Thermo-fluid Analysis Software
V14 Product Guide

scSTREAM
HeatDesigner
scFLOW
SC/Tetra
PICLS
scPOST
The Role of CFD in Engineering

One of the foremost expectations of today’s successful product driven companies is that they bring high value-added products, that meet customer needs, quickly to the market. In addition, successful companies proactively identify application scenarios that could result in unsatisfactory performance, product failures, customer dissatisfaction and/or develop design solutions that mitigate the potential risks.

Thermo-fluid analysis software

Since software simulation enables predicting performance without creating a hardware prototype, the tools can be used early in the planning stage of product development to sift through preliminary design concepts. Simulation can also be used to predict performance of products where it is difficult to make experimental measurements. In addition, simulation software can be used to visualize invisible fluid flow and heat transfer. This results in increased engineering understanding while providing a vehicle for communicating this knowledge to non-experts.

Where does thermo-fluid analysis software come into play?

Thermo-fluid analysis software is indispensable for “Front-loading” product development to ensure the best product concepts that are identified early in the design process. Design quality will be improved during the conceptual design phase by conducting basic studies of fluid and thermal phenomena that directly affect product performance. During the detailed design phase, analyses are conducted under conditions similar to what the actual product will experience. From this work, design engineers can understand the source of problems that limit performance and investigate alternate design solutions before production begins.

Structured and unstructured mesh: the differences

Software Cradle offers two different types of thermo-fluid analysis tools: scSTREAM and HeatDesigner with structured mesh, scFLOW and SC/Tetra with unstructured mesh.

Structured mesh is simple and easy to construct. Structured mesh is comprised of many small cuboids so it can only approximate curved or angled surfaces with stair-case patterns. It is most useful for applications where tiny details and surface curvature or angles do not have a strong effect on the overall results. Examples of applications for structured mesh include electronics cooling, HVAC, and architecture.

Unstructured mesh is created using polyhedral elements. Mesh is generated such that it fits along the ridge lines of the original geometry. As a result, unstructured mesh is used for applications where precise representation of geometry is crucial. Examples of applications for unstructured mesh include vehicle aerodynamics, fan blade designs, and flows inside ducts.
Software Cradle develops and provides thermo-fluid simulation software and optional tools that suit various industries and objectives.

### Thermo-fluid simulation software and main peripheral tools

**scSTREAM**
- Structured mesh (Cartesian/cylindrical coordinate systems)
- Designing thermal and air flow inside an office
- Evaluating air flow around buildings
- Evaluating heat island effect
- Heat dissipation design of electronics and precision instruments
- Dust proof and moisture proof review of electronics and precision instruments
- Multi-phase flow analysis such as mixing, spray, solidification, melting, boiling, and condensation
- Analysis involving moving objects such as cars, controlled equipment, hydraulic and pneumatic equipment, and robots

**HeatDesigner**
- Module for electronics
- Designing heat release of a printed circuit board
- Examining heat-releasing fins and the material
- Designing heat release of an enclosure with a fan

**PICLS**
- Dedicated tool for thermal analysis of printed circuit boards
- Real-time thermal analysis
- Board size design
- Layer composition design
- Parts layout design
- Review of the effects of wiring pattern and thermal vias

**scPOST**
- Comprehensive and versatile data visualization software
- Obtaining numerical information
- Creating animation
- Mapping temperature information of a fluid analysis result to a structural analysis
- Comparing multiple analysis results

**scFLOW**
- Unstructured mesh (Polyhedral elements)
- Aerodynamic simulation for automobiles
- Evaluation of rotational devices such as fans and pumps
- Prediction of cavitation and erosion
- Design of household electric appliances such as refrigerators and wash machines
- Internal flow analysis of ducts and nozzle
- Analysis involving chemical reactions including reactor, catalyst, furnace, combustor, and CVD
- Multi-phase flow analysis such as mixing, spray, solidification, melting, boiling, and condensation
- Analysis of multiphase flow phenomena such as mixing, blending, sprays, solidification, melting, boiling, and condensation
- Water tank test simulation of a vessel

**SC/Tetra**
- Unstructured mesh (Tetrahedral, pentahedral, and hexahedral elements)

**WindTool**
- Wind environment assessment tool

**Launcher (Autodesk® Revit®)**
- CAD add-in tool

**Launcher (ARCHICAD)**
- CAD add-in tool

**Launcher (SOLIDWORKS®)**
- CAD add-in tool

**Optimus® for Cradle**
- Optional tool for optimum solution search

**ElectronicPartsMaker**
- Tool for semiconductor package modeling

**scWorkSketch**
- Tool for creating automated workflow

**Structural Analysis**
- Structural analysis tool (linear static analysis)

**SmartBlades**
- Tool for modeling fan blades

**Fluid-Structure Interaction (Abaqus®)**
- Tool for two-way coupling

**FluidBearingDesigner**
- Tool for analyzing fluid bearing

**1D/3D Coupling (GT-SUITE)**
- Tool for two-way coupling

---

Autodesk and Revit are registered trademarks of Autodesk, Inc. and/or its affiliates in the United States and other countries.
ARCHICAD is a registered international trademark of GRAPHISOFT R&D Rt.
Is your analysis tool useful in years to come?

scSTREAM and HeatDesigner have proven track records for incorporating the latest leading edge technology.

**scSTREAM HeatDesigner**

scSTREAM thermo-fluid software has serviced the electronics and architectural industries for more than thirty years. The ever-evolving software is characterized by its overwhelmingly user-friendly interfaces and high speed processing. HeatDesigner is based on scSTREAM and is specially developed for thermal design of electronics products. HeatDesigner provides physical functions required only for thermal design with its simple interfaces and powerful computing performance.

### Various methods to represent shapes

The shape of a model to be analyzed can be represented by using the following methods: voxel method (slanted faces and curved faces are represented in cuboids), cell method (the shape of a model created with a CAD tool can be represented more accurately), and finite element model method (a model of an arbitrary shape defined with unstructured mesh can be overlapped on a model defined with structured mesh to use the shape created with a CAD tool as is).

### Moving objects

A flow generated by a moving rigid object can be calculated. Conditions including the motions of an object (translation, rotation, and elastic deformation), heat generation/absorption, and air supply/return can be set. The model of a moving object is created on another mesh. In this way, conditions such as the distance that the object moves are limited very little.

### 6-degree-of-freedom motion (6DOF)

The function can analyze passive translation and rotation of an object receiving a fluid force. A moving object is assumed to be a rigid body. Its movement whose maximum degree of freedom is six (3D translation + 3D rotation) can be solved. The function can simulate driftwood which is flowed by a force from water flow.
Multiblock
Mesh can be refined partially to represent a model shape more accurately and perform a calculation more efficiently.

Discrete element method (DEM)
Multiphase analyses can be performed, which enables coupling of fluid analysis and flow analysis of particles.

Parts library
The shapes and conditions of frequently used parts can be registered. Conditions include the allocation position, material, and heat generation.

HeatPathView
The information on temperature of each part and a comprehensive amount of heat release obtained in post-processing of a general CFD analysis is not enough to know the heat path. HeatPathView displays heat paths and the amount of heat transfer in the whole computational domain in a diagram, a graph, and a table, allowing you to find the bottleneck of the heat paths easily.

ElectronicPartsMaker
The tool can create detailed models of semiconductor packages including QFP, SOP, and BGA by specifying parameters, and simplified models using thermal resistor models such as DELPHI models and two-resistor models. Manufacturers of semiconductor packages can provide the data of semiconductor packages as thermal resistor models without releasing the inside information.

Reading wiring patterns
To calculate heat transfer conditions depending on wiring patterns of a printed circuit board (PCB) in detail, the module can read Gerber data output from an electric CAD tool and import the data as a model for a thermo-fluid analysis. By using Gerber data, a more realistic calculation result can be obtained with the consideration of heat transfer affected by an uneven wiring pattern.

Radiation
Radiation heat transfer with the consideration of diffusion, reflection, transmission, refraction, and absorption can be calculated. VF (view factor) method and FLUX method\(^*1\) can be used. The lamp function can simulate radiant heat by a filament without detailed shape information of a lamp. In addition to the filament, laser beam and defective radiation specified by half-value angle can be used as a heat source model.

\(^*1\) Only for scSTREAM.
**BIM**

The software interface supports BIM 2.0. Autodesk® Revit® and GRAPHISOFT ARCHICAD have a direct interface (optional) through which a target part can be selected and the tree structure can be kept and simplified. In addition, the module can load files in IFC format, which is the BIM-standard format.

**Illuminance analysis**

The software can calculate illuminance of various types of light; for example, daylight through an opening of a building and artificial lighting with consideration of its directivity. Object surfaces such as walls are treated as diffusive reflection surfaces. In general, the larger an opening of a building is, the larger heat loss tends to be. By calculating the illuminance, the balance between heat and light can be examined collectively.

**Electronic part model**

A wide range of models are available that enable to easily achieve thermal design of printed circuit boards and electronic enclosures, which includes DELPHI (multi-resistor) model, Peltier device and heat pipes. It is possible to consider pressure loss characteristics using slits, and P-Q characteristics of fans using swirling component. Generated models can be added in library.

---

* Measurment device is not included
Air-conditioner parts (CFD parts)
The model shapes of parts frequently used for room air-conditioning can be imported. The models include ceiling cassettes, anemostat models, and linear diffusers. The software can import CFD part data, such as air supply characteristics, provided by SHASE. Various parameters can be set to simulate air-conditioning operation in addition to simple air heating and cooling.

Solar radiation (ASHRAE, NEDO)
Climate data published by ASHRAE and NEDO is preset and can be used for condition setting. By entering arbitrary values of longitude, latitude, date, and time, the solar altitude and the azimuth angle of the sun at a specified location and time are calculated automatically. The effect of solar radiation can be examined in detail. Various parameters including absorption and reflectivity of solar radiation and materials which transmit light diffusely, such as frosted glass, can be set.

Thermal comfort, heat stress risk and ventilation efficiency indices
Comfort indices PMV and SET* can be derived from already obtained temperature, humidity, and MRT (Mean Radiant Temperature), as one of result-processing functions. WBGT (heat stress risk indices), and the scale for ventilation efficiency (SVE), of which some indices can be converted to a real time, can be set by one click, and the range of calculation area can be selected (for example, either one of two rooms).

Humidity / dew condensation
The software can analyze humidity in the air. Dew condensation and evaporation on a wall surface due to temperature change can be considered and the amount of dew condensation and evaporation per time can be obtained. The software supports the analyses of moisture transfer inside a solid, and the function can be used to analyze a permeable object and dew condensation inside a part.

Plant canopy model (flow and heat)
Air resistance caused by plant canopy can be considered by setting the coefficient of friction and the leaf area density. For frequently used plants such as oak tree, their parameters are preset as the recommended values. The software also simulates the cooling effect by the latent heat of vaporization on a leaf surface by using the fixed temperature and setting the amount of absorbed heat. The function can be used for analyses of outdoor wind environment and heat island effect.

WindTool (outdoor wind environment assessment tool)
This tool helps assess outdoor wind environment. The assessment criteria can be selected from the ones proposed by Murakami et al. and by Wind Engineering Institute. By specifying a base shape and parameters required for wind environment evaluation, the parameters for 16 directions are calculated and the wind environment is ranked automatically. Detailed distributions of air current and pressure per direction can be visualized.
**Electrostatic field**

In addition to fluid force, the effect of an electrostatic field, which applies external force to charged particles, can be considered. By setting electric charge of particles and electric potential of a wall surface, the function can be used for analyses to consider area control of electrostatic coating. Velocity at which charged particles do not adhere on a wall surface can also be examined by using the function.

**Flow of foaming resin**

The software calculates the behavior of filling up an object with foaming resin, which is used as a heat insulator for houses and refrigerators. To examine speed and pressure of filling-up and the position for injecting the resin, the software simulates the behavior in 3D. The simulation can provide more pieces of information in shorter time than an actual measurement.

**Mapping**

When a target phenomenon is in a small range and the phenomenon is affected by a wide range of its surrounding area, analysis results of the surrounding area can be used for an analysis of the target phenomenon as boundary conditions to decrease the calculation load. To analyze only the inside of an enclosure for an electronic device highly affected by its outside, the analysis results of the outside can be used as boundary conditions.

**Free surface**

The software calculates the shape of an interface between a gas and a liquid. Either MARS or VOF method can be used, and the calculation target phase can be selected: both gas and liquid, only gas, or only liquid. The function is useful in a wide range of fields: from an analysis of tsunami in the civil engineering and construction field to an analysis of soldering in the electronic device field.
**Solidification / melting**

The phase change between fluid and solid, for example, water to ice and ice to water, can be considered. The following phenomena related to solidification/melting can be considered: change of flow affected by a solidified region, change of melting speed depending on the flow status, and latent heat at melting. A phenomenon that water in an ice maker becomes ice can be simulated using the function.

**Boiling / condensation**

(bubble nucleation, bubble growth / condensation)

With the function, the user can analyze a boiling flow, which is a gas-liquid two-phase flow caused by temperature difference between a liquid and a heat conduction surface. A boiling flow is analyzed as a free surface analysis using MARS method, and latent heat generation and volume change due to bubble growth / condensation are considered using phase change model.

**Particle tracking**

The software simulates the behavior of particles depending on their characteristics (diameter, density, and sedimentation speed) and action/reaction between particles and a fluid. This includes sedimentation due to gravity, inertial force for mass particles, and movement due to electrostatic force, liquefaction at adhering on a wall surface, evaporation and latent heat, the behavior as bubbles in a liquid for charged particles.

**Panel**

(heat conduction / transfer / thermal transport)

Material properties and motion conditions can be applied to a panel having no thickness in model, which allows for heat conduction to other parts and heat dissipation to air. This enables the simulations of paper feeding and film drying processes, where thin objects move and go under heating repetitively.

---

1. Transfer and thermal transport are only available on scSTREAM
## Functions (scSTREAM, HeatDesinger)

<table>
<thead>
<tr>
<th>Preprocessor/Solver</th>
<th>Mesh generation</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Conditions</th>
<th>Operation and control environment</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Flow types</th>
<th>Turbulence models</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Solving</th>
<th>Particle analysis</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Multiphase Flow analysis</th>
<th>Thermal reaction analysis</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Electric field analysis</th>
<th>Thermal source model</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Flow conditions</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Thermal conditions
- Fixed temperature
- Amount of heat generation
- Heat transfer coefficient
- Contact heat transfer coefficient
- Wall conditions
- No-slip (stationary wall)
- Slip (symmetry wall)
- Log-law condition
- Power-law condition
- Surface roughness

Porous media

Source conditions
- Volume force / pressure loss
- Heat source
- Smoke source (diffusing materials)
- Turbulence generation
- Humidification

User-defined conditions
- Volume force / pressure loss
- Heat source
- Smoke source (diffusing materials)
- Turbulence generation
- Humidification

Calculation control environment
- Monitoring the calculation status
- Automatic translation of the calculation

Output for third party software
- Abaqus, Nastran, Femtet, ADVENTURECluster, JMAG-Designer, Optimus, Light, modelRKTHER, Autodesk Revit, ARCHICAD, ThermoRender, EnSight, FieldView

Drawing functions
- Mesh, vector, contour plots
- Isosurface, streamline, pathline, volume rendering
- Geometry display
- Isoparms, Cylinder
- Surface roughness
- Special effects
- Oil flow (on plane / surface)
- Texture mapping (on plane / surface)
- Lighting, gradient, shadowing (preset / arbitrary)
- Arbitrary scaling
- Arbitrary pick
- Animation
- Vector animation
- Flow line animation
- Cut-plane sweeping
- Marker particle (turbulent diffusion effect)
- Automatic translation of viewpoint (view / focus points can be set)
- Key-frame animation
- Animation interpolated between cycles

Analysis results
- Variable registration (function registration)
- Integral (surface / volume)
- Comparison
- Projected area calculation
- Analysis results of mesh elements / subdomains
- Evaluation of flow visualization
- Evaluation of solid visualization

Data/image output
- Microsoft* BMP, JPEG, PNG
- CradleViewer® (support steady-state / transient animation, attach to Office applications)
- AVI, WMV
- VRML

Operation and control environment
- Selectable help function
- Spatial simulation
- Streaming
- Selectable mouse operation modes
- Virtual reality data by video

Functions (scSTREAM, HeatDesigner)

<table>
<thead>
<tr>
<th>Product</th>
<th>Compliant OS</th>
<th>Recommended environment</th>
<th>Approx. measure of analysis size</th>
<th>Compiler environment (User-defined function)</th>
</tr>
</thead>
</table>

System Configuration

Windows is a trademark of Microsoft Corporation in the United States and other countries.
The official name of Windows is the “Microsoft® Windows® Operating System”.
Microsoft Visual Studio is a registered trademark of Microsoft Corporation in the United States and other countries.
Linux is a trademark of Linus Torvalds in the United States and other countries.
Intel is a registered trademark of Intel Corporation in the United States and other countries.
Red Hat is a registered trademark of Red Hat, Inc. in the United States and other countries.
SUSE is a registered trademark of SUSE LLC.
All other product and service names mentioned are registered trademarks or trademarks of their respective companies.

* Windows version
* Intel Parallel Studio XE 2018 Composer Edition for Fortran
* Intel Parallel Studio XE 2017 Composer Edition for Fortran
* Intel Parallel Studio XE 2016 Composer Edition for Fortran
* Linux version
* GFortran (GNU Fortran compiler)
* (Linux standard)
The ever-evolving latest CFD solution
Discover what you want from your CFD tool here

**scFLOW SC/Tetra**

SC/Tetra has been characterized by sophisticated mesh generation function, high speed computing capability, and user-friendly features throughout the operation. As its advanced version, scFLOW has been released. It is equipped with more stable Solver that achieves calculation speed three times faster (at maximum) than before, and new Preprocessor that helps entry-level users build complicated models and high-quality mesh. scFLOW, the new generation software, keeps on evolving.

**Simplification of Preprocessor operations**
From the CAD data to analysis mesh data, the required operations are grossly simplified compared to before. The conservation of assembly information and the settings of conditions on the parts bring the sense of continuity from the CAD operations and reduce the operational burden of the users.

**Polyhedral mesher**
Using polyhedral mesh elements improves stability and calculation accuracy of cell-centered solver. In scFLOWpre, mesh can be generated according to the target number of mesh elements and automatically refined near wall area. The automatic mesher function also enables users to specify mesh refinement level of each part and region.

**Modifying CAD data**
When CAD data to be used for simulation has a problem, the data can be modified with Preprocessor. Boundary conditions can be set based on the part names and color information set in the CAD data. When some regions are missing in the model, shapes such as cuboids and cylinders can be added.

**Viewer mode**
Preprocessor data can be displayed in the viewer mode without the Pre-/Post-processor license, when the license is taken by the mesher or by Postprocessor and is unavailable.
Mesh-adaptation analysis
With this function, mesh will be automatically refined where a flow or pressure changes greatly in a steady-state analysis. After the calculation in Solver is completed, Preprocessor automatically launches and executes gridding and meshing based on the calculation result. By specifying the target number of elements, coarse mesh is generated first and the mesh is automatically refined to be appropriate for the calculation. The function is useful for an analysis of flows in a tube with a complicated shape.

Discontinuous mesh
Flow with object motion can be calculated, including rotation of fans and turbines, and crossing travel of automobiles or trains (translation). The function enables an analysis with consideration on shear heating between rotor and pad in a disk brake. The function also makes it possible to analyze a combination of rotation and translation such as a piston pump.

Free surface (steady-state / transient)
The shape of an interface between a gas and a liquid can be simulated. Calculations by VOF method (new method: FIRM) are fast and accurate, and functions including moving boundary, overset mesh, and particle tracking can be used in combination. Because a phenomenon where the phase interface becomes stable can be analyzed in a steady-state calculation, the result can be obtained in a shorter time than before.

Overset mesh
Free movement of regions, that cannot be analyzed using existing functions such as stretching or rotating elements, can now be simulated by overlapping mesh elements for stationary and moving regions. This function supports an overlap of multiple moving regions, a contact between objects, and a 6-degree-of-freedom motion of rigid bodies. This is useful to analyze opening and closing of a valve of an engine port or a gear pump where gears engage with each other.

Stabilization of calculation
Even for mesh data with elements of extremely low quality, the calculation can be stabilized by the automatic processing to avoid divergence. This function helps Solver be more robust.

6-degree-of-freedom motion (6DOF)
Passive translation and rotation of a rigid body receiving a fluid force can be analyzed. With the function, the user can analyze a ball valve with consideration of the elasticity of the spring (1D translation), and paper airplane with consideration of 6-degree-of-freedom rigid-body motion (3D translation + 3D rotation). In addition, the function is applied to analyses of check valves, wind power generators, and blades of wave power generators.

Only scFLOW supports FIRM. FIRM cannot be used for overset mesh or steady-state analyses.
Fluid-structure interaction
This option is used for two-way FSI (fluid-structure interaction) with structural analysis software. With this option, not only rigid bodies but also elastic bodies can be treated. Deformation of an object caused by a fluid force and the change of fluid caused by the deformation can be analyzed.

Compressible fluid
The software can analyze phenomena such as supersonic flow and significant expansion/contraction of volume. For a compressible fluid, both the pressure-based and the density-based Solvers can be used. The density-based Solver keeps the calculation stable even with high Mach number. You can select either Solver depending on the analysis target and phenomenon.

Cavitation
This function enables simulation of a vaporization phenomenon called cavitation, which is caused at an area where pressure of a liquid becomes lower than in the surrounding area, such as with a propeller rotating at a high speed under water. The occurrence of cavitation can be predicted by applying the cavitation model based on the pressure values. The software also supports problems caused by cavitation such as erosion.

Aerodynamic noise analysis
Sound caused by pressure oscillation of a fluid, such as wind noise, and sound caused by resonance can be predicted. The calculation can be performed accurately by using LES and the weak compressible flow model. The frequency of aerodynamic noise can also be analyzed using the Fast Fourier Transform (FFT) method from the CFD analysis result.

Evaporation/Condensation
Free surface analysis function (VOF method) of this software can simulate phase change between gas and liquid, such as evaporation and condensation. By considering phase change, not only simple heat conduction but also heat transfer from latent heat can be calculated. For example, this method can be applied to internal flow simulations for heat transfer devices such as heat pipes, in which a refrigerant liquid changes to vapor by absorbing heat from an outer region.

Dispersed multi-phase flow
This function can simulate flows containing many bubbles, droplets, or particles (dispersed phase), which are difficult to be analyzed using free surface. This function is a multi-fluid model that can predict volume fraction distribution and velocity distribution of each phase by solving the governing equation under the assumption that the dispersed phase is a fluid (continuous phase). The function is useful to analyze the bubble jet effect and aeration tanks.
Particle tracking
Particle tracking function enables analyzing behavior of particles in flow. When analyzing small particles that follow the fluids movement (such as steam and dust), marker particle function can be used to evaluate particles in flow that change over time, which assumes that particle movement is in accordance with fluid velocity.

Humidity dew condensation
The amount of dew condensation on an object surface can be calculated from the surface temperature and water vapor in the air. You can output the amount of dew condensation per unit time in a steady-state analysis and the accumulated dew condensation in a transient analysis. Evaporation from a surface where dew condensation occurs can be calculated simultaneously, and this is useful for an analysis of a windshield defroster.

Liquid film model
The liquid film model is an extended function of the particle tracking function. By using the model, you the user can simulate the phenomenon that liquid particles change to a liquid film (water on a wall) when they reaching on the a wall. A liquid film on a wall flows with the influence of gravity and a gas-phase flowdown depending on an angle of the wall and collects in at a certain position. The analysis results are output as the thickness of a liquid film.

Thermoregulation-model (JOS)
Combination use of the thermoregulation-model (JOS) and a fluid analysis enables analyses of the surface temperature of a human body under a certain thermal environment. It can also be used to analyze temperature and humidity changes in the surrounding environment of a human body. The user can consider age, clothes, and physiological phenomena of the human body such as heat transfer by blood flow in addition to the surrounding environment of a human body such as temperature and velocity.

LES
LES is one of the turbulent flow models. It models eddies smaller than the mesh element in size and directly calculates other eddies. Although calculation load is large, LES enables simulations closer to real phenomena. LES is often used in noise analyses, significantly affected by time variation, to simulate the behavior of small eddies. The user can use the hybrid model with RANS, a turbulent model of small calculation load.

Radiation
Heat transfer by infrared-ray radiation can be considered by setting emissivity and temperature difference between objects. The user can choose VF (view factor) method or FLUX method as a calculation method. The user can also consider wavelength dependence, transmission, absorption, refraction, diffusion, and reflection of radiation. In FLUX method, the user can also consider directionality.
Mapping
When a target phenomenon is in a small range and the phenomenon is affected by a wide range of its surrounding area, analysis results of the surrounding area can be used for an analysis of the target phenomenon as boundary conditions to decrease the calculation load.

Coupled analysis with GT-SUITE
Coupled analysis with GT-SUITE is available. The entire flow in an intake and exhaust system is calculated with GT-SUITE and small flows of each part are interpolated with scFLOW or SC/Tetra. This will enhance calculation accuracy of the whole system.

Operation logging by VB interface
The operations in Preprocessor can be saved as a log file using the VB interface. Making the user scripting unnecessary, this makes the construction of an automated system affordable in a short period of time based on the files storing the operation logs.

Fan model (rotating blades)
With this model, an average flow field around rotating blades can be simulated only by entering characteristic properties regardless of real shapes of fans or propellers. The user can use the non-dimensional swirl coefficient model, the simplified propeller model, and the simplified rotor model. This model is useful to analyze axial-flow windmills and waterwheels.
Script functions
Before, complicated settings, including time- and coordinate-dependent material properties or boundary conditions, required a coding and compilation of user-defined function in C language. With the script functions, compilation is not required. Functions can be written in Preprocessor based on JavaScript.

SmartBlades
This function is useful for analyzing the shape of a fan automatically throughout creating the shape of a fan (CAD data), calculating the flow, and post-processing. The shape of a fan can be created easily by specifying parameters including the number of blades, fan diameter, rake angle, and skew angle.

FluidBearingDesigner
The function creates groove patterns of fluid bearings (dynamic-pressure bearing) and generates mesh. You can select the shape of grooves such as journal and thrust and materials such as porous material. From calculation results, you can obtain parameters for designing fluid bearings such as axial force and drag coefficient.
<table>
<thead>
<tr>
<th>Category</th>
<th>scFLOW</th>
<th>SC/Tetra</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Modeling</strong></td>
<td>CAD data interface (import)</td>
<td>Parasolid,STEP,IGES,ACIS,CATIA/V,Creo/SolidWorks,Stratasys,Pro/Engineer,CADENCE,Tera,STL,Parasolid,Amesim,OAEM,Creo/DesignModeler,Fortran,MAPS,HiL,Makarsa,OAEM,Parasolid,MAK</td>
</tr>
<tr>
<td></td>
<td>Preprocessor</td>
<td>Preprocessor</td>
</tr>
<tr>
<td></td>
<td>Solver</td>
<td>Solver</td>
</tr>
<tr>
<td><strong>Mesh generation</strong></td>
<td>Preprocessing</td>
<td>Preprocessing</td>
</tr>
<tr>
<td></td>
<td>Postprocessing</td>
<td>Postprocessing</td>
</tr>
<tr>
<td></td>
<td>Visualization</td>
<td>Visualization</td>
</tr>
<tr>
<td></td>
<td>Control environment</td>
<td>Control environment</td>
</tr>
<tr>
<td><strong>Conditions</strong></td>
<td>Flow types</td>
<td>Flow types</td>
</tr>
<tr>
<td></td>
<td>Fluid properties</td>
<td>Fluid properties</td>
</tr>
<tr>
<td></td>
<td>Mixture analysis</td>
<td>Mixture analysis</td>
</tr>
<tr>
<td></td>
<td>Dispersion</td>
<td>Dispersion</td>
</tr>
<tr>
<td></td>
<td>Mixing plane</td>
<td>Mixing plane</td>
</tr>
<tr>
<td><strong>Operation and control environment</strong></td>
<td>Selected mouse operation modes</td>
<td>Selected mouse operation modes</td>
</tr>
<tr>
<td></td>
<td>Filtering</td>
<td>Filtering</td>
</tr>
<tr>
<td><strong>Mesh</strong></td>
<td>Structured mesh</td>
<td>Structured mesh</td>
</tr>
<tr>
<td></td>
<td>Hybrid mesh</td>
<td>Hybrid mesh</td>
</tr>
<tr>
<td></td>
<td>Moving mesh</td>
<td>Moving mesh</td>
</tr>
<tr>
<td><strong>Numerical scheme</strong></td>
<td>Convergence term accuracy</td>
<td>Convergence term accuracy</td>
</tr>
<tr>
<td></td>
<td>Iteration scheme</td>
<td>Iteration scheme</td>
</tr>
<tr>
<td></td>
<td>Boundary condition</td>
<td>Boundary condition</td>
</tr>
<tr>
<td></td>
<td>Flow type</td>
<td>Flow type</td>
</tr>
<tr>
<td></td>
<td>Heat radiation</td>
<td>Heat radiation</td>
</tr>
<tr>
<td><strong>Turbulence models</strong></td>
<td>Chen-Othmer model</td>
<td>Chen-Othmer model</td>
</tr>
<tr>
<td></td>
<td>Realizable k-ε model</td>
<td>Realizable k-ε model</td>
</tr>
<tr>
<td></td>
<td>Spalart-Allmaras model</td>
<td>Spalart-Allmaras model</td>
</tr>
<tr>
<td></td>
<td>0-equation model</td>
<td>0-equation model</td>
</tr>
<tr>
<td></td>
<td>LES, DES, VLES</td>
<td>LES, DES, VLES</td>
</tr>
<tr>
<td><strong>Thermal analysis</strong></td>
<td>Heat conduction (fluid model)</td>
<td>Heat conduction (fluid model)</td>
</tr>
<tr>
<td></td>
<td>Convection heat transfer</td>
<td>Convection heat transfer</td>
</tr>
<tr>
<td></td>
<td>Heat conduction (solid model)</td>
<td>Heat conduction (solid model)</td>
</tr>
<tr>
<td></td>
<td>Local radiation</td>
<td>Local radiation</td>
</tr>
<tr>
<td><strong>Particle analysis</strong></td>
<td>Particle properties</td>
<td>Particle properties</td>
</tr>
<tr>
<td></td>
<td>Diffusion</td>
<td>Diffusion</td>
</tr>
<tr>
<td></td>
<td>Converting fluid model</td>
<td>Converting fluid model</td>
</tr>
<tr>
<td><strong>Reactor analysis</strong></td>
<td>Reaction kinetics</td>
<td>Reaction kinetics</td>
</tr>
<tr>
<td></td>
<td>Chemical reaction</td>
<td>Chemical reaction</td>
</tr>
<tr>
<td><strong>Multi-phase flow analysis</strong></td>
<td>Inter-particle interaction model</td>
<td>Inter-particle interaction model</td>
</tr>
<tr>
<td></td>
<td>Inter-particle interaction</td>
<td>Inter-particle interaction model</td>
</tr>
<tr>
<td></td>
<td>Inter-particle interaction model (VOF method, steady state)</td>
<td>Inter-particle interaction model (VOF method, steady state)</td>
</tr>
<tr>
<td><strong>Aerodynamic noise analysis</strong></td>
<td>Inter-particle interaction model (VOF method, transient)</td>
<td>Inter-particle interaction model (VOF method, transient)</td>
</tr>
<tr>
<td><strong>Current analysis</strong></td>
<td>Current density</td>
<td>Current density</td>
</tr>
<tr>
<td><strong>Thermo-regulation model</strong></td>
<td>CON, CON-2</td>
<td>CON, CON-2</td>
</tr>
<tr>
<td><strong>Flow conditions</strong></td>
<td>Viscosity</td>
<td>Viscosity</td>
</tr>
<tr>
<td></td>
<td>Density</td>
<td>Density</td>
</tr>
<tr>
<td></td>
<td>Volume flow rate</td>
<td>Volume flow rate</td>
</tr>
<tr>
<td><strong>Thermal conditions</strong></td>
<td>Amount of heat generation</td>
<td>Amount of heat generation</td>
</tr>
<tr>
<td></td>
<td>Heat transfer coefficient</td>
<td>Heat transfer coefficient</td>
</tr>
<tr>
<td><strong>Wall conditions</strong></td>
<td>Contact heat transfer coefficient</td>
<td>Contact heat transfer coefficient</td>
</tr>
<tr>
<td></td>
<td>Specified heat transfer coefficient</td>
<td>Specified heat transfer coefficient</td>
</tr>
<tr>
<td></td>
<td>Interface heat transfer</td>
<td>Interface heat transfer</td>
</tr>
<tr>
<td></td>
<td>Interface heat transfer (fluid-to-fluid)</td>
<td>Interface heat transfer (fluid-to-fluid)</td>
</tr>
<tr>
<td></td>
<td>Specified heat transfer (solid-to-solid)</td>
<td>Specified heat transfer (solid-to-solid)</td>
</tr>
<tr>
<td></td>
<td>Specified heat transfer (fluid-to-solid)</td>
<td>Specified heat transfer (fluid-to-solid)</td>
</tr>
<tr>
<td></td>
<td>Specified heat transfer (solid-to-fluid)</td>
<td>Specified heat transfer (solid-to-fluid)</td>
</tr>
</tbody>
</table>
Functions (scFLOW, SC/Tetra)

### Solver

| Pressure conditions | Fixed pressure | Pressure loss | Pressure loss
|---------------------|----------------|--------------|--------------|
| Source conditions   | Volume force / pressure loss | Heat generation | Smoke source (diff using materials)
|                     | Turbulence generation | Solid shear heating | Simplified propeller model
| User-defined conditions | Simplified rotor model | User-defined function (compilation required) | User-defined conditions

### Postprocessor

| Drawing functions | Mesh, vector, contour plots | Surface rendering | Volume rendering
|                   | Isosurface, streamline, pathlines, volume rendering | Geometry display | 2D graph

### Calculation control environment

- Monitoring the calculation status
- Email notification of the calculation
- Job management
- VB interface

### Output for third party software

- Abaqus, Nastran, Femtet, ANSYS Design
- ADVENTURE
designer
- LMS Virtual.Lab, Actran, FlowNoise
- OPTIMUS, Isight, modeFRONTIER
- LOGE, LMS Virtual.Lab, Actran, FlowNoise
- EnSight, FieldView, AVS

### Drawing position/orientation

- Arbitrary plane, surface, entire volume, cylinder
- Pathlines
- Arbitrary scaling
- Arbitrary pick

### Special effects

- Texture mapping
- Lighting, luster, gradation

### Animation

- Vector animation
- Flow line animation
- Cut-plane weeping
- Marker particle (turbulent diffusion effect)
- Automatic translation of view points
- Key-frame animation
- Analysis interpolated between cycles
- Velocity path animation
- Customize position / orientation

### Analysis results

- Scalable rendering
- Hexagonal data
- Hexagonal data
- Special effects
- Analysis interpolated between cycles
- Special effects
- Analysis interpolated between cycles
- Special effects

### Discharge output

- CRW
- Avi, WMV
- VRML

### Operation and control environment

- Selectable help function
- OpenGL emulation
- Stereoscopic view (side by side)
- Arbitrary plane, surface, entire volume, cylinder
- Streamlines, isosurfaces
- Pathlines
- Arbitrary scaling
- Arbitrary pick

### System Configuration

<table>
<thead>
<tr>
<th>Product</th>
<th>Compliant OS</th>
<th>Recommended environment</th>
<th>Approx. measure of analysis size</th>
<th>Compiler environment (User-defined function)</th>
</tr>
</thead>
</table>

Windows is a registered trademark of Microsoft Corporation in the United States and other countries.

The official name of Windows is the “Microsoft® Windows® Operating System”.

Microsoft Visual Studio is a registered trademark of Microsoft Corporation in the United States and other countries.

Linus is a trademark registered to Linus Torvalds in the United States and other countries.

Red Hat is a registered trademark of Red Hat, Inc. in the United States and other countries.

SUSE is a registered trademark of SUSE LLC.

All other product and service names mentioned are registered trademarks or trademarks of their respective companies.

© Fluid-Structure Interaction (Abaqus®) is not supported for HPC pack.

© Only compliant with partial functions in Solver, Monitor and Preprocessor.
Visualize your multiphysics phenomena in one environment
Postprocessor regularly installed in Software Cradle products can be purchased separately

**scPOST**

**Postprocessor**

In Postprocessor, you can visualize the simulation results calculated in Solver. It is effective for product design reviews because in Postprocessor, you can check, for example, temperature distribution at the places that cannot be measured or observed in the actual products. You can output not only still images but also animations, as well as output files for CradleViewer.

**Characteristics**

You can:
- Obtain numerical information with simple operation.
- Create beautiful animation quickly in Postprocessor.
- Easily map temperature information obtained in a fluid analysis to a structural analysis.
- Easily compare multiple analysis results.
- Easily calculate heat transfer and grasp a whole of heat-related matters.
- Output images supporting VR.

^Output in the equirectangular format with parallax

**Useful functions (example)**

- Creates animation automatically
- Saves display status
- Develops the image on the meridian plane
- Compares results
- Calculates (integral, registering functions)

**Compatible formats for import/export**

<table>
<thead>
<tr>
<th>Formats supported</th>
<th>Other format</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Import</strong></td>
<td><strong>Export</strong></td>
</tr>
<tr>
<td>• MSC Nastran2018 (.h5)</td>
<td>• Generic format for fluid data (.cgns)</td>
</tr>
<tr>
<td>• MSC Marc 2018 (119, 116)</td>
<td>[Only ADF can be exported through scCONVERTER)</td>
</tr>
<tr>
<td>• Generic format for fluid data (.cgns)</td>
<td>• Images (BMP, PNG)</td>
</tr>
<tr>
<td>[ADF only]</td>
<td>• 3D geometry data (STL, OBJ)</td>
</tr>
<tr>
<td></td>
<td>• Generic format for fluid data (.cgns)</td>
</tr>
<tr>
<td></td>
<td>[Only ADF can be exported through scCONVERTER]</td>
</tr>
<tr>
<td></td>
<td>• Images (BMP, JPEG, PNG)</td>
</tr>
<tr>
<td></td>
<td>• Animation (AVI, WMV)</td>
</tr>
<tr>
<td></td>
<td>• 3D geometry data (STL, VRML, FBX)</td>
</tr>
</tbody>
</table>
Co-simulation with MSC Software products
Integration of multidisciplinary analyses – from materials to systems

More realistic coupled fluid – mechanical – structural analyses
Capturing movement and deformation more precisely and expressing boundary conditions in fluid analyses with more reality

Co-simulation platform
The platform for coupled analyses with MSC mechanical and structural analysis solvers provides seamless co-simulation.

Co-simulation using FMI
Co-simulation using FMI a tool independent standard of 1D co-simulation interface

Co-simulation with Actran, acoustic analysis software
scFLOW and SC/Tetra are used to create fluid sound sources and Actran is used for propagation analysis of sound waves.

Analysis of an exhaust tube of a motorcycle
Analysis of an axial-flow fan

Acoustic analysis using fluid analysis results as a sound source

Compared with direct solution only by fluid analysis software, solution can be obtained with dramatically less calculation load.
Wow! Was it this easy?!
Non-experts can start thermal analysis right away with easy operation in 2D and real-time results

PICLS is a thermal simulation tool which helps designers easily perform thermal simulation of PCBs. Even if you are unfamiliar with thermal simulation, you will obtain a simulation result without stress through the tool’s easy and quick operation in 2D. You can import the data of a PCB created in PICLS to scSTREAM and HeatDesigner, that is, you can pass the analysis data seamlessly from the PCB design stage to the mechanical design stage.

Advantages
- Easy to use
  (Operation in 2D, integrated GUI for pre- and post-processing)
- Inexpensive
- Capable of real-time analysis

Thermal countermeasures using PICLS
- Checking the layout of components to avoid interference of heat between them
- Troubleshooting thermal issues of current products
- Considering heat release depending on a wiring pattern (coverage ratio)
- Examining the location and the number of thermal vias
- Examining the performance of a heatsink
- Examining the size of a PCB
- Considering natural/forced air cooling
- Considering radiant heat
- Considering heatsinks (number of fins, size)
- Examining heat dissipation performances by connection to enclosure
- Considering PCB mounting environment

Functions available in PICLS and PICLS Lite
- Multiple layers
- 3D preview
- Real-time display
- Radiation
- IDF3.0 interface
- Library
- Wiring area specification
- Displaying each layer
- Automatic report output
- Contact thermal resistance
- Considering a heatsink
- Wiring data (Gerber) import
- Thermal via
- Cutting out a PCB
- Forced air cooling
- Temperature margin, alert function
- Consideration of simple enclosure
- Drill data import

* PICLS Lite is provided online

http://www.cradle-cfd.com/picls/
Main features of PICLS and PICLS Lite

**Modeling**
- **External file interface**
  - You can import IDF 3.0 and Gerber data

- **Heatsink**
  - You can allocate and display parts such as plate fins and heat dissipation plates

- **Library**
  - You can register and reuse created parts to the library

- **Preview**
  - You can check the layout of components in the 3D image

- **Cutting out a PCB**
  - You can create a PCB of arbitrary shape using cut-out function

**Calculation and Post-Processing**
- **Real-time display**
  - The translation of components is displayed in real time

- **Report output**
  - You can output analysis results as reports

- **Alert function**
  - You can check parts whose temperature is higher than threshold

**System Configuration**

<table>
<thead>
<tr>
<th>Compliant OS</th>
<th>Recommended environment</th>
</tr>
</thead>
<tbody>
<tr>
<td>Windows 10</td>
<td>[Memory] 2.0 GB or more</td>
</tr>
<tr>
<td>Windows 8.1</td>
<td>[Hard disk] 0.5 GB or more free capacity recommended</td>
</tr>
<tr>
<td>Windows 7</td>
<td>[Display resolution] 1920 x 1080 or more</td>
</tr>
<tr>
<td>Windows Server 2016</td>
<td></td>
</tr>
<tr>
<td>Windows Server 2012 R2</td>
<td></td>
</tr>
<tr>
<td>Windows Server 2012</td>
<td></td>
</tr>
<tr>
<td>Windows Server 2008 R2</td>
<td></td>
</tr>
<tr>
<td>Red Hat Enterprise Linux 7</td>
<td></td>
</tr>
<tr>
<td>Red Hat Enterprise Linux 6 (6.1 onward)</td>
<td></td>
</tr>
<tr>
<td>SUSE Linux Enterprise Server 12</td>
<td></td>
</tr>
<tr>
<td>SUSE Linux Enterprise Server 11 (SP3 onward)</td>
<td></td>
</tr>
</tbody>
</table>

*1 Supports license manager only.
Analysis Procedure
− scSTREAM (HeatDesigner), scFLOW and SC/Tetra

There are three major steps in the workflow for obtaining simulation results.

**STEP.1 Preprocessor**

With Preprocessor, create or import analysis models, set analysis conditions, and generate mesh.

**STEP.2 Solver**

Flow/thermal calculations are performed using input data created in the Preprocessor. During the computation, calculation status can be monitored. The amount of time required for the computation depends on the size of the model (number of mesh elements), quality of the model, and hardware. A parallel Solver is available for reducing the computational time of large-scale models.

* Solver features (examples)
  - Setting the degree of parallelism
  - Monitoring job status
  - Visualizing results in real-time

**STEP.3 Postprocessor**

The Solver outputs field data for visualization using the Postprocessor. This permits examining flow, temperature, pressure, and other analysis results. Visualized results can be converted to images, animations and/or CradleViewer (details on P23) files for later use.

* Various drawing functions
  - Vector plot
  - Isosurface
  - Contour map
  - Pathline (available only in SC/Tetra)
  - Streamline
  - Oil flow
  - Still image and animation output

* See page 25 (HPC Solution) for more information about parallel calculation
Main Mutual Features

CAD Interface
Software Cradle analysis software can import native data from major 3D CAD software as well as import most generalized intermediate data formats. This eliminates the cumbersome process of data conversion.

### CAD / geometry data
<table>
<thead>
<tr>
<th>Compliant software</th>
<th>Format</th>
<th>V14 compliant versions</th>
</tr>
</thead>
<tbody>
<tr>
<td>CATIA V5</td>
<td>ST</td>
<td>HD, FLOW, SCT</td>
</tr>
<tr>
<td>CATIA V4</td>
<td>ST</td>
<td>HD, FLOW, SCT</td>
</tr>
<tr>
<td>Creo Elements/Pro/Pro/E</td>
<td>ST</td>
<td>HD, FLOW, SCT</td>
</tr>
<tr>
<td>SOLIDWORKS</td>
<td>ST</td>
<td>HD, FLOW, SCT</td>
</tr>
<tr>
<td>ug nx</td>
<td>ST</td>
<td>HD, FLOW, SCT</td>
</tr>
<tr>
<td>SolidEdge</td>
<td>ST</td>
<td>HD, FLOW, SCT</td>
</tr>
<tr>
<td>Autodesk® Inventor*</td>
<td>ST</td>
<td>HD, FLOW, SCT</td>
</tr>
<tr>
<td>Autodesk® Revit*</td>
<td>ST</td>
<td>Compliant with Launcher</td>
</tr>
<tr>
<td>ARCHICAD</td>
<td>ST</td>
<td>Compliant with Launcher</td>
</tr>
<tr>
<td>IGES</td>
<td>ST</td>
<td>HD, FLOW, SCT</td>
</tr>
<tr>
<td>VDAFS</td>
<td>ST</td>
<td>HD, FLOW, SCT</td>
</tr>
<tr>
<td>ACIS</td>
<td>ST</td>
<td>HD, FLOW, SCT</td>
</tr>
<tr>
<td>Parasolid</td>
<td>ST</td>
<td>HD, FLOW, SCT</td>
</tr>
<tr>
<td>STEP</td>
<td>ST</td>
<td>HD, FLOW, SCT</td>
</tr>
<tr>
<td>IFC</td>
<td>ST</td>
<td>SHP (Polyline, polygon)</td>
</tr>
<tr>
<td>SHAPE</td>
<td>ST</td>
<td>3ds</td>
</tr>
<tr>
<td>3ds</td>
<td>ST</td>
<td>-</td>
</tr>
<tr>
<td>STL</td>
<td>ST</td>
<td>HD, FLOW, SCT</td>
</tr>
<tr>
<td>Nastran</td>
<td>ST</td>
<td>HD, FLOW, SCT</td>
</tr>
<tr>
<td>SketchUp</td>
<td>ST</td>
<td>-</td>
</tr>
<tr>
<td>Albaque*</td>
<td>ST</td>
<td>-</td>
</tr>
<tr>
<td>DesignSpace</td>
<td>ST</td>
<td>-</td>
</tr>
<tr>
<td>Plot3D</td>
<td>ST</td>
<td>-</td>
</tr>
<tr>
<td>CGNS</td>
<td>ST</td>
<td>-</td>
</tr>
<tr>
<td>DFX</td>
<td>ST</td>
<td>-</td>
</tr>
<tr>
<td>IDF</td>
<td>ST</td>
<td>-</td>
</tr>
<tr>
<td>GERBER</td>
<td>ST</td>
<td>-</td>
</tr>
<tr>
<td>Parasolid</td>
<td>ST</td>
<td>HD, FLOW, SCT</td>
</tr>
<tr>
<td>STL</td>
<td>ST</td>
<td>HD, FLOW, SCT</td>
</tr>
<tr>
<td>Nastran</td>
<td>ST</td>
<td>HD, FLOW, SCT</td>
</tr>
<tr>
<td>CGNS</td>
<td>ST</td>
<td>HD, FLOW, SCT</td>
</tr>
</tbody>
</table>

### Import

#### CAD Interface

**HPC (High Performance Computing) Solution**
Large-scale, high-speed simulation with parallel computing technologies
Parallel computing makes possible solving existing models faster, conducting more analyses, and/or solving more detailed models with a greater number of mesh elements.

#### scSTREAM V14

- **Arena**: Approx. 70 million elements
- **Projector**: Approx. 20 million elements

#### scFLOW V14

- **Converter**: Approx. 15 million elements
- **Sirocco fan**: Approx. 15 million elements
Main Mutual Features

**scMonitor**
You can visualize the progress of the simulations in scMonitor during the Solver calculations. You can check, for example, pressure contour of a registered surface and temperature contour and flow vector on axial planes.

* To use this function, a Postprocessor license is required in some areas.

**LFileView**
LFileView is a dedicated viewer for L files, which are output during the simulations automatically. You can check the progress of the simulations numerically with variable values for each cycle and the maximum/minimum/average values for the specified output.

**VB Interface**
The software supports COM technology provided by Microsoft. You can control the software by using Microsoft Office products and Visual Basic (VB). A tool to create and execute the automatic operation flow, scWorkS ketch, is bundled with the software. By using the tool, you can create your original automatic operation flow easily. In addition, you can register the created flow as a template and reuse it.

**Parametric Study Tool**
Using the parametric study tool, you can set analysis conditions to multiple cases all at once - for instance, when you run several calculations with modified parameters such as flow rate or amount of heat. The interface is user-friendly with spreadsheet-like settings. You can check, in the same interface, the status of each case and the output parameters such as the maximum/minimum temperature or average pressure on a specified plane.

* This tool is available in scSTREAM, HeatDesigner, and SC/Tetra

**Useful functions**
- Tool to create a report automatically
- Unique GUI
- Tool to create a model from the 2D data automatically
CradleViewer
The simulation result visualized in Postprocessor can be saved in a file and the file can be opened in a simple viewer. In the viewer, the viewpoint and the distance can be changed with the mouse and by touch operation*. CradleViewer is provided free of charge. You can share the simulation result even in an environment without Postprocessor installed.

* Operation using two fingers is supported on a multitouch-compatible screen in a Windows 10 or Windows 8.1 environment.

HeatPathView
Using HeatPathView, you can review heat dissipation measures with focus on each component. The tool enables the intuitive and comprehensive evaluation of heat balance and search of heat dissipation paths. By understanding the flow of heat, you can make your heat dissipation designs more reliable.

scCONVERTER
Data (FLD/FPH files) such as pressure, temperature, and heat transfer coefficient obtained in thermo-fluid analyses can be mapped to input data of structural analysis software (Abaqus, I-DEAS, Nastran). In addition, input data of structural analyses can be converted to an FLD or FPH file. scCONVERTER can create an animation file from multiple still images (BMP/PNG files), edit FLD/FPH files, and convert a P file to an FLD file or an FLD file to an iFLD file.
Introducing Optimus®
Optimus is an integration platform of simulation tools with optimization and automation as its cores

1. Optimus®
Optimus has a direct interface with scSTREAM, scFLOW and SC/Tetra, and the optimization can be performed without any additional customization. It also supports a creation of an original GUI using API and an optimization using Quality Engineering (Taguchi Method).

Automation/Integration
Executes the processes automatically just by constructing a simulation workflow with icons.

Data Mining
Visualizes data immediately. Relationships between the parameters can be grasped easily from sensitivity and correlation analyses.

Optimization
Optimization algorithm automatically searches the parameters yielding the best performance.

Robustness, Reliability
Predicts the variations in performance from the variations of parts. This enables the design with consideration on the variations in advance.

Comparison Table: Optimus® and Optimus® for Cradle

<table>
<thead>
<tr>
<th></th>
<th>Optimus®</th>
<th>Optimus® for Cradle</th>
</tr>
</thead>
<tbody>
<tr>
<td>Linkage to Software Cradle products</td>
<td>Condition setting</td>
<td>(Direct interface)</td>
</tr>
<tr>
<td>Linkage to third party products</td>
<td>Shape modification using CAD</td>
<td>(Direct Interface)</td>
</tr>
<tr>
<td>Calculation method</td>
<td>DOE</td>
<td>Total 23 methods</td>
</tr>
<tr>
<td></td>
<td>Response surface</td>
<td>All 5 + Optional 11 methods</td>
</tr>
<tr>
<td></td>
<td>Single-objective optimization</td>
<td>Local: Total 5 methods</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Global: Total 7 + Optional 5 methods</td>
</tr>
<tr>
<td></td>
<td>Multi-objective optimization</td>
<td>Total 11 + Optional 5 methods</td>
</tr>
<tr>
<td></td>
<td>Robustness / Reliability / Quality Engineering</td>
<td>Total 7 methods, orthogonal table L4-512, static/dynamic characteristics</td>
</tr>
<tr>
<td>Postprocessing</td>
<td>Method</td>
<td>Total 23 types</td>
</tr>
<tr>
<td></td>
<td>Model</td>
<td>Total 13 types</td>
</tr>
</tbody>
</table>
License type
We provide various license types based on customer operations from on-premise to cloud

1. On-premise license: Features
- Underpriced
- Internally manageable machines
- Existing hardware resources
- No data transfer
- Internally controlled security
- In-house tools (e.g. automation)
- Multiple tools in combination

2. Cloud license: Features
- On-demand offer*
- No hardware required
- Large-scale calculations
- Support for sudden need
- No maintenance required
- Underpriced for infrequent users
- Not an asset

Recommended for customers who want to...
- Incorporate analysis into design workflow constantly and reduce cost
- Elaborate a combinational use of multiple analysis tools with a simple system
- Simplify analysis operations for obtaining design pointers and allow several users short-term uses

Recommended for customers who want to...
- Finish large-scale calculations in a short time although ordinary calculations can be performed with on-premise licenses
- Use outside resources temporarily because in-house resources is insufficient at the time
- Handle intensive calculation jobs efficiently for one project without placing burden on in-house resources

License type lookup table

<table>
<thead>
<tr>
<th>Software</th>
<th>License</th>
<th>Agreement type, period</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Floating</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Monthly</td>
</tr>
<tr>
<td>scSTREAM SC/Tetra</td>
<td>Pre/Post</td>
<td>Standard edition</td>
</tr>
<tr>
<td></td>
<td></td>
<td>HPC edition</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Solver</td>
</tr>
<tr>
<td></td>
<td></td>
<td>HPC edition</td>
</tr>
<tr>
<td>HeatDesigner</td>
<td>Pre/Solver/Post</td>
<td>Standard edition</td>
</tr>
<tr>
<td>scFLOW</td>
<td>Pre/Post</td>
<td>Standard edition</td>
</tr>
<tr>
<td></td>
<td></td>
<td>HPC edition</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Solver</td>
</tr>
<tr>
<td></td>
<td></td>
<td>HPC edition</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Unlimited edition</td>
</tr>
<tr>
<td>PICLS</td>
<td></td>
<td>●</td>
</tr>
<tr>
<td>Optimus® for Cradle</td>
<td></td>
<td>●</td>
</tr>
</tbody>
</table>

*1 Only available in a certain region
*2 Agreement type available with Altair Partner Alliance (APA) provided by Altair Engineering, Inc. For details, go to the official website of APA.
Links with other software

1. Electromagnetic Field Analysis Software

Using the data output from the electromagnetic analysis software, the effect of heat source distribution due to an electromagnetic field can be analyzed.

- **JMAG-Designer**
  - Developed by JSOL Corporation (Japan)

- **Femtet®**
  - Developed by Murata Software Co., Ltd (Japan)

- **EMsolution®**
  - Developed by Science Solution International Laboratory, Inc. (Japan)

- **FlowNoise**
  - Developed by CEDIC (Korea)

- **LMS Virtual.Lab**
  - Developed by Siemens PLM Software (USA)

2. Acoustic Analysis Software

The acoustics of aerodynamic noise can be analyzed using SC/Tetra output data.

- **LMS Virtual.Lab**
  - Developed by Siemens PLM Software (USA)

- **FlowNoise**
  - Developed by CEDIC (Korea)

- **Actran**
  - Developed by Dassault Systèmes S.A. (France)

3. Structural Analysis Software

Using the output data from scFLOW, SC/Tetra and scSTREAM, structural analysis can include the influence of heat transfer and other fluid interactions.

- **Abaqus®**
  - Developed by Dassault Systèmes S.A. (France)

- **Nastran**
  - Developed by Dassault Systèmes S.A. (France)

- **Femtet®**
  - Developed by Murata Software Co., Ltd (Japan)

- **ADVENTURECluster**
  - Developed by Allied Engineering Corporation (Japan)
4. Thermal Environment Simulation Software
The surface temperature distribution is output from the thermal environment simulation software. The output data can be used as boundary conditions for scSTREAM calculation to analyze the distribution of wind velocity and temperature.

5. One-Dimensional Analysis Software
Computational load can be reduced by not solving all of the thermo-fluid analysis in three dimensions but using one-dimensional analysis software for some part.

6. Chemical Reaction Analysis Software
Using material property parameters and chemical reaction database of LOGE, coupled analysis with SC/Tetra can be performed. This enables analysis of overall chemical reactions and detailed chemical reactions, which could not be analyzed by SC/Tetra alone.

7. 3D CAD Software
The direct interface (Launcher) equipped with scSTREAM enables the software to directly load original 3D CAD data.

8. Optimization Software
Software Cradle products can be used in conjunction with optimization software for automation and/or optimizing product design.

9. Visualization Software
Read, visualize and edit FLD data (analysis results file from Software Cradle products) using other visualization software.
About Software Cradle

Software Cradle Co., Ltd. is an innovative provider of computational fluid dynamics (CFD) simulation software. Established in 1984, the company has pursued to offer unique, innovation focused, and highly reliable CFD solutions that enhance customers’ product quality and creativity. In 2016, the company joined MSC Software Corporation (headquartered in Newport Beach, California, US), the worldwide leader in the field of multidiscipline simulation. As a truly global company, Software Cradle delivers all-inclusive multi-physics solutions.

For more information about MSC Software Corporation, please visit: