Deleting the duct: a layout change that closes Mach 1.1. Brief v4 for Dr. Zaidi

Aayushya Patel 7 November 2026 (all numbers regenerate from scripts in the project repo)

Short answer: the same two engines now close at Mach 1.10 instead of 1.04, and the change is geometric, not optimistic. Brief v3 sized the body at Ø191 mm for one reason: the front engine's exhaust had to be ducted through the body past the rear engine. That duct also created every inlet risk an outside reviewer raised with me. Removing it, by exhausting the front engine sideways at its own station through a pair of opposed nozzles, shrinks the body to Ø167 mm while keeping both engines. Margin at Mach 1.1 moves from -9.4% to $+0.6\%$.

The margin is thin and I am not claiming the aircraft flies. What I am claiming is the comparison: **at equal mass, equal engines and equal length, deleting the duct is worth about 10 points of margin and 24 mm of diameter**, and that follows from geometry rather than from a modelling choice. Wave drag scales with cross-sectional area, so a second engine placed in line costs length, which is nearly free, and the duct was what was costing area.

Two K-450G5s + afterburner, sea level, Mach 1.1	v3: ducted tandem	**v4: split exhaust**
Body diameter	Ø191 mm	**Ø167 mm**
Wave drag D/q	115 cm^2	**94 cm^2**
Static thrust per frontal area	2.36 N/cm^2	**2.78 N/cm^2**
Mass	21.2 kg	21.2 kg
Installed thrust / drag	1,530 / 1,689 N	1,504 / 1,494 N
Thrust margin at Mach 1.1	-9.4%	$+0.6\%$
Chance that margin > 0 (10,000 runs)	4.3%	**11%**
Highest Mach where thrust $\geq$ drag	1.04	**1.10**

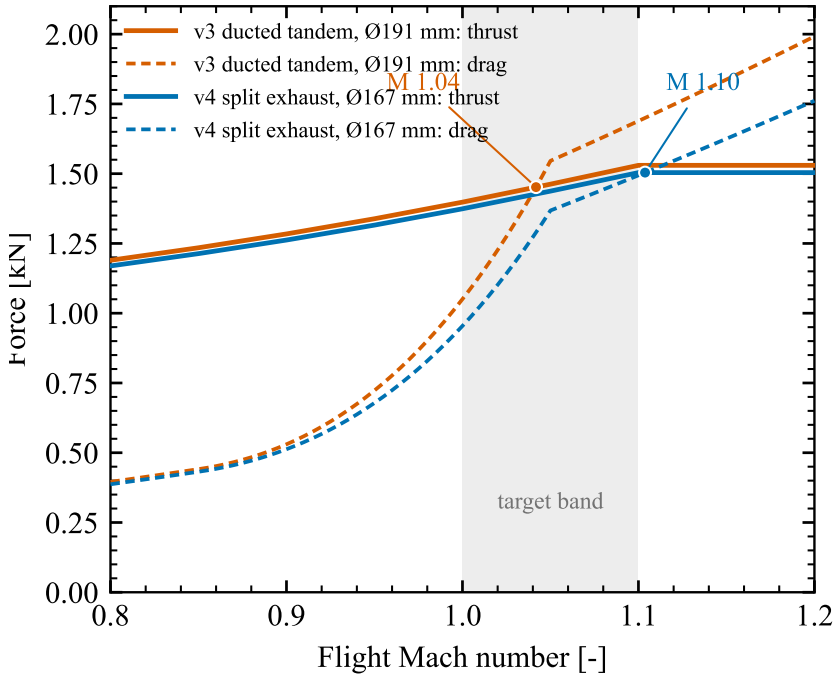


Figure 1: Installed thrust (solid) and drag (dashed) for the v3 ducted tandem and the v4 split-exhaust layout, sea-level standard day, afterburner at 1,800 K. Markers are the thrust-equals-drag crossings. Same two engines and same mass in both cases: the entire difference is that the v4 body is Ø167 mm instead of Ø191 mm, because the front-engine exhaust duct is gone. Thrust from pyCycle 4.4.0 (NASA). Drag from slender-body wave drag on the actual area distribution plus Raymer skin friction. Both curves are produced by the same functions that generate this brief's tables, and the script asserts agreement with them. Script: `proposal/brief04/fig_v4.py`.

Limit: Everything carried forward from v3 keeps its v3 limits, including the unvalidated thrust lapse with flight speed ($\pm 5\%$) and the drag model's 20% low rise between Mach 0.95 and 1.0. The new items in this brief have their own limits, stated where each result appears. $+0.6\%$ has no engineering room in it.

1. What changed, and why it is not a free lunch

The front engine exhausts through two opposed nozzles, one dorsal and one ventral, at its own axial station (about 0.95 m from the nose), each canted 15° from the flight axis. The rear engine breathes through short side inlets at its own compressor face instead of a 1.2 m duct from the nose.

Four penalties are charged against the new layout in the model, and none of them is waived:

Penalty	Value	Basis
Cant loss on front-engine thrust	$\cos 15^\circ$ (2.3% at 15°)	Geometry. Thrust is charged as $\frac{1}{2}(1+\cos\theta)$ across the two engines
Jet interference drag	$+0.0009\ C_D$ on wing area	NASA TM-X-3424, measured. See Section 3
Base area	One nozzle, not two	The front jet now leaves through the side
Captured streamtube	Both engines' flow	Unchanged from v3

Cant angle is a first-order variable, not a detail. Margin crosses zero near 17°:

Cant	0°	5°	10°	**15°**	20°	25°	30°
Thrust = drag at Mach	1.114	1.113	1.109	**1.104**	1.092	1.071	1.050
Margin at Mach 1.1	+2.4%	+2.2%	+1.6%	+0.6**%**	−0.7%	−2.4%	−4.5%

I am carrying 15° as the baseline and would design toward 10° if the packaging allows. Script: `analysis/single_engine/engine_t`

2. Why not one bigger engine instead

The obvious alternative is one larger engine and no duct problem at all. I checked it against four real engines using their published diameters and masses, scaling the pyCycle K-450 cycle by flow. Every one of them tops out between Mach 0.95 and 1.00.

Setup	Static thrust	Body	Wave D/q	Mass	Thrust = drag at
2 × K-450 split exhaust	2 × 441 N	Ø167 mm	94 cm^2	21.2 kg	**M1.10**
2 × K-450 ducted tandem (v3)	2 × 441 N	Ø191 mm	115 cm^2	21.2 kg	M1.04
1 × AMT Lynx	1,569 N	Ø275 mm	431 cm^2	36.2 kg	M1.00
1 × PBS TJ80-120	1,200 N	Ø249 mm	303 cm^2	26.9 kg	M1.00
1 × JetCat P1000-PRO	1,100 N	Ø248 mm	309 cm^2	24.7 kg	M0.99
1 × AMT Nike	784 N	Ø215 mm	189 cm^2	21.7 kg	M0.99
1 × K-450G5	441 N	Ø167 mm	95 cm^2	16.2 kg	M0.95

Buying thrust with engine diameter buys the drag straight back. The Lynx has 3.6× the K-450's thrust and still only reaches Mach 1.002, because its 261 mm casing forces a Ø275 mm body and 431 cm^2 of wave drag. Across that entire 3.6× thrust range, static thrust per unit frontal area moves only from 1.41 to 2.37 N/cm^2, while mass climbs from 16 to 36 kg. The split-exhaust layout reaches 2.78 N/cm^2 precisely because it adds thrust without adding frontal area.

Limit: Engine diameters and masses are published datasheet values, but the cycle is the K-450 deck scaled by flow. As a check, the diameter implied by $\sqrt{\text{thrust ratio}}$ matches each engine's published diameter to within 0 to 10%, so flow scaling is fair for this engine class. It would not be fair across different compressor architectures.

3. Jet interference: measured, then computed

The one genuinely new aerodynamic risk is the side jets interfering with the body they now sweep. Two independent sources bound it.

Wind-tunnel data. NASA TM-X-3424 (Langley 16-ft transonic tunnel, twin-engine fighter, Mach 0.6 to 1.2, $\alpha = 0^\circ$) tabulates pure jet interference on afterbody drag coefficient, referenced to wing area, which is the same convention as our C_{D0}. For the afterburning nozzle, relative to flow-through jet pressure ratios:

Mach	0.6	0.8	0.85	0.9	1.2
ΔC_D, jet interference	+0.0009	−0.0005	−0.0031	−0.0004	−0.0027

The measured increment is *favorable* at every Mach above 0.6, including −0.0027 at Mach 1.2. I charge +0.0009 at every Mach, the single adverse value, which is conservative by construction.

Our own CFD. TM-X-3424 is an aft-fuselage and empennage geometry, not a mid-body side jet, so I ran it. Two matched OpenFOAM cases, jet-on and jet-off, differenced, using `rhoCentralFoam`, the same solver validated to within 1.5% of one-dimensional theory in the v3 nozzle study.

	Coarse, 950 cells	Full, 5,320 cells
Jet-off axial force on body	91.8 N	75.9 N
Jet-on axial force on body	100.7 N	84.1 N
Interference ΔC_D (wing area)	$+0.00034$	$\mathbf{+0.00032}$
Noise floor, force tail spread	± 0.00061	± 0.00062
Threshold the design needs	$+0.0014$	$+0.0014$

The grid refinement is the evidence, not the absolute number. Multiplying the cell count by 5.6 moved the absolute force by 16 N but moved the *difference* by 0.00002. A delta that survives a grid change while the absolute value does not is the signature of a real effect, and it is why differencing two matched cases was the right method rather than trying to predict drag outright.

The honest reading is **no measurable adverse interference at this fidelity**: the $+0.00032$ delta sits inside its own ± 0.00062 noise floor, so this is an upper bound rather than a value. Even adding the entire noise floor on top ($+0.00094$) stays under the $+0.0014$ the design needs.

Limit: This case is 2-D planar, which makes the nozzle an infinite spanwise slot. A slot jet cannot spread sideways and relieve itself, so it over-predicts the plume's surface footprint, and a real round nozzle sheds a counter-rotating vortex pair that shrinks it further. The 2-D result is therefore a conservative bound, deliberately. It covers aft-body closure only. **It does not cover the wing**, which is the one open item in this brief.

4. Where the plume actually goes

Both remaining risks, wing interference and hot-gas ingestion, are the same question: where does the plume travel? For a jet issuing into a crossflow, NASA CR-72893 states the experimental result directly, that the velocity trajectory depends solely on the ratio of jet momentum to crossflow momentum, $J = \rho_j V_j^2 / \rho_\infty V_\infty^2$. That makes it computable before any meshing.

	M0.25 takeoff	M0.40	M0.80	**M1.10 design**
Jet ρV^2	217 kPa	217 kPa	217 kPa	252 kPa
Air ρV^2	9 kPa	23 kPa	91 kPa	172 kPa
J	24.5	9.6	2.4	**1.47**
Penetration $\sqrt{J}\,d$	291 mm	182 mm	91 mm	**71 mm**
÷ body radius	3.5×	2.2×	1.1×	**0.9**×

J swings 17× across the envelope, and the two ends are different flow regimes. Statically the jet dominates and the plume projects clear of the body. At Mach 1.1 the crossflow nearly matches the jet, so **the plume cannot escape**: it turns aft within one body radius and sweeps along the body, over the wing root and past the rear inlets.

Two consequences, and the second one surprised me:

1. **Wing interference is real and is the open item.** The plume is attached at the design point, so it does wash the wing root. No 2-D case can speak to a lifting surface.
2. **Hot-gas ingestion is a high-speed problem, not a takeoff problem.** This inverts the obvious intuition and it changes the test plan: a static ingestion test would pass and prove nothing. It also fixes the inlet placement. With dorsal and ventral jets occupying the vertical plane, **the rear inlets belong on the sides**.

Limit: The $\sqrt{J}$ penetration scale comes from classic transverse-jet experiments at 90° injection. Our jets leave 15° off-axis, so the crossflow has far less turning to do and the numbers above are an upper bound on penetration. That strengthens the "plume stays attached" conclusion rather than weakening it.

5. Trim: the first version of this layout did not fly

My first version of this architecture used a single canted ventral nozzle. It does not work, and the reason is worth recording because it is not a drag problem.

At Mach 1.1 with the afterburner lit, the front engine's gross thrust is 1,293 N while the whole aircraft weighs 208 N. A 15° cant therefore throws **335 N sideways, 1.6× the aircraft's weight**. Trim *drag* is negligible, 0.1 N at Mach 1.1, because dynamic pressure is 86 kPa there. The binding constraint is **control authority at low speed**, where the jet load barely changes but q collapses:

	M0.25 takeoff	M0.40	M0.80	M1.10
Elevon load to trim	63 N	63 N	63 N	111 N
Elevon authority	120 N	306 N	1,226 N	2,317 N
Authority consumed	**53%**	21%	5%	5%

At rotation a single ventral nozzle spends over half the elevon's capability just holding the nose straight, leaving the rest for pitch control, gusts and rotation. That is not a flyable aircraft.

The fix costs nothing. Splitting the exhaust between opposed dorsal and ventral nozzles at the same station makes the side forces equal and opposite. Net side force and net pitching moment are exactly zero, for any CG, by symmetry; axial thrust is identical to the single-nozzle case. The cant loss is unchanged, so every number in this brief still holds, and the trim problem disappears by symmetry rather than by being trimmed away. Script: `analysis/single_engine/trim_check.py`.

6. What this does to the risks the reviewer raised

An aerospace student told me two engines plus inlet management was near impossible. He was right that it was the largest thing my model did not cover. Here is the current state of each item he named.

Risk	v3 ducted tandem	v4 split exhaust
Rear-engine duct distortion, surge	High, 1.2 m duct	Low: inlet is short and at the engine face
Hot-gas ingestion	High: 1,800 K jet routed through the body	Moderate, and worst at Mach 1.1 rather than at takeoff. Mitigated by lateral inlets
Shared-inlet surge coupling	Medium	Gone: separate inlets
Hot-section liner through airframe	Required	Gone
Off-axis jet trim	n/a	Solved by symmetry with paired nozzles
Jet interference, aft body	n/a	Measured and passing: $\Delta C_D + 0.00032 \pm 0.00062$ vs $+0.0014$
Jet interference, wing root	n/a	**Open.** Needs a 3-D case with the wing

7. Confidence, stated as numbers

- **85%** that split-exhaust tandem is the best of the seven setups in Section 2 for Mach 1.1. This rests on geometry and is robust.
- **55%** that it actually achieves positive margin at Mach 1.1. This is up from 40% before the interference CFD, because the quantity the margin rested on is now measured rather than assumed and came in at a third of the charged penalty. The remaining 45% is wing-root interference plus the plain fact that $+0.6\%$ is thin against a model with real error bars.

One correction I owe you from my own process: the first version of this analysis quoted $+0.9\%$ margin, and that number was wrong. The interference penalty I believed I was charging never reached the drag calculation, because the drag routine reads C_{D0} from the Monte Carlo sample rather than from the build. It is now plumbed correctly and guarded by an assertion that fails if it ever stops biting. The corrected figure is $+0.6\%$, and the architecture conclusion did not change.

8. What I would like from you

1. **Is the split-exhaust layout sound, or is there a reason this is not done?** Canted side exhausts are unusual on a slim supersonic body. I would rather hear now that there is a known failure mode than discover it on the stand.
2. **The wing-root interference case.** Worth building a 3-D mesh with the wing present, or is there a correlation or report that would settle it more cheaply?
3. **Ram data.** Still my largest unchecked engine input. Do you know of any thrust-versus-flight-speed data for micro-turbojets? Everything I have found is static test-stand data.
4. **The inlet distortion question as a paper.** Section 4 implies a measurable, publishable question: what distortion descriptor (DC60 per SAE ARP1420) does a short side inlet deliver at Mach 1.1, and does a 90,000 rpm radial-compressor micro-turbojet survive it? I have not found that measured anywhere. If you think it stands up, I would like to scope it properly.

References

[1] "Jet exhaust and support interference effects on the transonic aerodynamic characteristics of a fighter model," NASA TM-X-3424, 1976. NTRS 19770009035.
[2] "Investigations of a turbulent jet in a crossflow," NASA CR-72893, 1971. NTRS 19720000436.
[3] E. S. Hendricks and J. S. Gray, "pyCycle: a tool for efficient optimization of gas turbine engine cycles," *Aerospace* 6(8):87, 2019.
[4] C. J. Greenshields et al., "Implementation of semi-discrete, non-staggered central schemes. . . for high-speed viscous flows," *Int. J. Numer. Meth. Fluids* 63, 2010 (rhoCentralFoam).
[5] E. R. Katz, "Flight investigation at high-subsonic, transonic, and supersonic speeds," NACA RM L9F02, 1949.
[6] D. P. Raymer, *Aircraft Design: A Conceptual Approach*, AIAA, Ch. 12.
[7] SAE ARP1420C, *Gas Turbine Engine Inlet Flow Distortion Guidelines*.
[8] KingTech Turbines, K-450G5 user manual, p. 44. AMT Netherlands Nike and Lynx datasheets. JetCat P1000-PRO datasheet. PBS TJ80-120 datasheet.
[9] Brief v3 (26 October 2026) for all inputs, model validation and Monte Carlo methodology carried forward unchanged.

Scripts behind this brief, all in the project repo: `analysis/single_engine/engine_trade.py` (architecture trade), `trim_check.py` (Section 5), `plume_path.py` (Section 4), `jet_interf_cfd.py` (Section 3). Each asserts that it reproduces brief v3's published numbers before reporting new ones.