NEMESYS

CFD Game Changer

What’s wrong with today’s CFD tools?

- **Slow -** Long simulation runtimes delay decisions
- **Expensive -** High license and hardware costs
- **Complex -** Steep learning curve and dependency on experts

Nemesys Solution

- **Fast -** simulations with GPU-optimized algorithms

- **Simple -** can run on standard GPU Desktop or Laptop

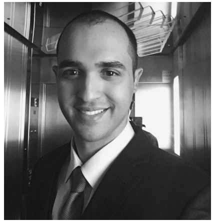
- **Accuracy with no compromise**

NEMESYS

Meet the Team Behind Nemesys

- Established by a group of CFD experts who saw a need for a faster,reliable and simple simulation platform.
- Many years of experience in high-performance computing, aerospace, oil and gas, and automotive industries.
- We serve clients worldwide, delivering cutting-edge simulation solutions.

/elisan-magalhaes



PhD. Elisan Magalhães, CEO

/arthurmazevedo

PhD. Arthur Mendonça, CIO

/danielbotezelli

PhD. Daniel Botezelli, CTO

Revolutionizing CFD with Nemesys

What Makes Nemesys Unique?

- Unstructured High-Order Formulations: Specialized for Implicit Finite Volume methods.
- GPU-Centric Computation:
  - 99.99% of the computation performed on the GPU, maximizing performance.
  - Minimal CPU-GPU data transfers.
- Innovative Mathematics: Reduces billions of operations and solver iterations.
- Precision Focus:
  - 64-bit floating-point accuracy (no precision reduction for speed).

Performance and Accessibility

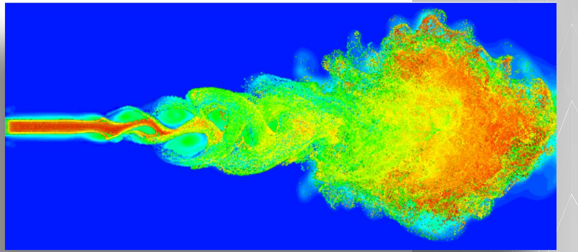
Efficient, Cost-Effective Hardware

- Standard GPU Compatibility:
  - Works with Nvidia RTX 20, 30, 40, or 50 series.
  - No need for costly hardware—handle meshes up to 12M elements with a US$ 1,500 GPU.
- VRAM Optimization: 2–3 GB per 1M volumes.
- Multi-GPU Support: Coming in 2025/Q2

Independence and Simplicity

- In-House Development: No third-party dependencies for the core libraries.
- Not AI-Based: No data training—just set it up and start running.

Supported Physics – Turbulent Flows



Applications

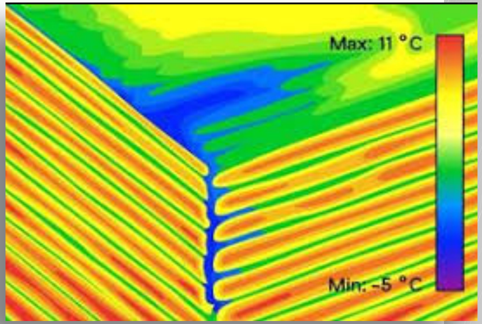
- Used in aerospace, automotive, and energy industries.
- Ideal for modeling fluid motion at high speeds.

Capabilities

- Supports **RANS-based turbulence models**.
- **LES (Large Eddy Simulation)** currently under development.

NEMESYS

Supported Physics – Total Energy



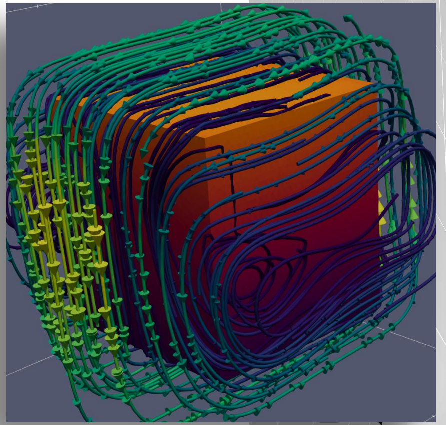
Applications

- Essential in power plants, HVAC systems, and electronics cooling.
- Addresses all heat transfer modes to regulate temperatures and improve system performance.

Key Innovation - Thermal Capacitor Method:

- Advanced, highly accurate tool for nonlinear energy capture.
- Specifically designed for cooling applications.
- Reference: Read the Study

Supported Physics – Conjugate Heat Transfer



Applications

- Critical in aerospace, automotive, and chemical industries.
- Integrates fluid flow with solid conduction for detailed thermal analysis.

Key Feature - Coupled Interface Boundary:

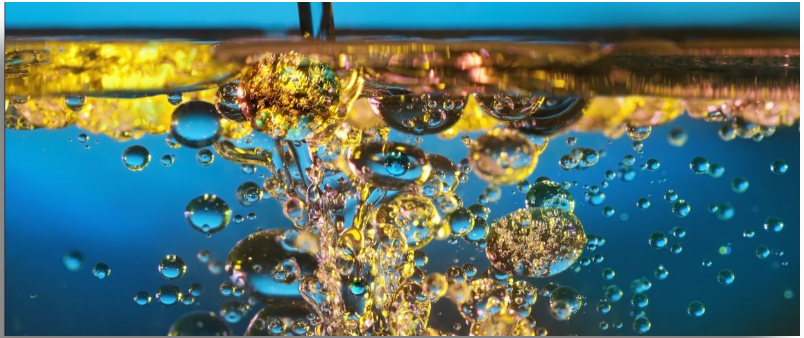
- Optimized for computing interfaces between different phases.
- Ideal for components like turbine blades and heat exchangers.

ΝΣΜΣSΨS

Supported Physics – Multi-Phase & Phase-Change

Applications

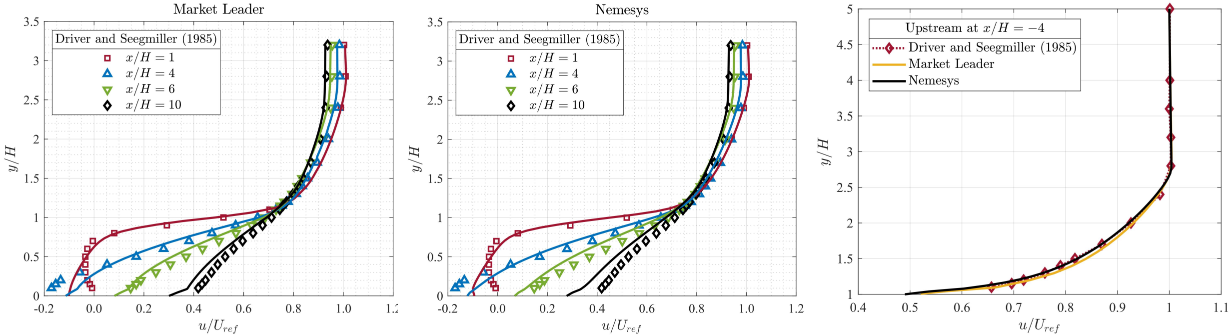
- Models interactions between phases (e.g. water-oil).
- Captures transitions from one phase to another (e.g. solid-liquid).
- Optimizes the design and operation of pipelines, reactors, and other critical systems.

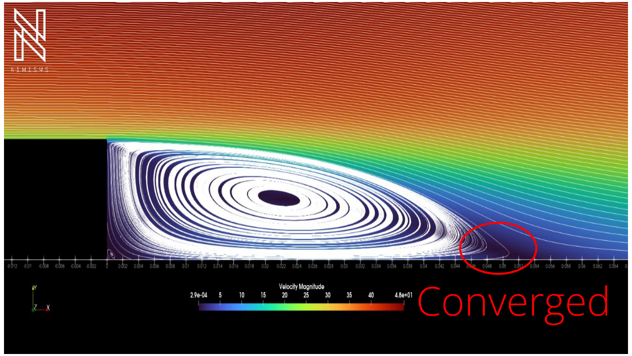


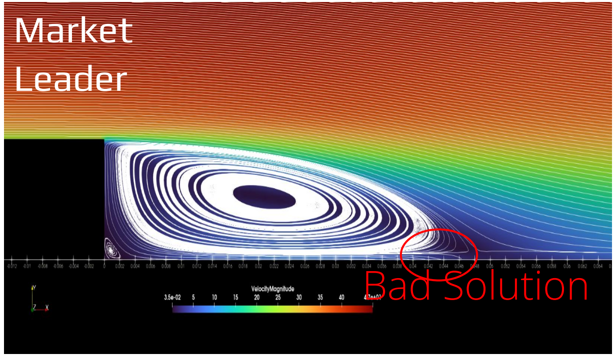
Our Portfolio

Feature	Status	Details
Heat Transfer	Launched	Heat Conduction, Non-Linear Properties
Fluid Dynamics	Launched	Total Energy, Newtonean Fluid
Multiphysics	Launched	Conjugate Heat Transfer, Volume Of Fluid, Discrete Phase Model (2025/Q3)
Multiphase	Launched	Solid-Fluid, Fluid-Gas
Laminar Regime	Launched	Pressure-Based
Turbulent Regime	Launched	k-ε Standard, k-ω SST, k-ω Standard
Incompressible Flow	Launched	Pressure-Based
Compressible Flow	Launched	Pressure-Based
Supersonic Flow	In progress	Launch in 2025/Q2
Hypersonic Flow	Not started	Launch in 2026/Q1

Benchmark Example – RANS Turbulence Model







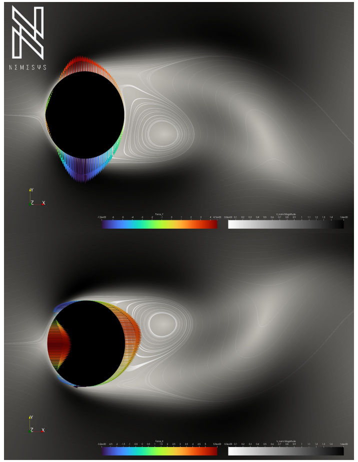
Same methodology, precision, and computational resources.

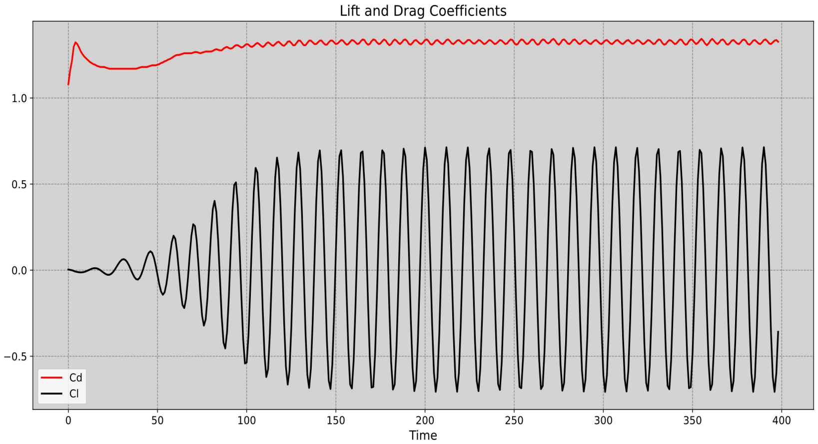
1,000x Faster.

The results obtained from NEMESYS CFD were validated for the RANS (k-ε and k-ω SST) turbulence models, showing excellent agreement with both experimental data and other industry-standard CFD tools, ensuring reliability in complex simulations.

Driver, D. M., & Seegmiller, H. L. (1985). Features of a reattaching turbulent shear layer in divergent channel flow. AIAA Journal, 23(11), pages. https://doi.org/10.2514/3.8890

Benchmark Example – Flow Past Cylinder





Study	Year	Methodology	C_L	C_D	St
Present Study	2024	Computational	± 0.76	1.31 ± 0.019	0.198
Russel and Wang	2003	Computational	± 0.75	1.3 ± 0.02	–
Choi et al.	2007	Computational	± 0.8	1.25 ± 0.03	–
Norberg C.	1994	Experimental	–	–	0.197

Same methodology, precision, and computational resources.

1,500x Faster.

Transient, drag, and lift analyses, demonstrating robust correlation with experimental findings and alignment with other leading computational results. This validation confirms the tool's accuracy in handling dynamic and aerodynamic simulations.

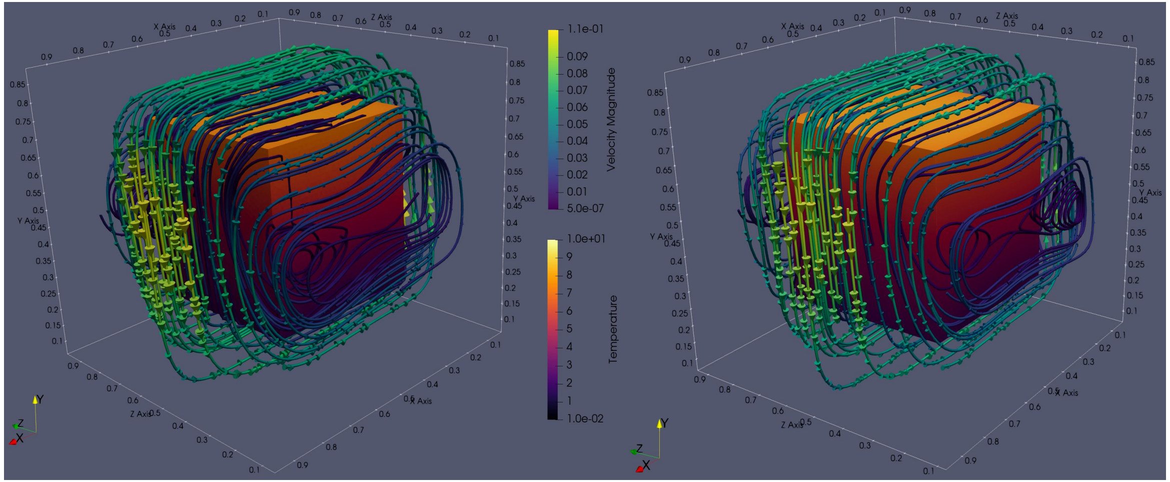
Russell, D. and Jane Wang, Z., (2003) "A cartesian grid method for modeling multiple moving objects in 2d incompressible viscous flow," Journal of Computational Physics, 191(1), pp. 177–205. https://www.sciencedirect.com/science/article/pii/S0021999103003103

Choi, J.-I., Oberoi, R. C., Edwards, J. R., and Rosati, J. A., (2007) "An immersed boundary method for complex incompressible flows," Journal of Computational Physics, 224(2), pp. 757–784. https://www.sciencedirect.com/science/article/pii/S0021999106005481

Norberg, C., (1994) "An experimental investigation of the flow around a circular cylinder: influence of aspect ratio," Journal of Fluid Mechanics, 258, pp. 287–316

Benchmark Example - CHT - Natural Convection With Obstacle

6 Million elements Three-Dimensional Natural Convection CHT Benchmark. Our model brings an unique interface solution. While we solved in 112.0s, market leader took 97,201.5s to achieve same results.



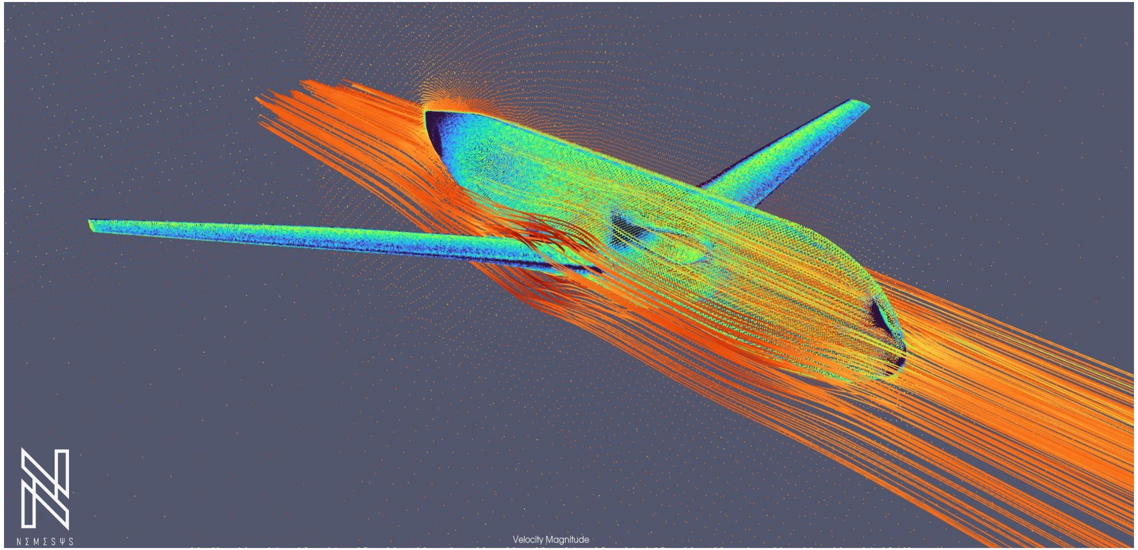
Same methodology, precision, and computational resources.

800x Faster.

The findings from NEMESYS were rigorously validated for Conjugate Heat Transfer (CHT) models, exhibiting strong concordance with experimental observations and consistency with other established CFD applications. This validation underscores the software's precision in simulating thermal interactions and heat transfer phenomena.

Benchmark Example - Complex Geometries and Subsonics

Simulation of a Flow 15 Million Elements under RANS Turbulent Regime. Nemesys shown great capability under complex geometries and higher mesh density. Test case for High Lift Prediction Workshop (HLPW) NASA model.



Same methodology, precision, and computational resources.

1,200x Faster.

The results for simulations involving complex geometries and subsonic flows were meticulously tested, showing excellent alignment with data and comparability with other prominent CFD platforms. This validation ensures the tool's effectiveness and reliability in accurately modeling intricate structures and High Reynolds aerodynamics.

Mouton, S., Charpentier, G., & Lorenski, A. (2023). Test Summary of the Full-Span High-Lift Common Research Model at the ONERA F1 Pressurized Low-Speed Wind Tunnel. Paper presented at the AIAA SCITECH 2023 Forum, Special Session: High Lift Common Research Model Ecosystem. https://doi.org/10.2514/6.2023-0823

model at: https://hiliftpw.larc.nasa.gov/

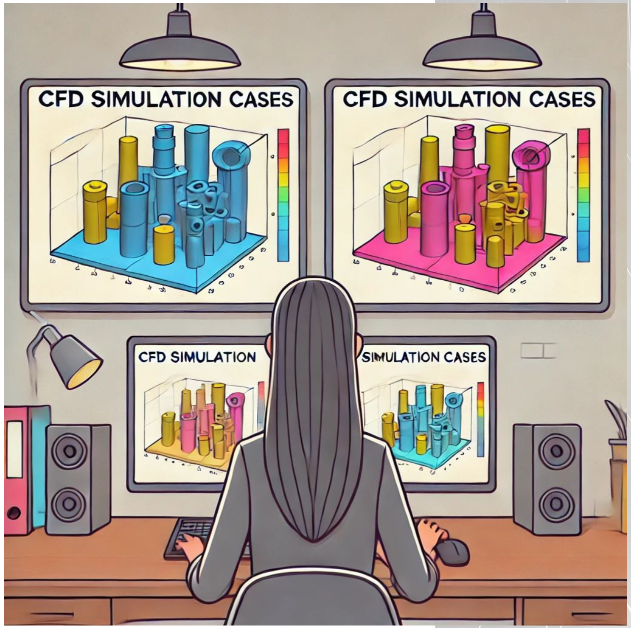
Let's test Nemesys with Your Model

What we need from you:

- Define your **CFD test requirements**.
- Share a **CAD file** (*.STEP* or *.IGES*).
- Provide details on:
  - **Physics and models** (e.g., fluid/thermal interactions).
  - **Thermal properties** of materials.
  - **Boundary and initial conditions** for the simulation.

What we'll do:

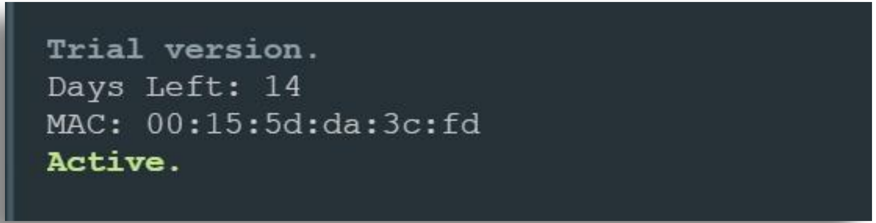
- Generate a **mesh file** (*.su2*) using Gmsh.
- Handle **pre-processing, solver setup**, and simulation preparation.
- Run the **Nemesys solver**.
- Export results in standard formats (*.vtk* or *.vtu*).



Trial Version - Nemesys CFD on your computer

- **System requirements:**
  - **Nvidia GPU** that supports Compute Capability 8.6 or higher.
  - Check yours at (https://developer.nvidia.com/cuda-gpus).
  - We recommend a GPU with 16 Gb or more.

- **Get your free-trial license:**
  - Provide MAC address of the machine where Nemesys CFD will be installed.



- **Installing:**
  - We will schedule a **guided setup and run** to ensure you unlock Nemesys full potential.
  - You may run unlimited number of simulations throughout the trial period.

Integrating Nemesys into Your Workflow

Pre-Processing:

- Generate mesh in `.su2 format` using your preferred software.
- Modify YAML configuration files for:
  - `control`: Solver settings and physical parameters.
  - `conditions`: Initial and boundary conditions.
  - `output`: Specify data and results to save.
  - `thermal_properties`: Material properties and types.

```yaml
boundary_conditions:
  velocity:
    enabled: true
    conditions:
      - markertag: 3
        type: solid_wall
        order: first
        value: [0.0, 0.0, 0.0]
```

Running the Solver:

- Execute the Nemesys solver to process your simulation.

Post-Processing:

- Locate results in `../ans/` folder (`.vtu`/`.vtk`).
- Analyze outputs using ParaView or Visit.

Thank You!

info@nemesys.solutions

NΣMΣSΨS