FlowVision

Unique Capabilities

CAPVIDIA

FlowVision

Unique Capabilities

No CAD preparation/simplification for CFD

FlowVision SGGR^{TM}: Ultimate CAD Resolution Accuracy and Unique Meshing Approach

1. In traditional CFD solvers, there are two main problems regarding the mesh generation and CAD handling. The first is the excessive manual work for meshing geometries for CAD. Engineers spend a lot of time preparing CAD models for CFD. **Weeks can be spent transferring a CAD file, de-featuring/simplifying the geometry, meshing the design** and creating the fluid domain. The second problem is the **loss of original CAD geometry** during these defeaturing or simplification processes.

2. Thanks to the Sub-Grid Geometry Resolution **(SGGR^{TM})** technology, FlowVision directly uses **original CAD data without any loss in details unlike other traditional methods (See Fig. 1).** The SGGR^{TM} establishes the basis for a lossless transition from the original CAD to CFD. Without SGGR^{TM} the Cartesian grid technology could never have attained the desired accuracy.

3. The mesh in FlowVision is a **parametric Cartesian grid**. The user starts the project with a CAD model and creates an initial Cartesian grid. Then applies geometry or solution based adaptations. Adaptations on geometries can be based on real or arbitrary shaped objects.

4. When the mesh is built in a parametric manner, the users can replace the calculated geometries without losing the original mesh behavior. **The geometries can be replaced with one click and the adaptation regimes will be strictly preserved**.

5. When all these are combined with the strict CAD evaluation approach of FlowVision for self intersections etc., the engineers can be sure of the CAD correctness and resolution. **Thus, they can focus on the real physics of the problem instead of focusing the process of building the CFD project itself.**

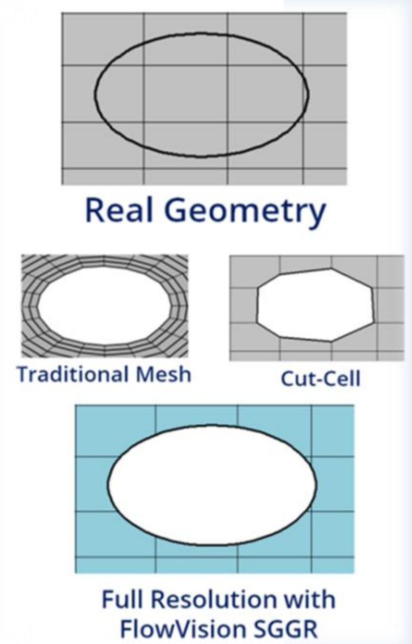

Figure 1: Difference between traditional methods and SGGR^{TM}.

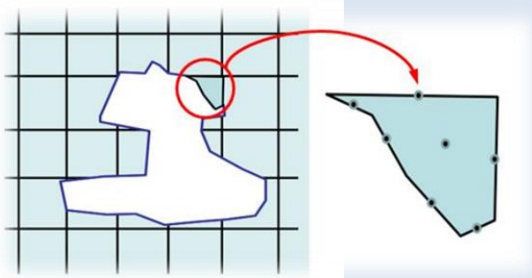

Figure 2: Surface reconstruction in SGGR^{TM}.

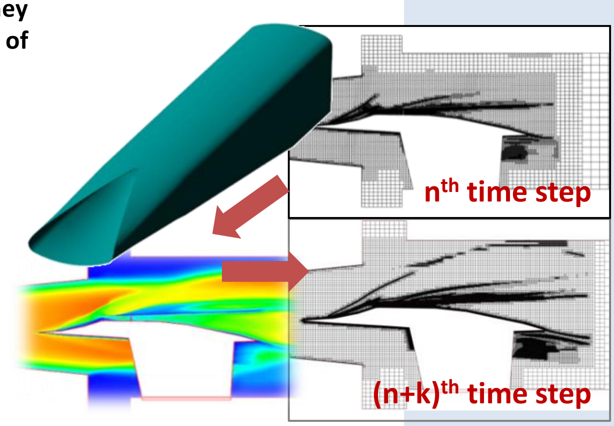

Figure 3 : Adaptation to solution, based on Mach number in a hypersonic aerodynamics case. (courtesy of Central Institute of Aviation Motor Development)

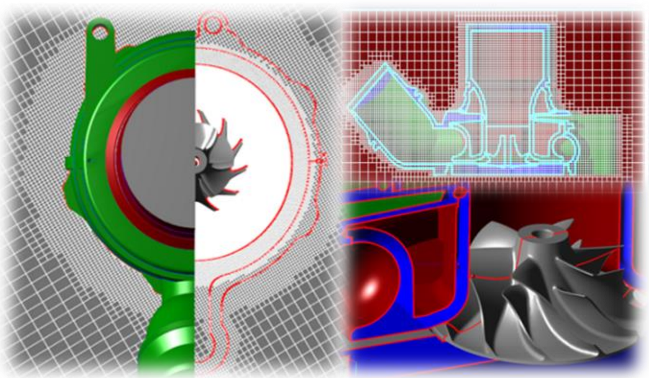

Figure 4 : Adaptation to real objects in a turbocharger conjugate heat transfer case. (courtesy of DSM, Mitsubishi Motors Corporation, USA)

No mesh deformation dependency – No overset mesh:
13 years of experience in moving body calculations

Moving Parts in the Computational Domain

1. In FlowVision any geometry can move with respect to the computational domain during the course of calculation. The movements can be in **6 degrees of freedom**, and unlike other solvers FlowVision is not restricted by any mesh deformation etc.

2. The movements can be implemented through **kinematic (based on user input) or dynamic (based on fluid and inertial forces) definitions.**

3. FlowVision allows body **contacts/intersections (full closure of the flow channel)**. While obtaining the full closure is an important point to address, FlowVision Moving Bodies takes it one step further and lets the geometries exit/enter the computational domain when needed.

4. The users can define movements through the **Formula Editor (Fig. 8 shows the interface).** The Formula Editor of FlowVision has the following capabilities:
   - ❖ Acts like a scientific calculator:
     - **Scalar inputs & vector operations**
     - **Logarithmic functions**
     - **Trigonometric functions**
     - **Logical operators**
   - ❖ **Any calculated variable can be used as input for the formula editor (a basic example is shown on Fig. 8)**

Figure 5: Blood flow simulation of a beating human heart. (The Living Heart Project)

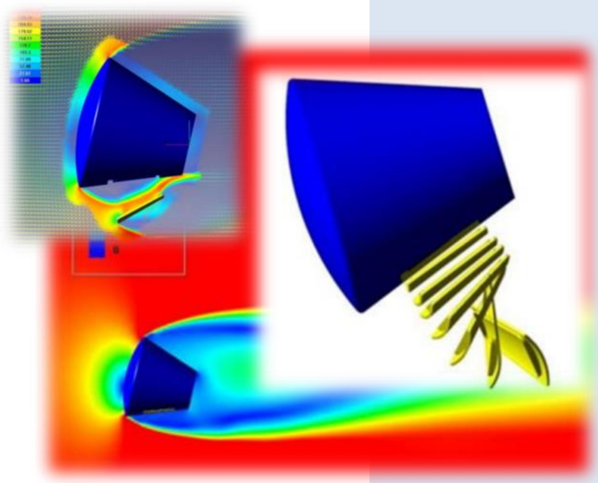

Figure 6: Moving Bodies based on fluid induced forces. Calculation of Emergency Escape Panel Release of a Spacecraft. (courtesy of Korolev Rocket and Space Corporation - Energia, Russia)

Figure 7: Aerodynamics of skating. The body motion is defined via motion capture.

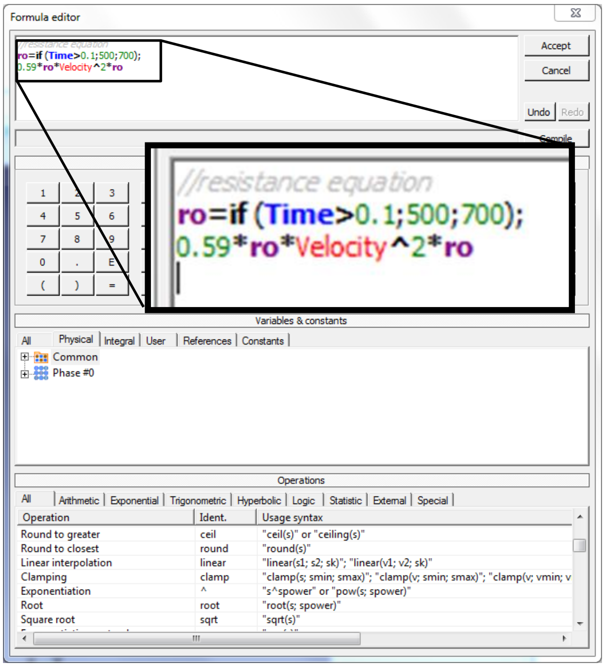

Figure 8: FlowVision Formula Editor.

No third party software for FSI – No accuracy loss: Meet the most accurate and the most flexible FSI solution of the world

Fluid Structure Interaction with *Seamlessly Coupled* FlowVision and SIMULIA/Abaqus

1. FlowVision grid generation is structured based on SGGR™ (explained above) technology which allows for **matched and accurate data exchange between CFD and FEA meshes**.

2. The **"wetted interface" is automatically established between fluid and solid domains**. It can involve **moving objects and moving boundaries with arbitrary complexity**.

3. The FlowVision Multi-Physics Manager (MPM) controls both FlowVision and SIMULIA Abaqus during the co-simulation process**. It manages data transfer, solver start, pause, restart and provides feedback on the simulation progress**.

4. The data is exchanged through the SIMULIA Co-Simulation Engine (CSE) which provides an optimal and accurate data transfer with parallel processing support.

5. Key Features:
   - MPM manager for simple **setup, start and restart of simulation and real-time results monitoring**
   - **Automatic coupling of not matched CFD & FEA meshes** and **automatic re-meshing** (not limited to mesh deformation)
   - Support of **highly deforming structures**, **moving bodies, moving boundaries and free surface tracking**
   - **Individual time step control for FEM and CFD** solvers
   - **Time step control of data exchange**
   - **Reduced simulation time** with parallel computing

6. FlowVision and Abaqus provide an optimized solution approach for highly coupled FSI calculations with ultimate accuracy. However, FlowVision can be easily coupled with other FEA tools as well.

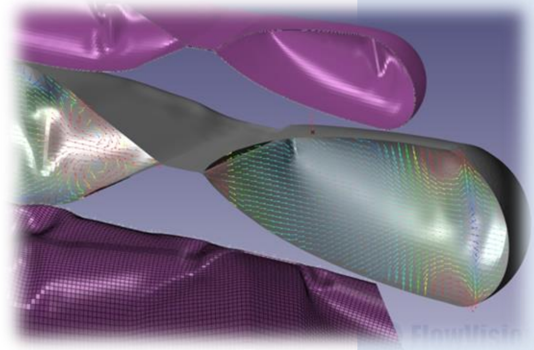

Figure 9: Pouch forming process in packaging process. (courtesy of Tetra Pak Italiana S.p.A.)

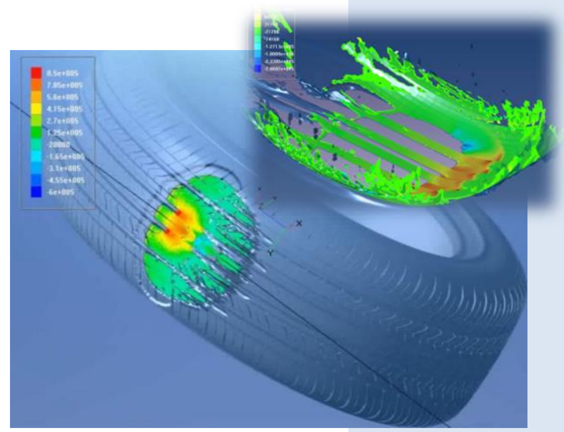

Figure 10: Tire Aquaplaning. (courtesy of Kenda Tires)

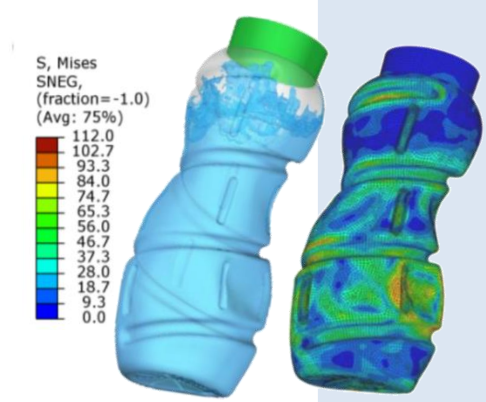

Figure 11: Bottle Drop Test

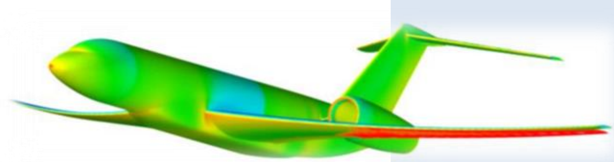

Figure 12: Aeroelasticity (courtesy of Ilyushin, Russia)

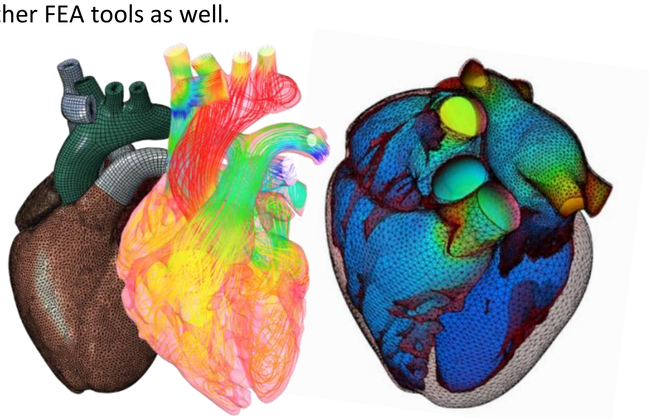

Figure 13 : FSI of personalized human heart models (The Living Heart Project)

FlowVision

Unique Capabilities

The best answer to dimensionality problems in CFD: Meet FlowVision's Unique Gap Model

FlowVision Gap Model for Dimensionality Issues & Clearance Flows

1. Thin clearances/gaps compared to the size of the flow domain exist in many CFD calculations. Resolving the flow phenomena within these clearances is crucial; however, results in tremendous computational costs.

2. FlowVision's unique gap model feature attacks this problem with **a sub-grid model to simulate the clearance flows using coarser computational grid configurations without sacrificing accuracy**.

3. The gap model was initially developed in 2005 to fulfill the demands of screw compressor design teams. **Today it is a validated, fast and accurate tool for general CFD problems including rotating machinery**.

4. Key Features:
    - User follows the normal locally adapted automatic meshing routine with Cartesian grid **(no need 3rd party meshing tools & excessively fine mesh due to dimensionality issues)**.
    - "Gap cells" are automatically detected based on the thresholds given by the user.
    - The model is automatically applied to the gap cells and governing equations are modified to accurately model the viscous forces.
    - **FlowVision's Gap Model is developed to resolve flows where dimension ratio increases up to $5 \cdot 10^7$. This corresponds to sub micron clearances in common turbomachine calculations.**

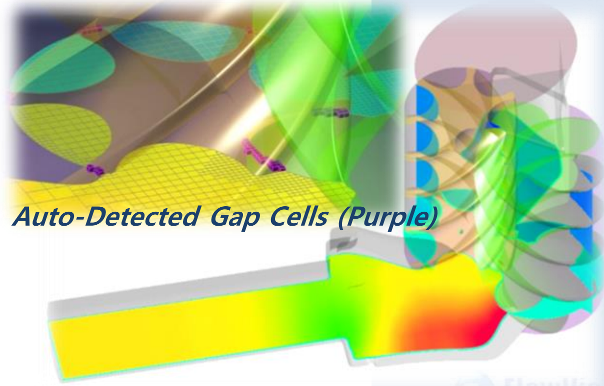

Figure 9: Screw compressor performance and FSI calculations

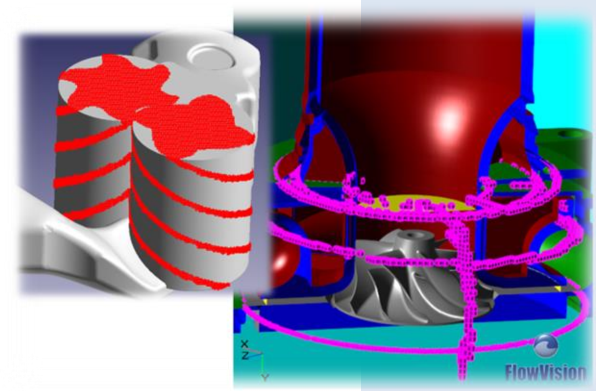

Figure 10 : Detected Gap Cells in a screw compressor (left) and turbocharger calculation (right). (courtesy of DSM, Mitsubishi Motors Corporation, USA)

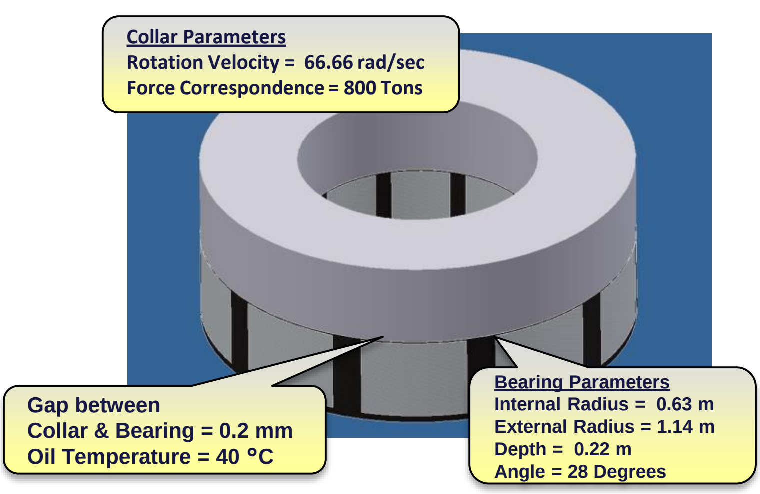

Figure 12 : Dimensionality issues in a thrust bearing FSI calculation. (courtesy of Taiwan Power Company, Taiwan)

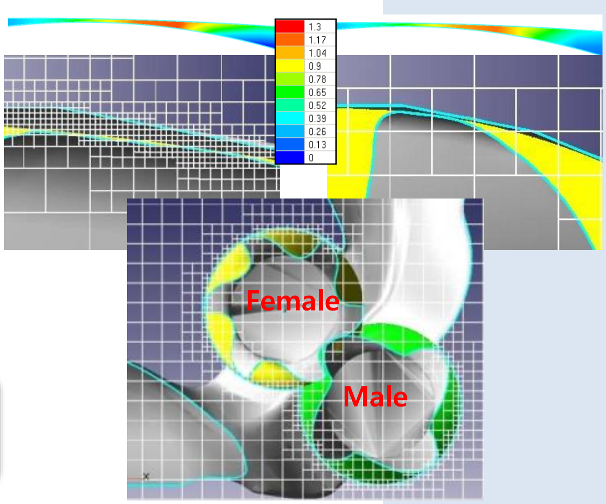

Figure 11 : Traditional approach vs FlowVision Gap Model

Reconstruction of free surfaces even at a sub-grid level and strict mass conservation, resulting in high accuracy.

Free Surface Tracking with FlowVision Advanced VOF Method

1. In the traditional approach, free surface regions are defined by VOF (volume of fluid) ratios ranging from 0 to 1. As a result the free surface is approximated based on the VOF value distribution. On the other hand, **FlowVision Advanced VOF Method accurately resolves the free surface topology even at a sub-grid (sub-element) level through cell splitting and reconstruction**. Together with the SGGR^{TM} technology, the **fluid mass is fully conserved in FlowVision calculations**, thus achieving ultimate accuracy.

2. Advanced VOF consists of cell splitting by free surface and reconstruction based on VOF function. Based on this approach, **FlowVision works with real free surfaces where boundary conditions between phases are specified.** In other words, FlowVision does not approximate a free surface region only based on VOF value (0-1). **Instead, the free surface is accurately reconstructed within every cell through SGGR^{TM} technology.**

3. In FlowVision Advanced VOF Method, if the VOF is bigger than zero for the regions which are geometrically away from free surface, **FlowVision captures these as droplets and calculates their movements through the computational domain. In following time steps, these droplets can join back to bulk flow resulting in accurate multi-phase flow simulations**.


Figure 14: Car water tightness

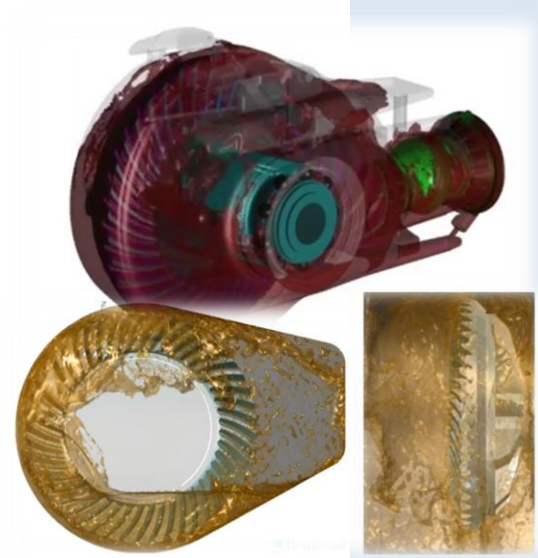
Figure 15: Lubricant behavior in mechanical parts.

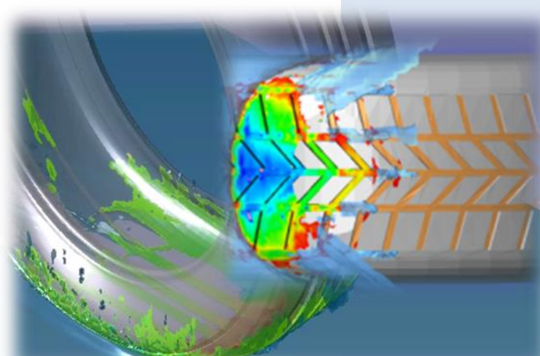
Figure 16: Tire aquaplaning

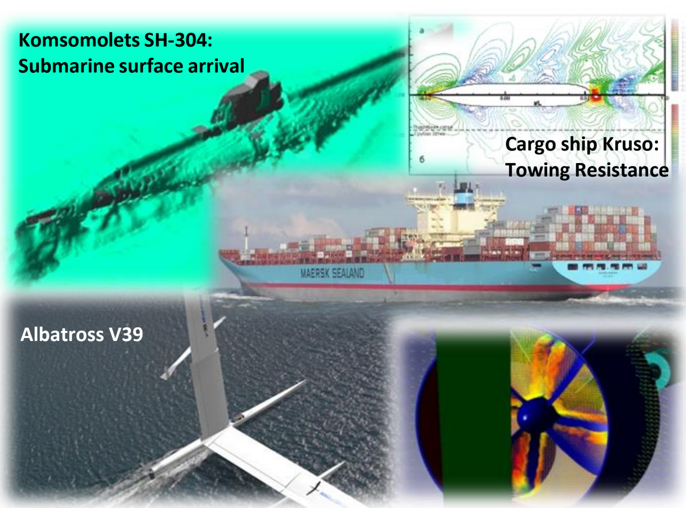

Figure 17: Marine Applications: Steady/unsteady behavior of watercraft, with and without the propulsion systems. Stability calculations in wave and wind, maneuverability through thrusters and auxiliary systems, dynamics of buoyancy at emergency flooding.

Extremely Powerful, Extremely Flexible

FlowVision Universal Solver

1. Flexible Solver Control: FlowVision provides the possibility to use **different solver configurations in different parts of the simulation domain** to obtain the most reliable simulation results and to minimize computational effort: For example, a user can simulate **one part of a domain with turbulence models while the remaining parts are modeled as laminar**. Or one can simulate **heat transfer with a time step larger than time step used to solve turbulence equations**.

2. One Solver for All Mach Numbers and Flow Conditions:
   - **Subsonic, transonic, supersonic and hypersonic flow regimes can simultaneously exist in the computational domain.**
   - **Users are not expected to manually address regions of different flow regimes and shock generation points.**
   - FlowVision's unique single numerical algorithm is capable of accurately resolving subsonic, transonic, supersonic and hypersonic. This avoids necessity of using different solvers in a solution.

3. High Level of Scalability: FlowVision solver is optimized to operate on large multiple-processor clusters and ensures highest level of accuracy and performance. When FlowVision is run on multiple processors, the **Dynamic Balance (see Fig. 20)** feature evenly distributes the computational workload to increase the overall calculation efficiency and impede single processor overload.

4. Flexible Configuration: FlowVision can operate under both Linux and Windows platforms. It offers the ultimate flexibility to the users and IT managers. Depending on the user preferences, only one solver can run on all available processors or multiple solvers can run in parallel through sharing available processors.

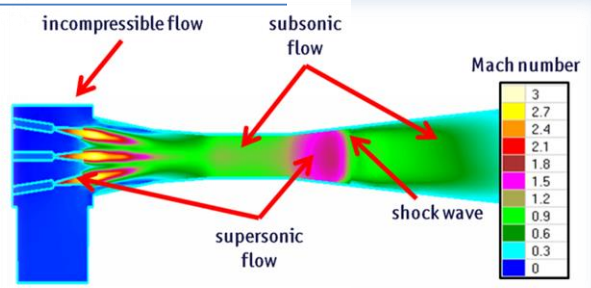

Figure 18: Single numerical algorithm applied throughout the entire computation domain. No user input required to address flow condition. In addition to this different solver settings (turbulence –on off, different heat transfer time step etc.) can be assigned to different parts of the flow domain.

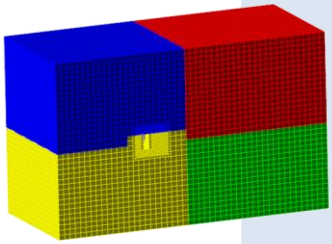

Figure 19: Overload of a single processor due to dynamic adaptations. Such cases are resolved by FlowVision Dynamic Balance.

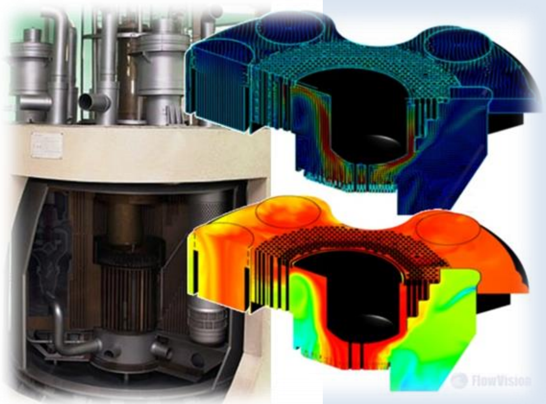

Figure 18: Upper mixing chamber of a fast neutron reactor BN-600 (courtesy of Afrikantov OKBM, Russia)

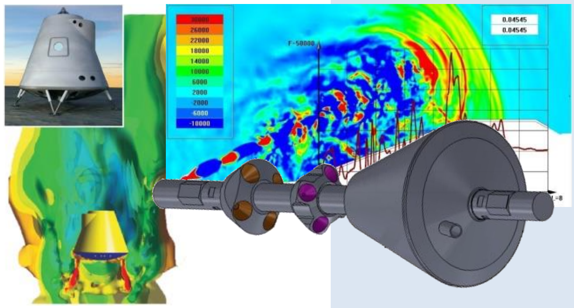

Figure 19: Spacecraft vertical landing (left), Shock Wave Interaction between Spacecraft and Emergency Rescue System (right). (courtesy of Korolev Rocket and Space Corporation - Energia, Russia)

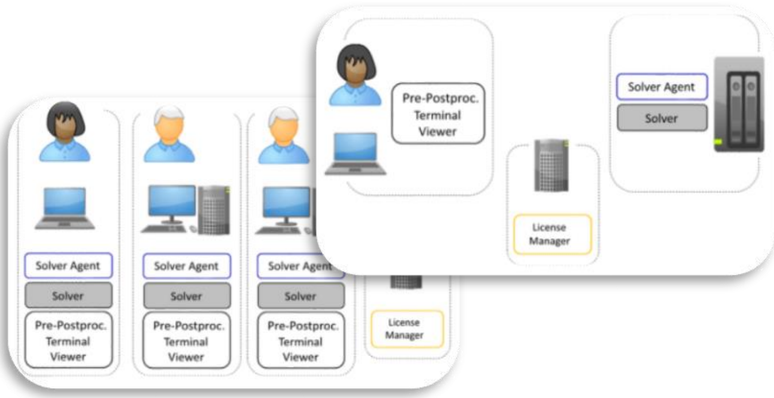

Figure 20: All FlowVision licenses are "floating" licenses, providing full flexibility for the deployment of different modules.

Notices:

This file may contain confidential or proprietary information and may be subject to export control laws and regulations. If you are not the intended recipient, you are notified that any dissemination, copying of this file or use of the contents by any means whatsoever is strictly prohibited. Unauthorized export or re-export is prohibited. If you have received this file in error, please advise the sender immediately and delete this file from your computer system.

Capvidia NV
Technologielaan 3 B-3001 Leuven Belgium
+32 16 40 27 47
+32 16 40 32 71
www.capvidia.com
info@capvidia.com

(주) 솔루션랩
(35260) 대전광역시 서구 둔산중로 20, 1203호
042 628 0789
042 623 0780
www.solution-lab.co.kr
info@solution-lab.co.kr