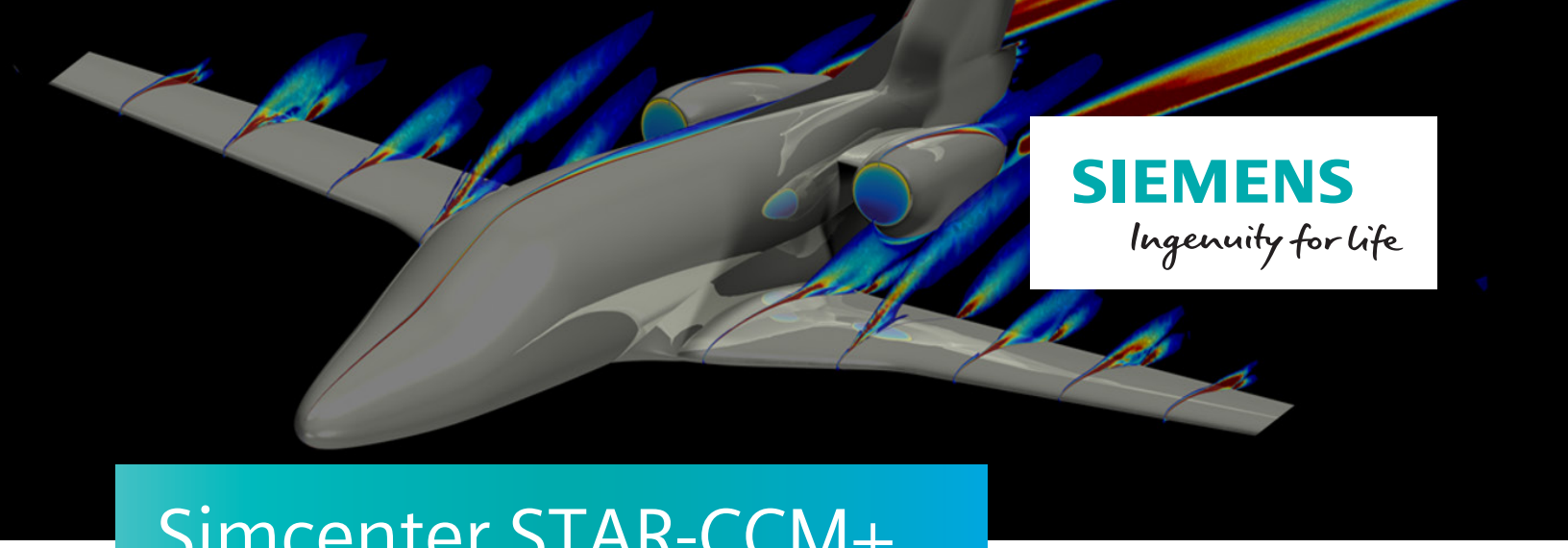


SIEMENS
Ingenuity for life

Simcenter STAR-CCM+ for aerospace aerodynamics

Improving performance with multiphysics CFD simulation and design exploration

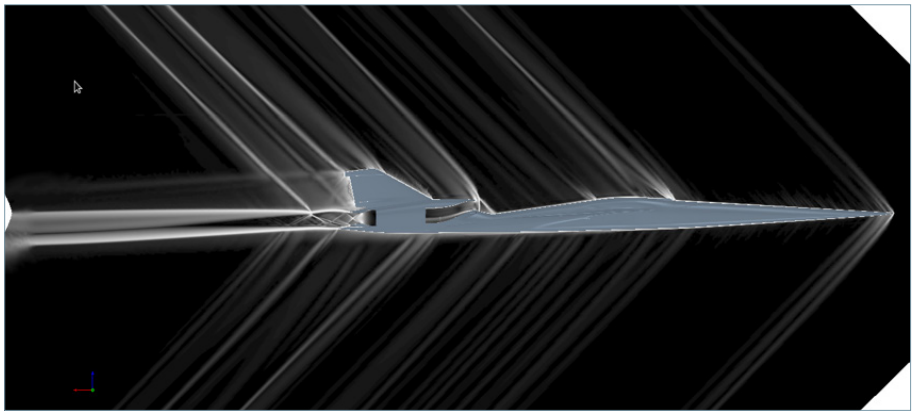
Benefits

- Improve aerodynamic performance and explore the full flight envelope with built-in design exploration
- Save engineering time using an integrated and automated CAD-to-solution workflow
- Predict real-world aerodynamics with a comprehensive multiphysics digital twin
- Minimize time-to-solution with efficient algorithms and parallelization across the simulation workflow

Summary

Today's aerospace and defense market is awash with change. New urban air mobility and unmanned aircraft designs are pushing the envelope with novel approaches, including using integrated propulsion systems, lifting bodies and/or electric motors. Likewise, a resurgence of interest in supersonic and hypersonic designs is driving the incorporation of more sophisticated physics models into production computational fluid dynamics (CFD). At the same time, aerospace manufacturers are faced with ever-tightening requirements, both regulatory and competitive, which translates to greater demands on CFD tools. Therefore, a modern CFD solution must facilitate the following capabilities:

- Model the full geometry and physics needed to predict real-world aerodynamic performance including influences from propulsion systems, control surfaces and landing gear
- Minimize engineering time and manual work to set up, run and postprocess aerodynamics simulations
- Automate all steps in the simulation process to quickly generate aerodynamic databases and investigate a full range of design configurations
- Shorten time-to-solution with parallelization and efficient computational algorithms designed for modern high-performance computing (HPC) systems



Simcenter STAR-CCM+ for aerospace aerodynamics

- Enable automated optimization studies to maximize aerodynamic performance in a production environment

Simcenter STAR-CCM+ offers the solution

Simcenter™ STAR-CCM+™ software, a part of the Xcelerator™ portfolio, the comprehensive and integrated portfolio of software and services from Siemens Digital Industries Software, is a general purpose multiphysics CFD simulation tool. It can be applied to a wide range of external aerodynamics problems, but it offers aerospace and defense organizations much more than that. The same software that computes the aerodynamic database can be used for propulsion integration studies, tank sloshing simulations, avionics cooling, e-powertrain analysis and cabin comfort modeling. With Simcenter STAR-CCM+ you can:

- Stay productive: Build every simulation with a pipelined workflow that includes all steps of the simulation process, allowing you to re-use a single setup for hundreds of cases

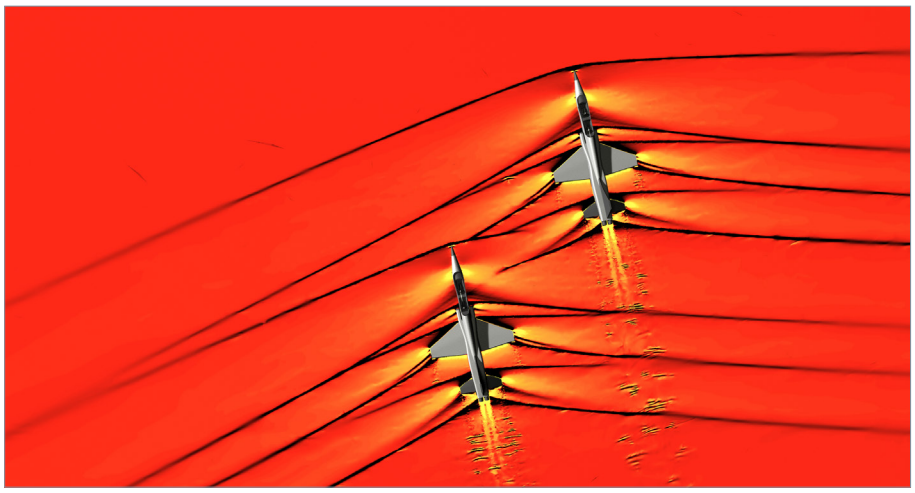


Figure 1. Interacting wave patterns radiating from a pair of supersonic T-38 aircraft flying in formation.

- Model the complexity: Model geometries with up to thousands of parts using advanced geometry creation/preparation and robust meshing tools
- Cover more applications: Simulate all speed ranges, from incompressible to hypersonic as well as chemical reactions, multiphase flow, transition and conjugate heat transfer
- Get results faster: Leverage parallelization across the entire simulation workflow and take full advantage of available hardware, whether in-house or on the cloud
- Explore the possibilities: Automatically sweep through the flight envelope or optimize for aerodynamic performance with built-in intelligent design exploration

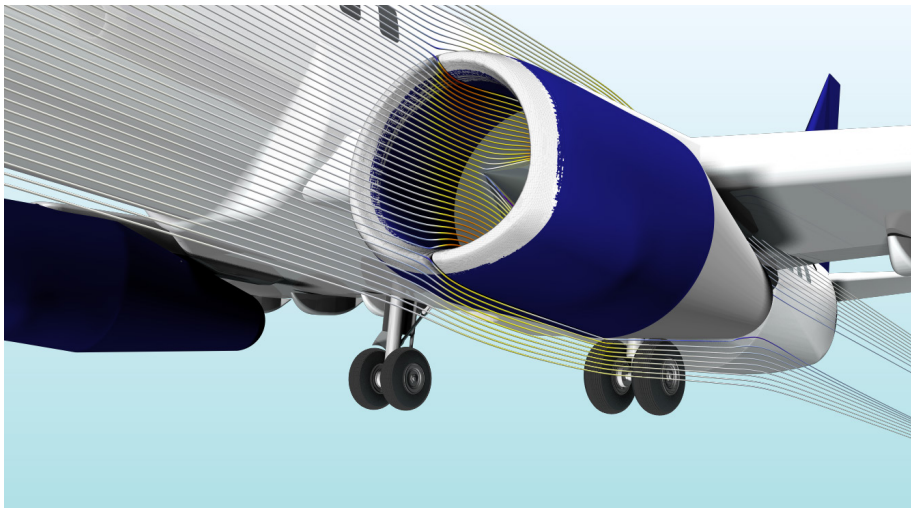


Figure 2. Predicted ice accumulation and streamlines around an engine nacelle.

Model all of the geometry
Simcenter STAR-CCM+ can be used to handle a broad range of geometric fidelity, including extremely complex designs with thousands of parts. This no-compromise approach to geometry representation is enabled by:

- Computer-aided design (CAD) clients that can be used to provide live, two-way geometry and parameter transfer between CAD tools and Simcenter STAR-CCM+ to perform design and configuration changes. For example, you can change a nacelle toe-out angle in Simcenter STAR-CCM+ by modifying the native CAD from within the CFD environment
- The built-in 3D CAD tool can be used to quickly clean up or repair CAD as needed to prepare it for meshing. For example, it is quick and easy to seal up the gaps in the CAD of the wing tank to create a watertight part

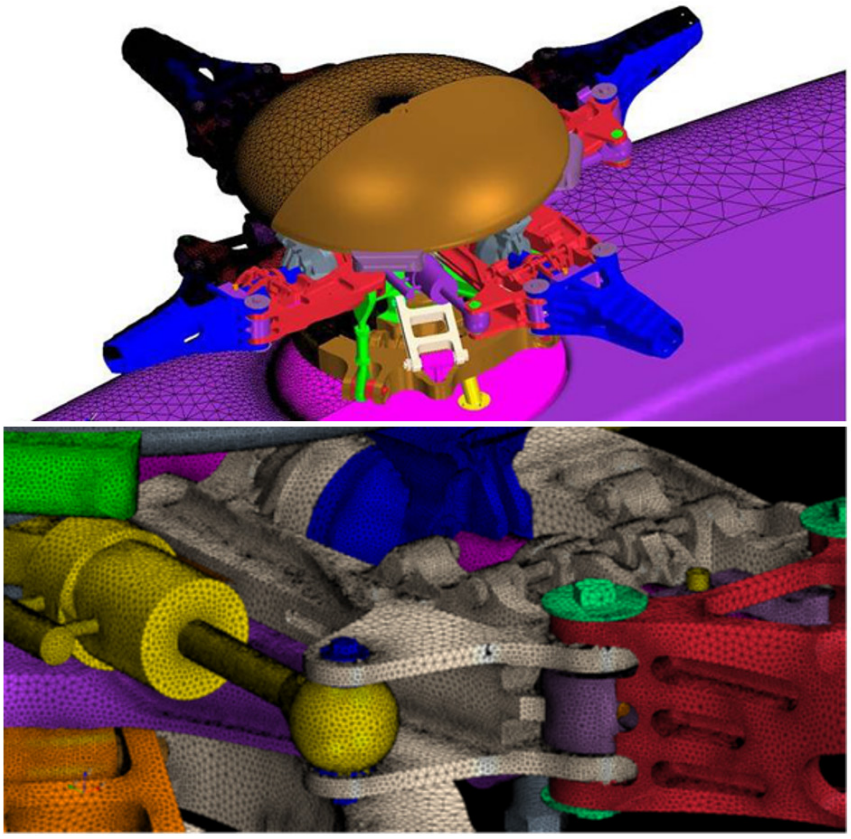


Figure 3. Details of the Sikorsky S-92A hub design with surface mesh. Hub drag accounts for 30 percent of total drag. Using Simcenter STAR-CCM+ allows the full geometry to be included in the simulation.

- Efficient parallel algorithms allow even the largest computational grids to be generated in a reasonable amount of time
- The CAD-to-mesh pipeline is designed so that an initial setup can be re-used for further studies. For example, the control positions of a high lift wing configuration can be parameterized so an entire family of meshes can be automatically created simply by entering the varying flap angles. Further, this entire process can be automated to reduce the human intervention to a bare minimum

Include all physics
Simcenter STAR-CCM+ provides comprehensive physics coverage, allowing the user to analyze a vast variety of flow conditions:

- Model all speed ranges: incompressible, subsonic, supersonic and hypersonic. The same software used to predict the performance of a supersonic airliner cruising at twice the speed of sound can be used for take-off and landing as well as the heating, ventilation and air conditioning (HVAC) systems in the cabin and the cooling of the avionics. The wide

variety of turbulence models lets you choose the right one for your simulation, whether it's a steady-state Reynolds-averaged Navier–Stokes (RANS) case or a time-accurate Detached eddy simulation (DES) run

- Simulate bodies in relative motion with fixed paths, constrained motion or a full six degrees-of-freedom (DOF). Whether you are modeling a launch vehicle stage separation or delivery drone package drop-off, you only need to invest in one software product
- Use a full range of chemistry and thermodynamics options to model the physics of your application with the right fidelity. Hypersonic re-entry flows, helicopter exhaust reingestion, rocket launches and much more can be explored with a high degree of realism
- Choose from a large suite of simplified physics options to investigate the major effects of specific phenomena or devices without the full expense of modeling them in detail. For example, the impact (steady or time-accurate) of a rotor or propeller on the flow can be modeled using the blade element method at a significant savings in computational effort compared to a fully-meshed, rigid-body motion simulation

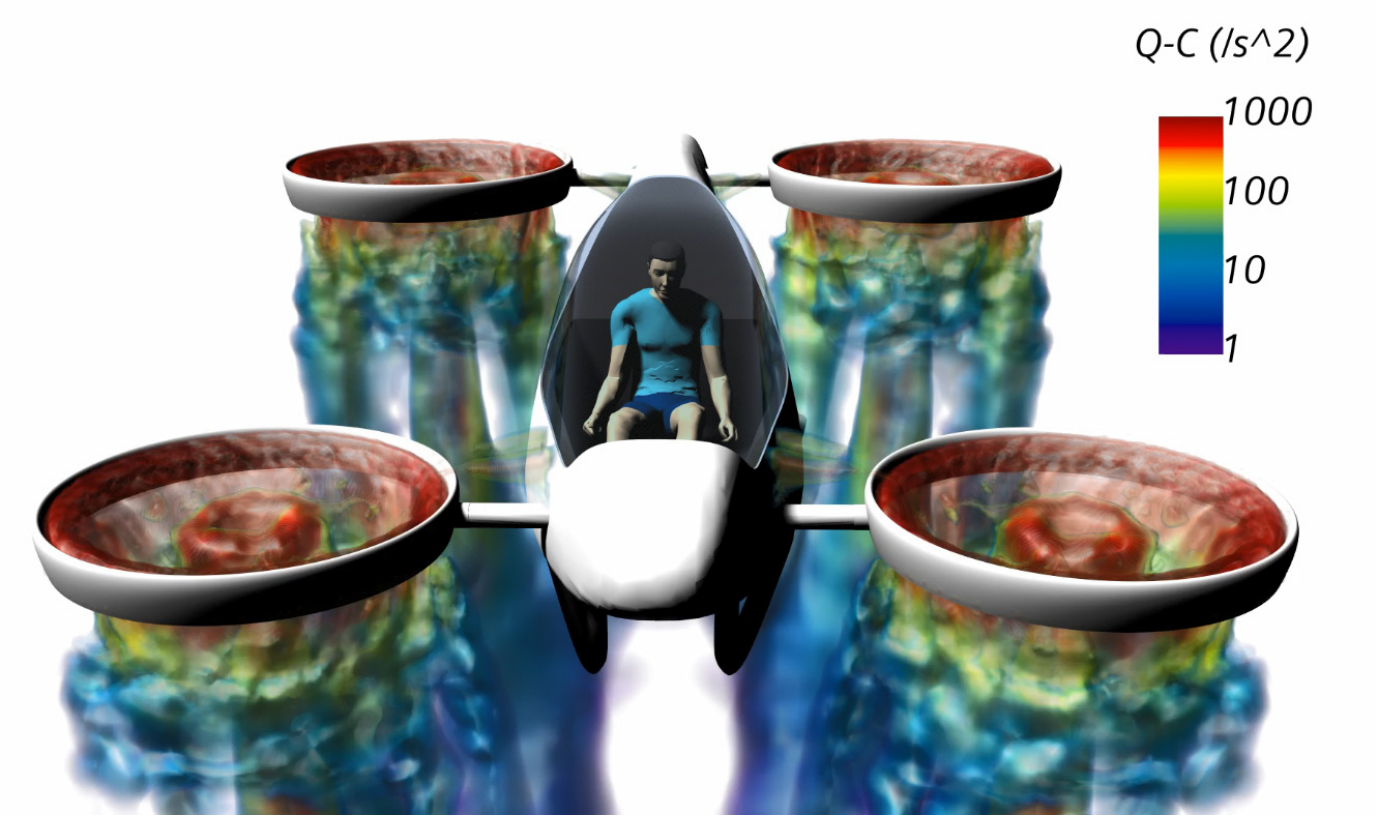


Figure 4. Visualization of the prop wash from an urban air mobility vehicle where the propeller effects were computed using the blade element method.

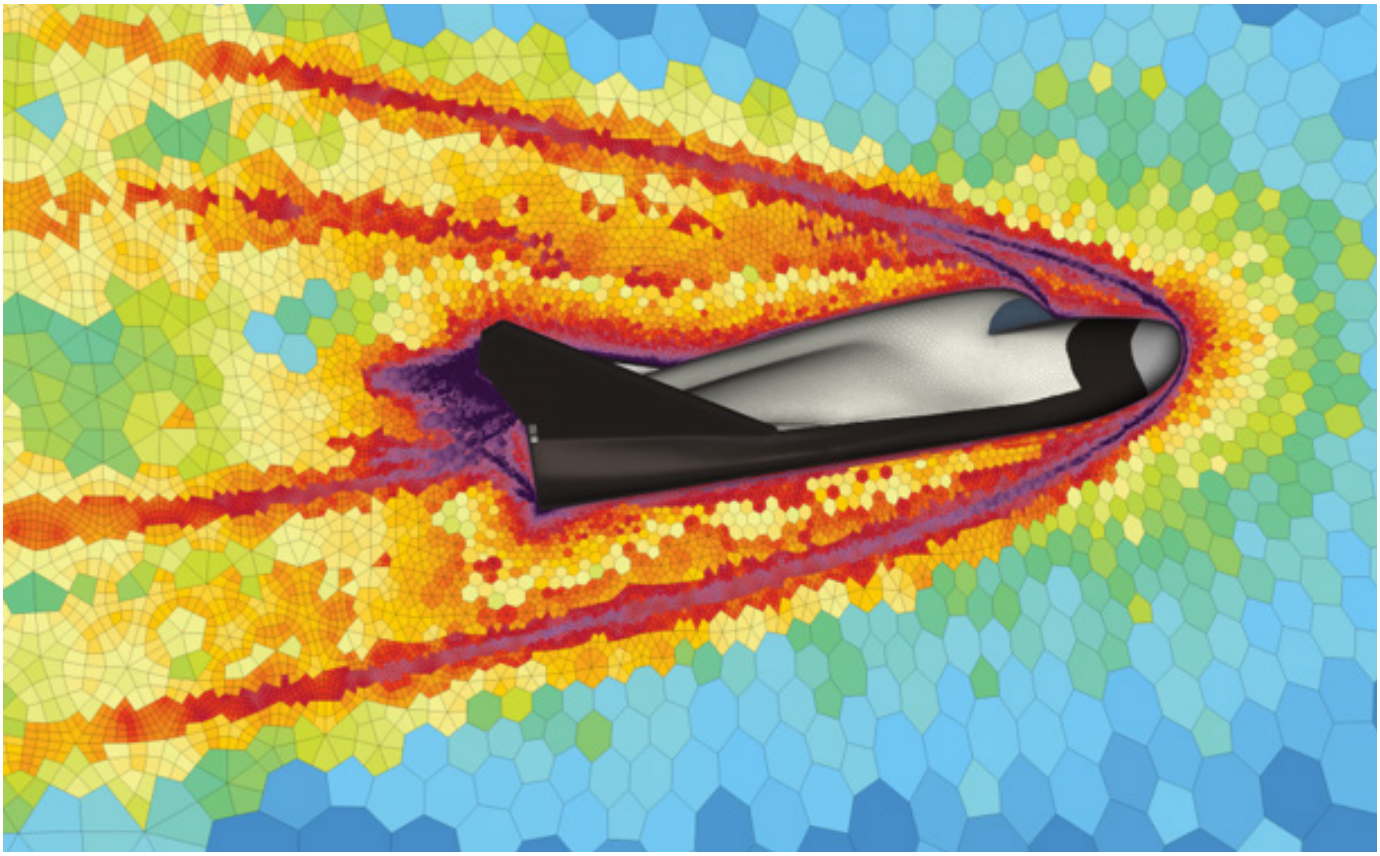
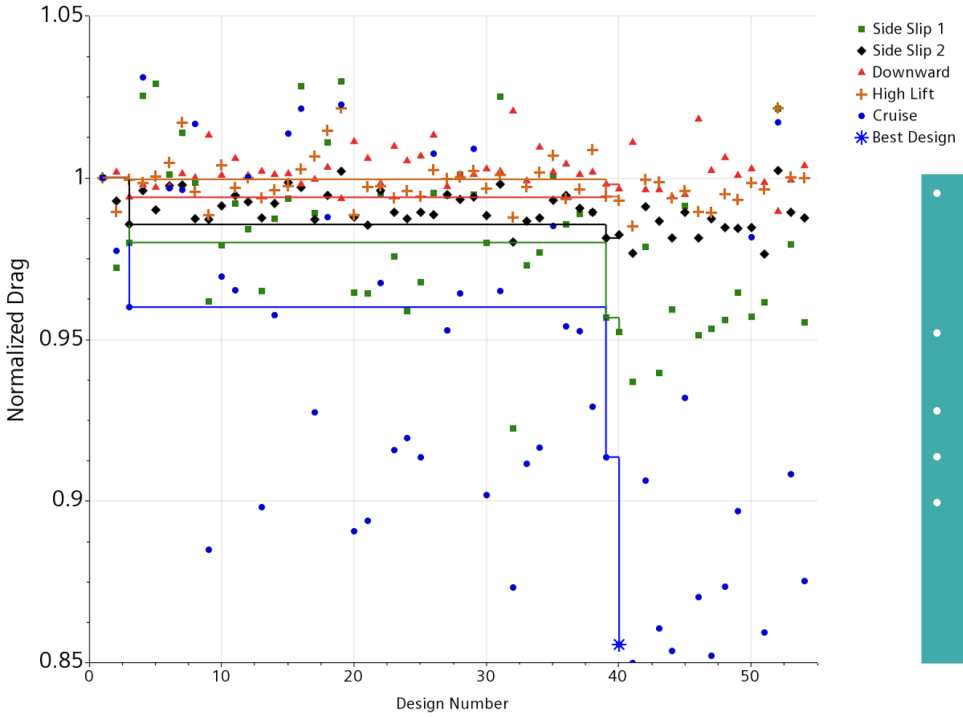


Figure 5. Adaptive mesh refinement criterion function values in Mach 10 equilibrium air model flow around the HL-20 vehicle.

Go faster

Simcenter STAR-CCM+ is designed to get results to engineers as quickly and efficiently as possible. Tools are included that eliminate most or all of the manual intervention that was required in the past when running a series of simulations.

- With the flexible "pipeline" approach, the complete simulation process – geometry preparation, meshing, solving the physics and postprocessing the results – can be done in one action. Template files dramatically reduce the initial setup and ensure consistent use of best practices. Once a case is prepared, entire families of related conditions can be explored automatically. Running an angle of attack sweep or Mach number variation is simple
- Adaptive mesh refinement (AMR) lets users set up a single baseline mesh that is refined on the fly to capture the physics of interest. For example, let the code figure out where you need resolution in order to capture the shocks in an angle of attack sweep in supersonic flow
- Set up sophisticated data analyses and flow visualization ahead of time. Whether you run one case or hundreds, Simcenter STAR-CCM+ automatically creates images and plots to help your team make critical design decisions quickly and with confidence. Look at forces, moments, loads, heat flux, vortex locations, pressure coefficient; if there isn't already a built-in function definition, you can create your own using any combination of field variables
- Efficient parallel algorithms let you use all your available compute resources to get the turnaround you need for both meshing and flow solution. Whether your resources are in-house or in the cloud, you can use thousands of cores to cut days or weeks off the runtime for that full airliner with gear down and flaps extended
- Flexible licensing options enable you to use Simcenter STAR-CCM+ on your workstation, as part of an in-house cluster or in the cloud to maximize throughput of your high-fidelity simulations or to finish that aerodynamics database before the critical design review



- Full external aerodynamics design exploration of a commercial business jet at 5 operating conditions
- Weighted sum of 15 unique objectives
- 10 performance constraints
- 30 days compute time
- Design exploration results:
  - Reduced cruise drag by 14%
  - Improved 14 other objectives
  - Maintained 10 design constraints

Figure 6. Summary of design optimization study and results for the integrated engine nacelle on the Eclipse business jet.

Explore the possibilities

- Run through a full design sweep with the push of a button. Creating an aerodynamic database has never been easier and you can interrogate the results for all the cases from a central console. For example, you can run through the full range of Mach numbers, angle of attacks, side slip angles, altitude and engine thrust settings with a single submission
- Run full-featured, multi-objective, multi-constraint design optimization analyses. Include as many flight conditions as needed in the evaluation of each design. Don't just optimize a nacelle by itself; include the entire vehicle and see how each new design impacts overall performance
- Use powerful adjoint-based methods to find optimal shapes for specific parts on a vehicle. For example, minimize drag over a wing or maximize mass flow through an air intake duct

Siemens Digital Industries Software
siemens.com/software

Americas +1 314 264 8499
Europe +44 (0) 1276 413200
Asia-Pacific +852 2230 3333

83589-C3 3/21 K