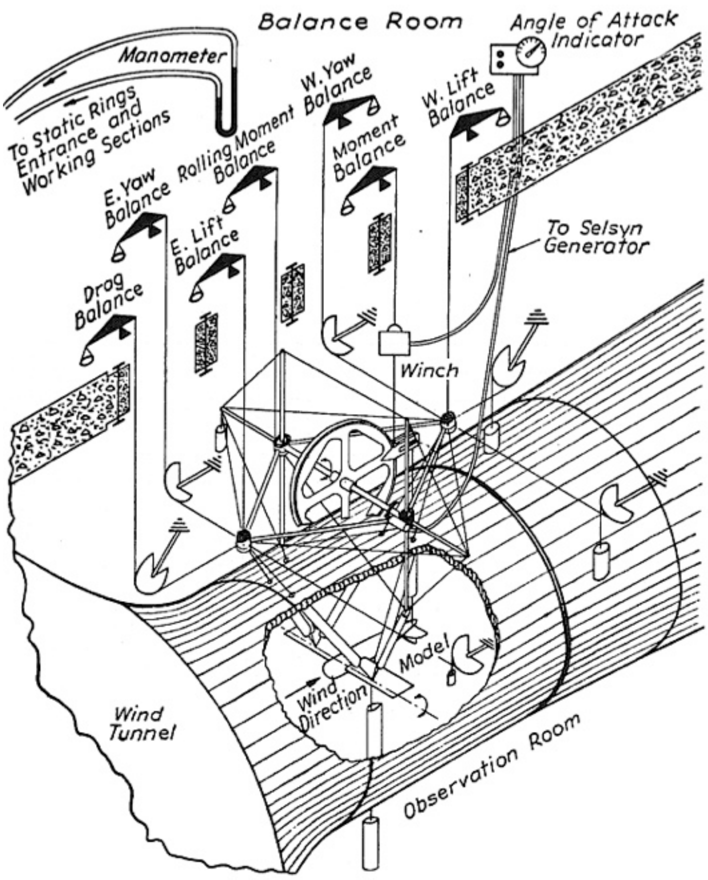








Force balances

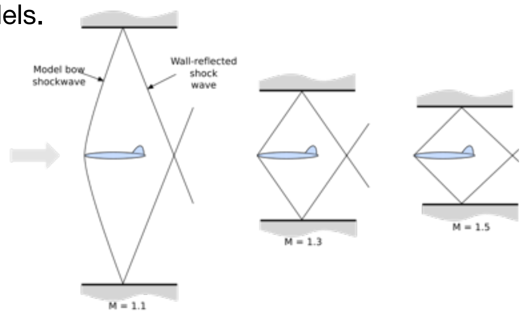


Technical challenges — 1

1. Scale effects: where Reynolds and/or Mach numbers don't match full scale.
2. Interference effects: where the presence of tunnel walls and model supports alters the flow from that about an unsupported model in free flight.
3. Tunnel flow disturbances: the fans that drive the tunnel, guide vanes to turn the flow, and tunnel walls may each introduce disturbances/turbulence to the flow which may affect transition behaviour of the model BL. (A 'clean' aerodynamic test tunnel should have a test-section turbulence intensity — standard deviation/mean — less than ~0.2%. This is difficult to achieve: no Monash flow facility is this clean: best values are of order 1%.)

Interference effects

1. Model and wake blockage effects (speed-up/distortion of flow, variation of pressure gradient).
2. Lift correction effects (image vortices).
3. Wall boundary layers:
    a. As they grow they reduce the effective cross-sectional area of the working section and produce streamwise pressure gradients;
    b. They may interfere particularly severely with half-span models.
4. At transonic/supersonic speeds:
    a. Shock wave reflections;
    b. High sensitivity of flow to model blockage.



Technical challenges — 2

Dealing with interference effects

1. Apply theoretical/empirical corrections.
2. Reduce model size but beware of Reynolds number effects.
3. Use tunnel with ventilated (slotted or perforated) walls.
4. Reduce the effects of shock-wave reflection.
5. Use adaptive walls — walls are aligned with far-field flow streamlines.

Dealing with flow disturbances

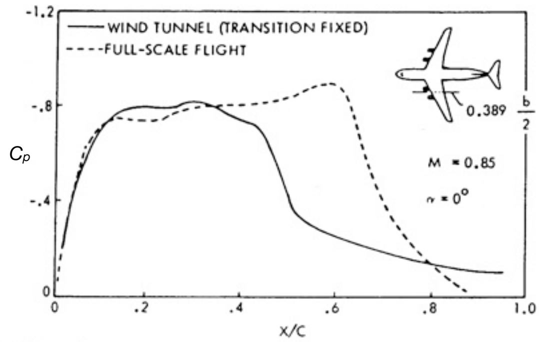
1. Use large-ratio contractions upstream of working section (subsonic only).
2. Use screens upstream of working section (again, subsonic).
3. Use flow straighteners/vanes.
4. Attention to tunnel and fan aerodynamic design to avoid flow separation.
5. Attention to tunnel aeroacoustic design to avoid duct resonances.

Semi-span models

1. Symmetry provides a 2x gain in effective tunnel dimensions.
2. Reduces cost of model and support.
3. Convenience for connecting pressure tappings and air supply (engine simulation).
4. Can only deal with symmetric flight conditions.
5. Difficulties in eliminating sidewall BL interference.

Technical challenges — 3

Wind tunnel data are significantly different from flight data if *Re* is not adequately matched.



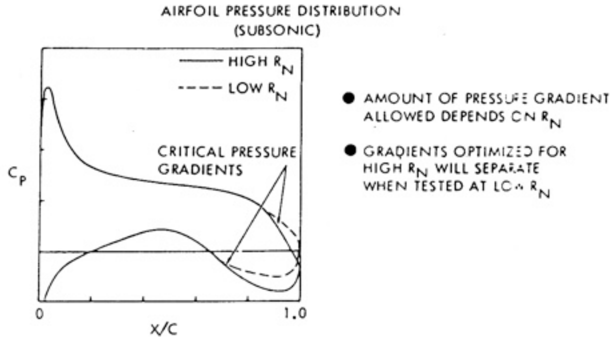
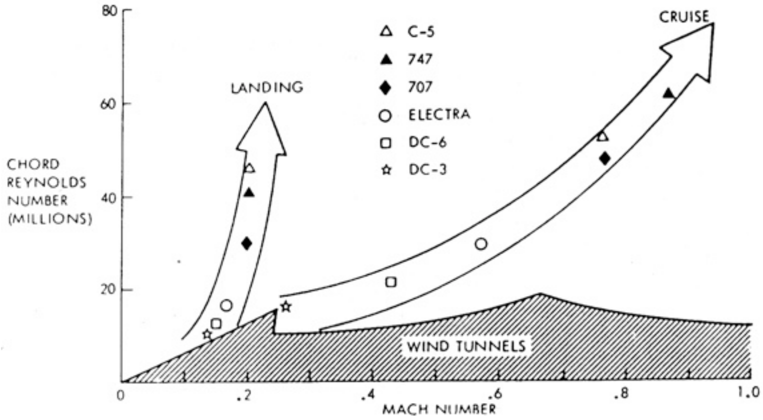


Fig. 24 Lockheed C-141 pressure distributions.

With the trend to larger transport aircraft, flight Reynolds numbers have increased over time.



High-Reynolds number facilities — 1

Facility	Reynolds No. (millions)[a]	Productivity (polars / hour)[b]	User Cost ($ / polar)	Age (years)[c]
Subsonic Tunnels				
LSWT (Proposed)	***20.4***	***5.00***	***752[d]***	***N/A***
NASA-ARC 40x80-ft	16.6	0.34	5965	51
NASA-ARC 80x120-ft	10.8	0.34	5865	11.5
DRA 5-m (Britain)	7.7	1.50	3000	16
NASA-ARC 12-ft[1]	7.6- 9.5[e]	2.85	1300	Rebuilt[f]
ONERA F-1 (France)	7.5	1.70	3000	17
Lockheed 16x20-ft	3.9	3.50	225	29
DNW (Netherlands)	3.6	4.00	1000	14
NASA-LaRC 14x22-ft	3.2	0.60	1050	24
Lockheed 8x12-ft	2.5	4.00	250	47
Transonic Tunnels				
NASA-LaRC NTF, Nitrogen	119.0	0.36	14300	11
ETW (Europe)	50.0	1.50	5600	0
TSWT (Proposed)	***28.1***	***8.00***	***644[d]***	***N/A***
NASA-LaRC TDT	16.0	0.20	5000	36
NASA-ARC 11-ft	10.3	4.00	1172	39
Calspan 8-ft	10.0	4.00	825	48
AEDC 16T	9.6	4.50	1170	38
Russian T-128	9.2[g]	1.00	2750	10
Rockwell 7-ft	7.0	2.00	1500	36
NASA-LaRC NTF, Air	6.0	2.00	1537	11
Boeing TWT	3.9	4.50	725	41

[1]The Ames 12-ft tunnel has been demolished and reconstructed. The new tunnel will have a maximum pressure capability of 6 atmospheres which will yield a Reynolds number of 12 million /ft at Mach 0.3. Additionally, the new control system and test section design will assist in attaining a 4 polar / occupancy hour productivity goal for the new facility.

High-*Re* facilities tend to be large, costly to run, and have poor productivity.

High-Reynolds number facilities — 2

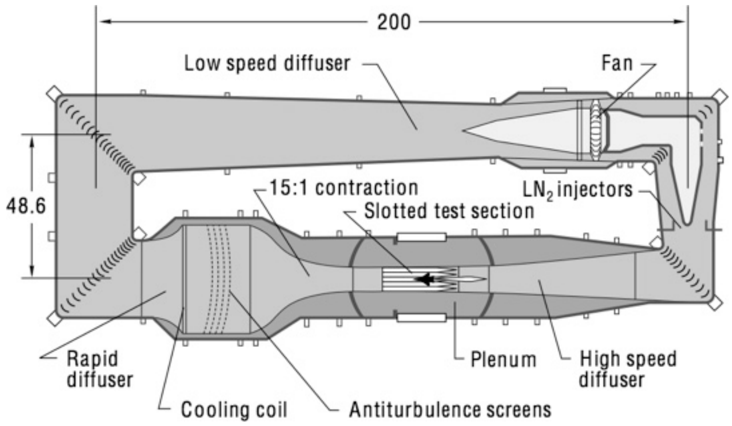
The most capable transonic tunnels are capable of testing aircraft configurations at Reynolds and Mach numbers equivalent (or nearly equivalent) to full-scale flight conditions.

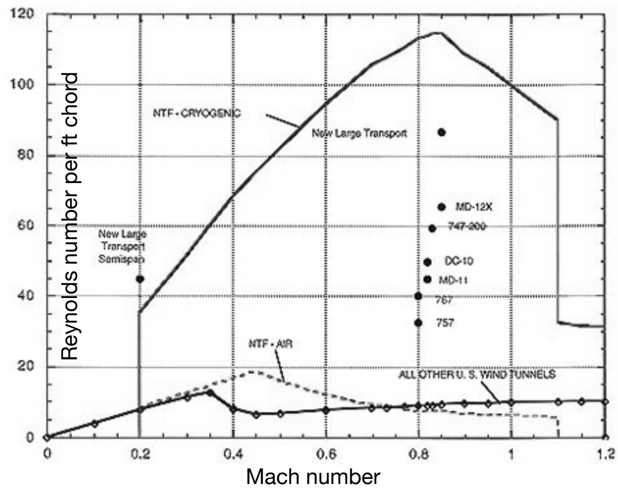
$$Re \propto 1/\mu \propto \rho/\mu$$

Cryogenic cooling by injection of liquid nitrogen into the tunnel lowers the temperature to 140K — cooling a gas both reduces its viscosity and increases density. The working fluid is nitrogen (or air).

In addition the tunnels are pressurised up to 9atm — pressurising a gas increases its density. These tunnels usually need an external pressure vessel.

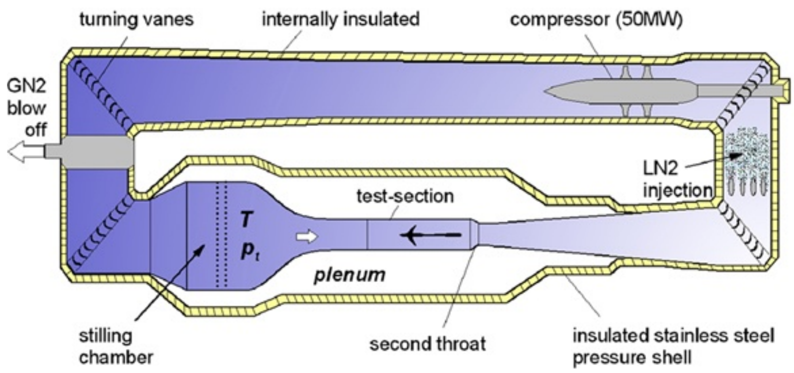
NASA Langley NTF

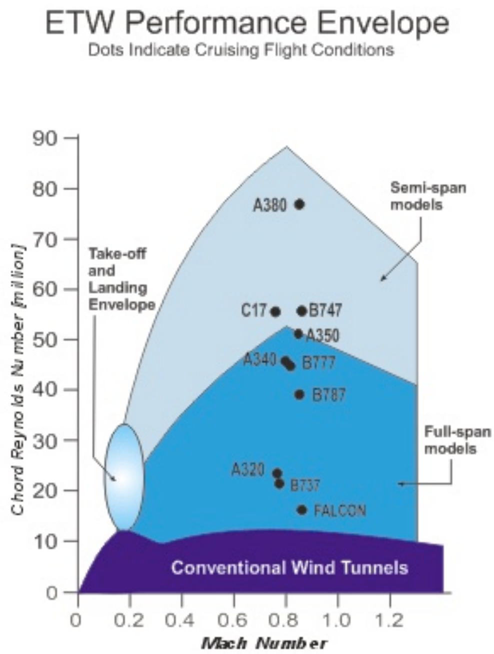




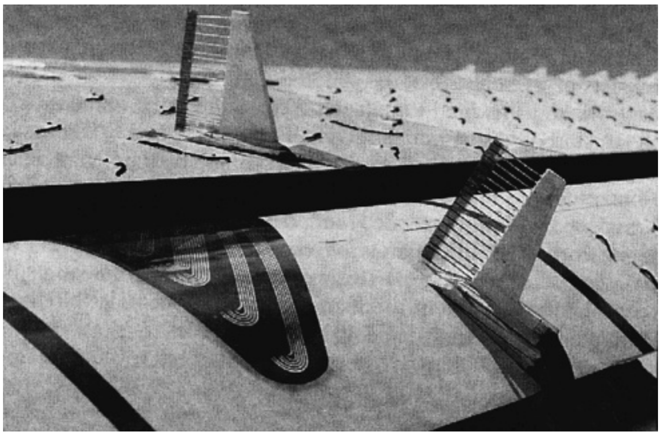
High-Reynolds number facilities — 3

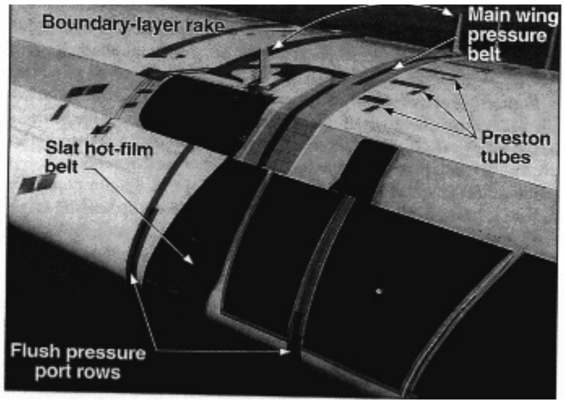
European Transonic Windtunnel (ETW), Cologne

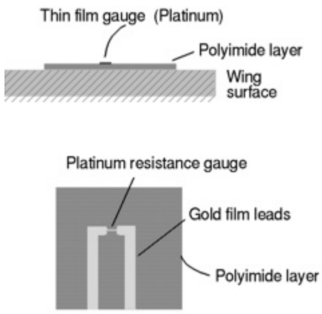


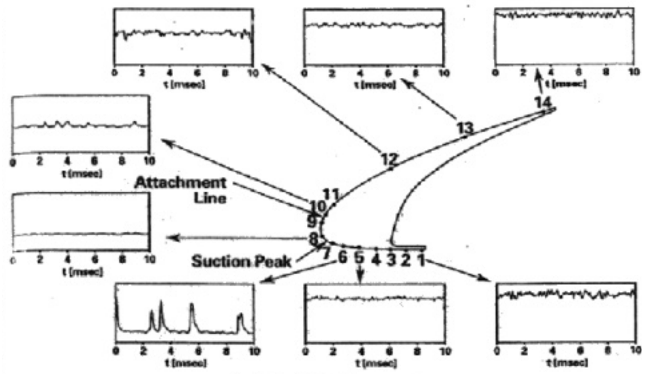


Flight testing — 1









(a) Hot-film traces.