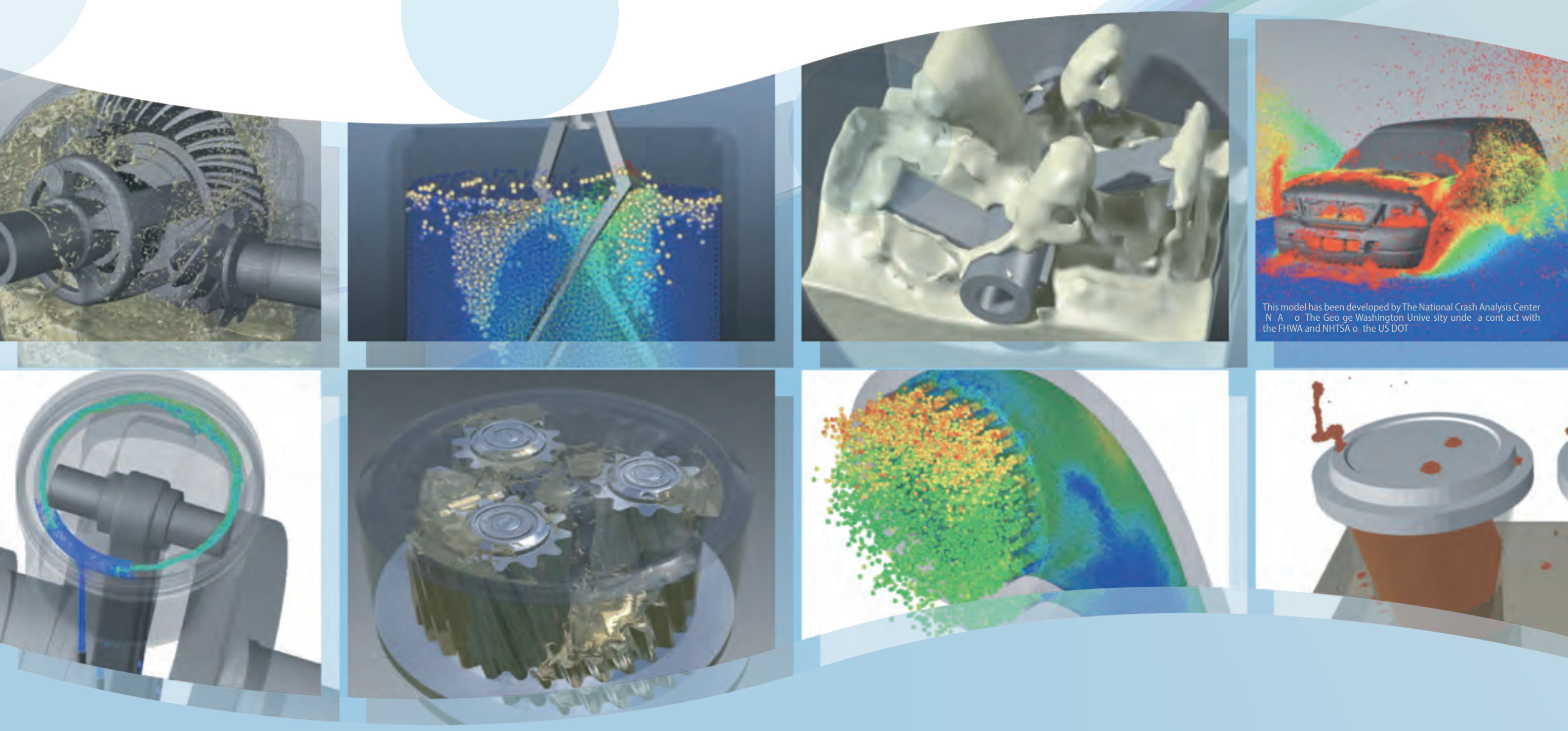
# Particleworks™

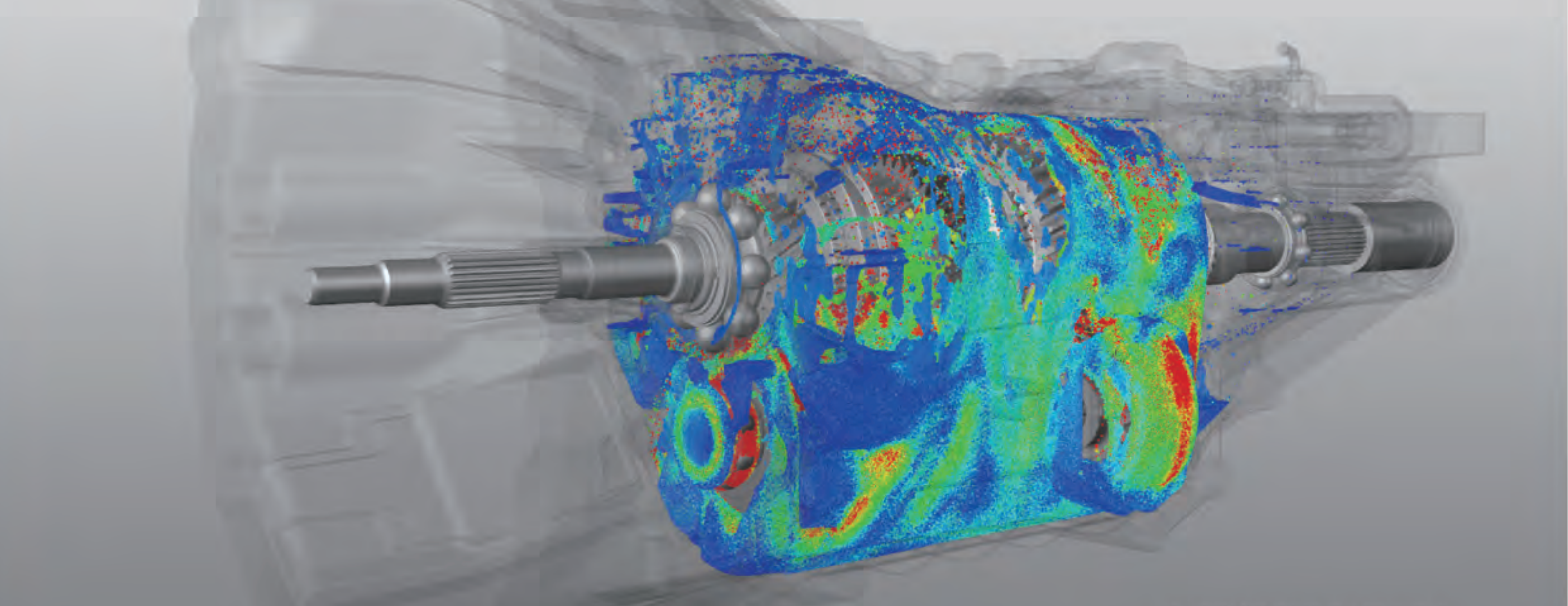
Particle-based simulation software for CAE

**Meshfree liquid flow simulation**




PROMETECH

# What is Particleworks?



Oil distribution calculation in transmission
Courtesy of UNIVANCE CORPORATION

## High-speed simulation of liquid behavior such as water and oil

Particleworks is a CFD simulation software for evaluating the behavior of liquids with large scale deformation, such as water and oil. It specializes in the analysis of free surface and incompressible fluids with large scale deformation and is used in the analysis of fluid problems in a wide range of fields such as transmission and engine lubrication, motor cooling, flooded road running, agitation and mixing of chemicals, resins, food, etc., as well as sediment disasters and flooding.

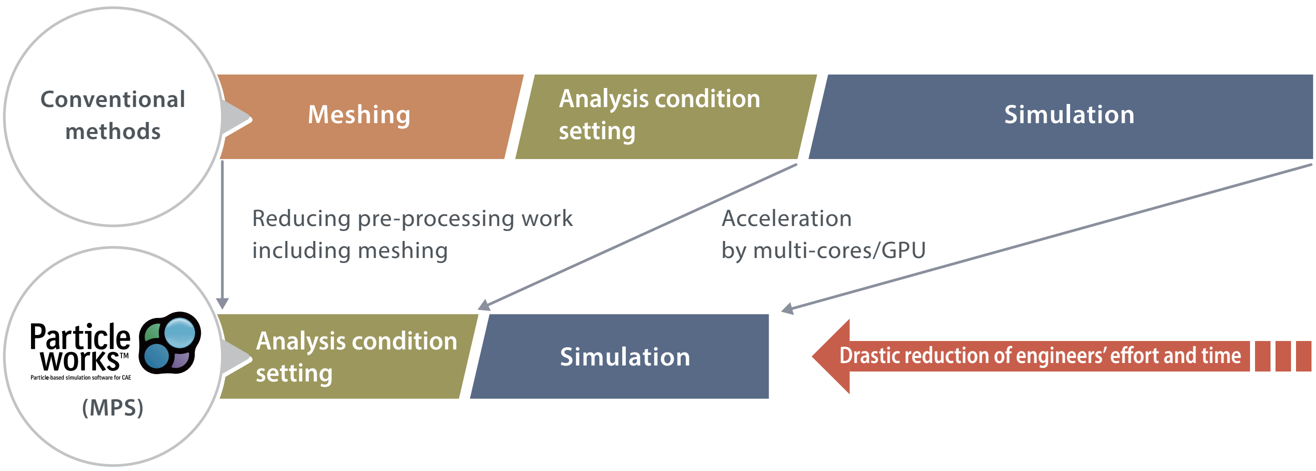
### Mesh-free

Particleworks uses MPS, the Moving Particle Simulation Method, developed by Prof. Seiichi Koshizuka of the University of Tokyo. It does not require meshing of the computational domain as in the FDM, FVM, and FEM, and uses particles that can be moved based on the calculation results as computational points.

Therefore, once CAD geometry data is imported, you can immediately move on to pre-processing for setting analysis conditions, significantly reducing simulation time and man-hours.

Seiichi Koshizuka, Ph.D.

**Comparison of man-hours between the conventional methods and the Particleworks**

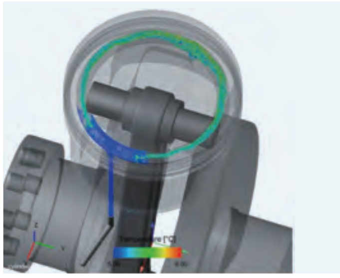


## Incorporating the latest research findings to help solving a variety of industrial problems

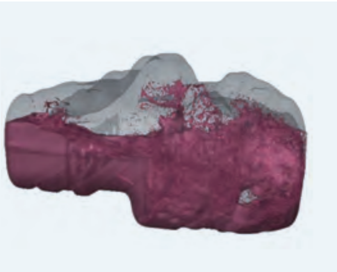
Since its release in 2009, Particleworks, a CFD simulation software from Japan, has been improved and enhanced continuously by incorporating analysis know-how from universities and industrial companies. Now its applications are diverse, and Particleworks has been in a wide range of fields, including automotive, steel/metals, medical/pharmaceutical, food/consumer goods, civil engineering, electric appliances/machinery, materials, energy/power.

## Particleworks Application

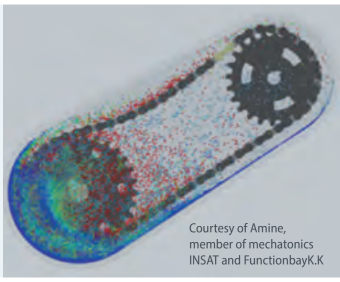
* including CG rendering images



Piston cooling



Fuel tank sloshing




Chain oil lubrication


Flooded road running

**Particleworks** — Particle-based simulation software for CAE

**Automotive**
- Flooded road running and water/mud splash analysis
- Gear/chain oil lubrication
- Motor cooling
- Oil sloshing/oil jet in the engine
- Aeration in the engine breather system

**Iron/steel Metals**
- Cooling spray water behavior
- Extrusion molding

**Medical & Pharmaceutical**
- Blood flow
- Mouth washing
- Shape optimization of tablets
- Pharmaceutical mixing

**Food & Consumer goods**
- Food processing mixer
- Cosmetics emulsification
- Shower heads

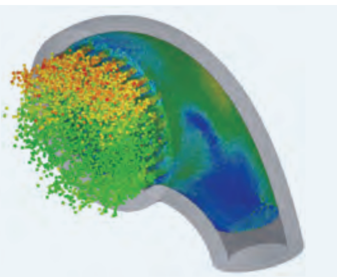
**Energy & Power**
- Spent fuel pool sloshing
- Water flow in pressure vessel

**Materials**
- Soldering
- Mixing
- Melting and kneading analysis of resin
- Impregnation of composite material

**Electric Appliances & Machinery**
- Washing machine balancer
- Swarf (metal chips, fillings, shavings, or turnings) cleaning
- Leak test

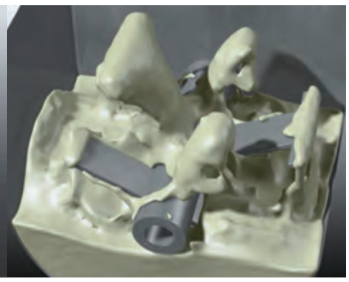
**Civil engineering**
- Debris flow/Tsunami run-up
- Fresh concrete flow
- Tunnel excavation

Coffee spilling

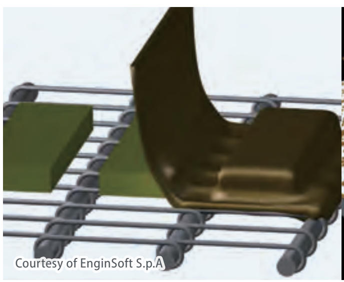


Shower head water flow

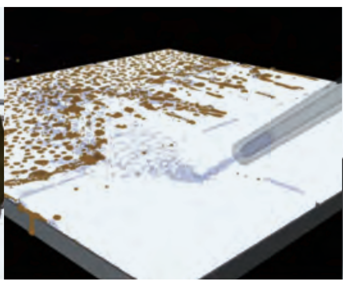

Washing machine mixing



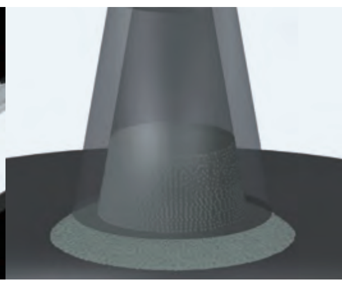
High-viscosity resin mixing



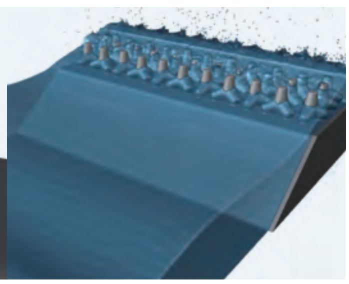

Chocolate coating



High-pressure water sprayer



Fresh concrete flow



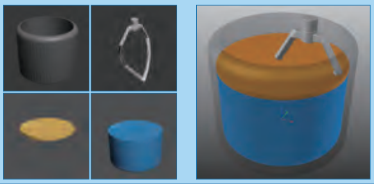
Surge to the breakwater

# GUI & Pre-post

## Simulation Flow

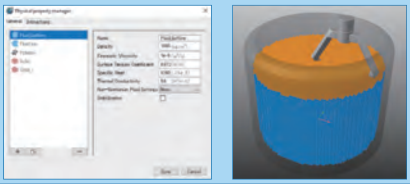
### STEP 1 Modeling

Import CAD data and define the resolution for pre-processing. The STL, OBJ, and NASTRAN formats are supported.



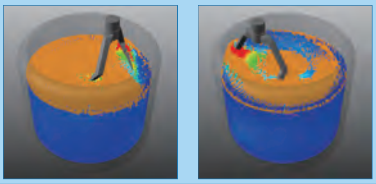
### STEP 2 Condition Settings

Simply apply physical properties and movements to the model. No tedious adjustments are needed for boundary conditions.



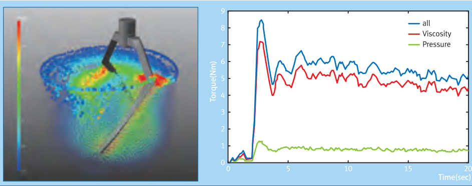
### STEP 3 Simulation

You can accelerate the calculation using multiple CPU cores or GPUs. Additionally, you can view the results of a simulation while it is still in progress.



### STEP 4 Post-Processing

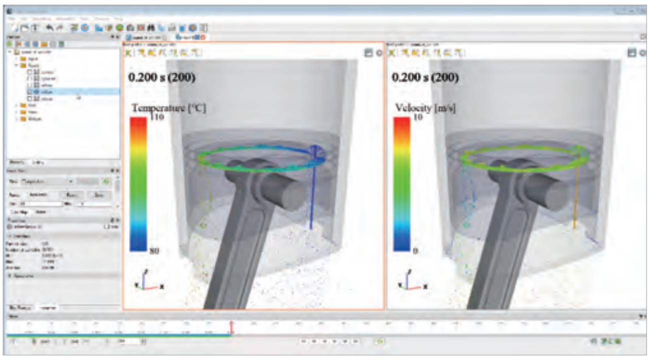
Visualize and evaluate the simulation results using various post-processing tools. For example, you can create surface meshes and export CSV and video files.




## User Interface

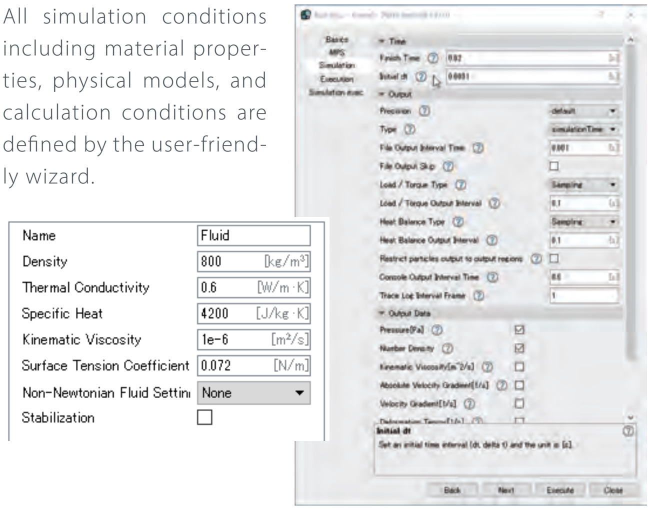
Particleworks' intuitive user interface lets you handle an entire simulation, from pre-processing through post-processing. You don't have to be an expert to edit simulation parameters or keep track of multiple projects.

The 3D view window features ultra-fast, high-quality OpenGL rendering optimized for large-scale simulation with millions of particles. The window system is highly customizable, letting users compare multiple results side by side. Both Windows and Linux are supported.



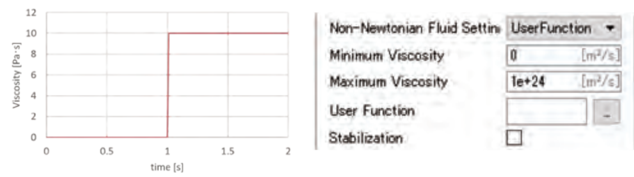

## Pre-Processing

All simulation conditions including material properties, physical models, and calculation conditions are defined by the user-friendly wizard.




Definition wizard

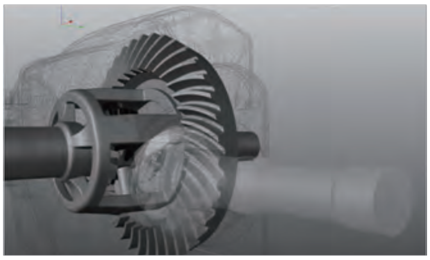
Because you can set the viscosity as a user function, it is possible to perform analyses using time-varying viscosity models and viscosity models which viscosity changes significantly under certain conditions.



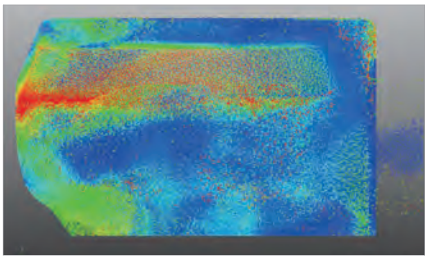

Time Dependent Viscosity

# Post-Processing

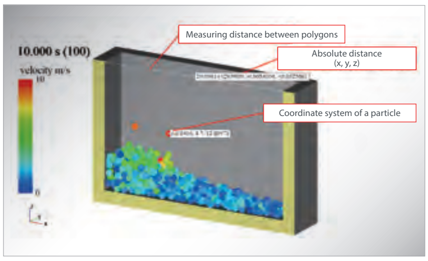
## Visualization



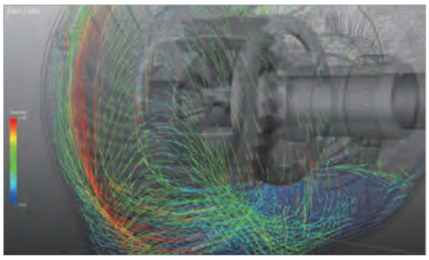
Solid, wire, and transparent views



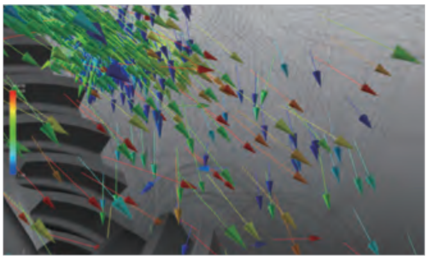
Cross-section display



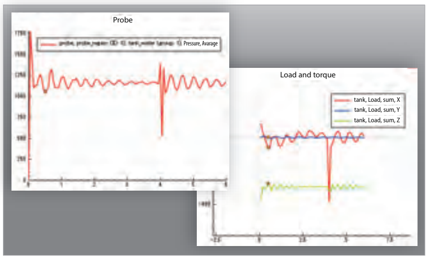

Measuring



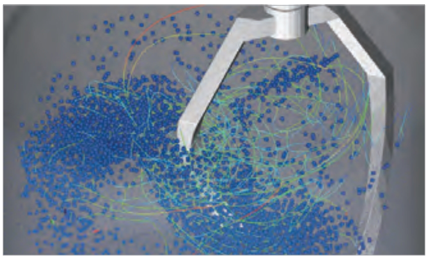
Streamline



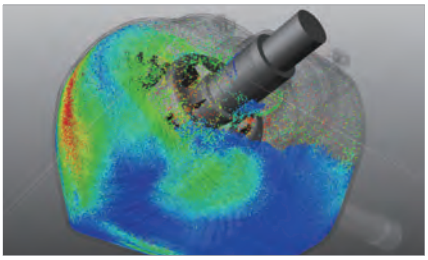
Vector view



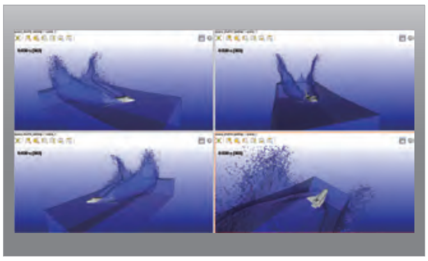

A variety of graphs



Path lines



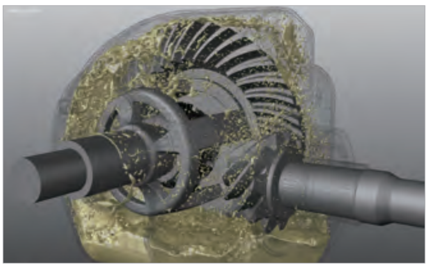
Color mapping of pressure, velocity, and temperature



Multi-view

## Surface Mesh Generator

Surface meshes can be generated using particle locations, letting users evaluate the behavior of a fluid surface or calculate the area of a surface. Mesh data can be exported to the STL and OBJ formats.

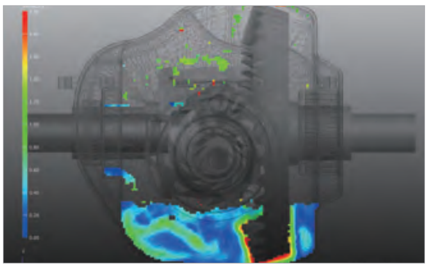


## Image and Video Export

Simulation results and motion data can be exported to video or sequential image files, as well as still images (screenshots).
The PNG, JPEG, MP4, and WMV formats are supported.

## Grid Data Generator

The physical quantities each particle carries can be projected onto grid points. Further visualization including contours, vectors, iso-surfaces, isolines and streamlines can be performed by using the grid data.

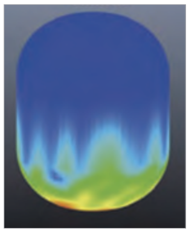


## Probing Particles, and others

Particles can be selected and probed with a single click. Probe and particle filters allow users to calculate statistics over particles that exist within a certain range (of any quantity) or near a probe point. Other tools including the color bar, rulers, and time code guides the users through every post-processing step.

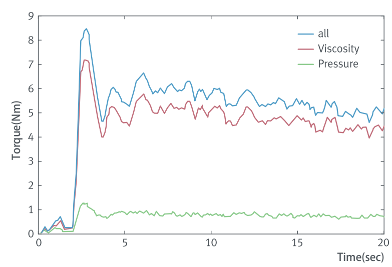
## Data Mapping to Polygons

Particle data can be projected onto vertices of a polygon mesh, which can be exported as CSV or binary files. These files can be used as input to third-party mesh-based programs.

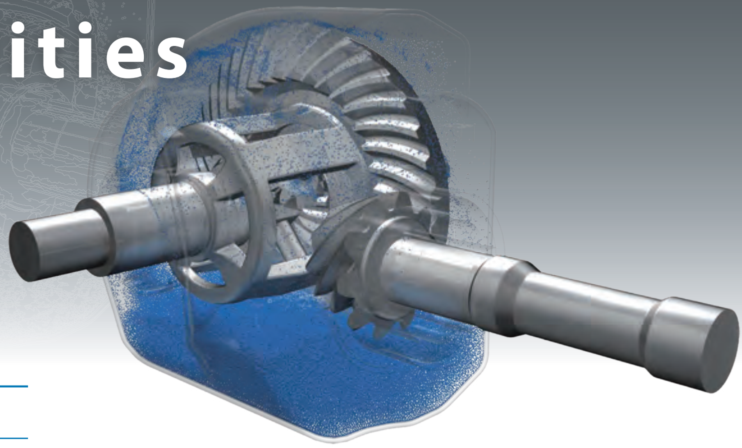


## CSV Export

Particle data can be exported to CSV files, allowing for further data processing on quantities including coordinates, velocity, pressure, number density , and sheer velocity. Force and torque against polygon walls can also be exported.



# Solver Capabilities and Physics



## Boundary conditions

**Wall Boundaries / Moving Boundaries**

Both particle and polygon walls can be selected as wall boundaries. Polygon walls are effective in reducing memory usage and speeding up calculations. It is also possible to reproduce complex movements of walls (objects) by using the motion setting function.

**Inflow Boundaries**

Inflow boundaries allows for the generation of fluid or powder over time. The flow can be specified by its velocity or flow rate (volume). Inflows are movable.

**Moving / Periodic Boundaries**

The mesh-free method allows the simulation region to be moved. This saves computational resources when simulating a large region, such as a waterway driving test. Periodic boundaries are also supported.

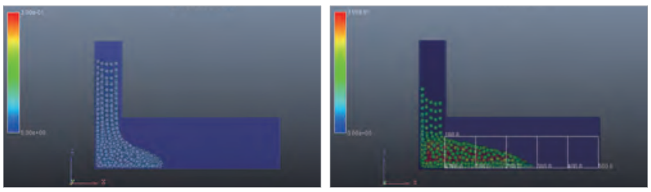
## Viscosity

**Newtonian / Non-Newtonian Fluids**

Particleworks can simulate non-Newtonian fluids such as power-law or Bingham fluid as well as Newtonian fluids. For more detailed control over viscosity, you can specify custom functions or data tables.

**High-Viscosity Fluids**

When simulating high-viscosity fluids, the explicit method tends to give a smaller time step, resulting in a longer calculation. In contrast, Particleworks' implicit method maintains a constant time step, making it an ideal solution for such simulations.



## Pressure

**Implicit / Explicit Methods**

The explicit method speeds up calculation by giving a suitable speed of sound.

**Suppression of Pressure Oscillation**

Spatial pressure oscillation can be suppressed using this function, resulting in higher accuracy.

## Rigid Bodies

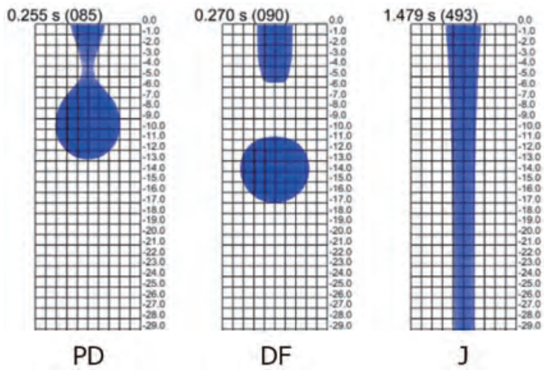
The interaction between complex flow and non-deforming objects or rigid bodies can be analyzed straightforwardly.

Washout simulation of scrap metal (CG rendering)

## Surface Tension

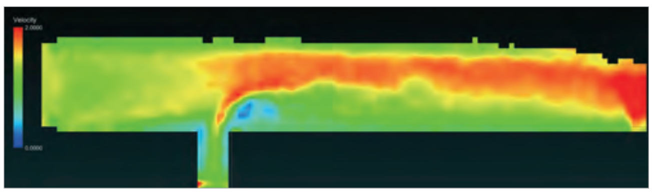
Particleworks offers two models: The CSF model calculates surface tension from the geometric shape of the object, whereas the Potential model uses interfacial energy between objects.

You can set contact angles between two different states of matter, such as wall-fluid and fluid-fluid. Multiple non-mixable fluids can be simulated, such as oil and water.



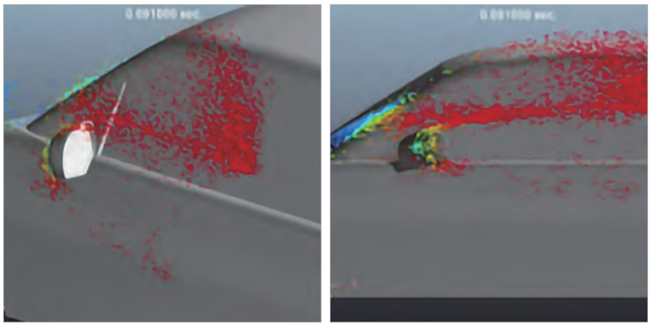

## Turbulence

To simulate turbulence flows, Particleworks uses a hybrid model in which LES (Large Eddy Simulation) is combined with resolution enhancement near walls.



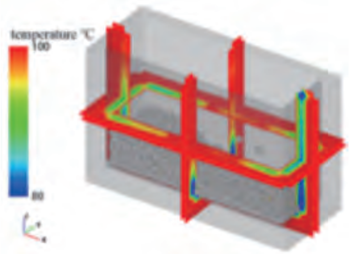
## Air Resistance

Particleworks can import data points calculated by external CFD programs (in CSV format), such as airflow field around a car body . This function is useful when analyzing the behavior of droplets with air resistance.



## Conjugate heat transfer analysis New in Version 7.0

Heat transfer analysis between fluids and structures has been enhanced and its performance improved.

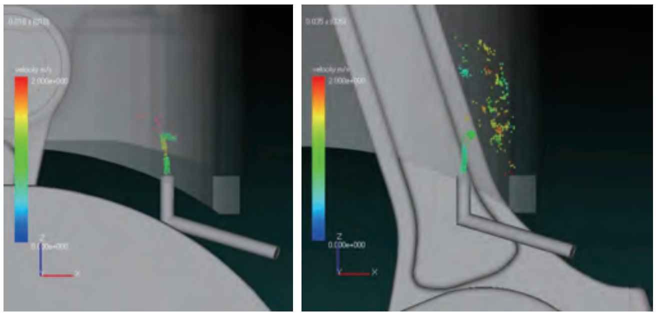


## User function definition of heat transfer coefficient

As a user function, you can define a formula for calculating the heat transfer coefficient according to the flow field.

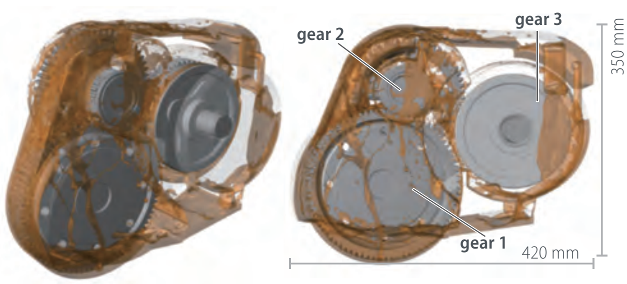
## Gas-liquid two-phase flow analysis Beta capability in Version 7.0

It enables efficient analysis of gas-liquid two-phase flows.



# GPU/CPU High Performance Computing

With the addition of the GPU option feature, it enables fast simulations that take advantage of NVIDIA's CUDA GPU boards. This can significantly reduce computation time and allow you to perform high-speed calculations on a desktop PC, which is equivalent to supercomputers and HPC servers. Also, a new HPC Pack is available to expand the number of cores for parallel computing. The number of cores can be increased efficiently based on 1 Pack (16 cores). You can also install multiple packs, which allows you to assign packs to different jobs. Also, multiple packs can be installed, allowing you to assign a pack to each job.

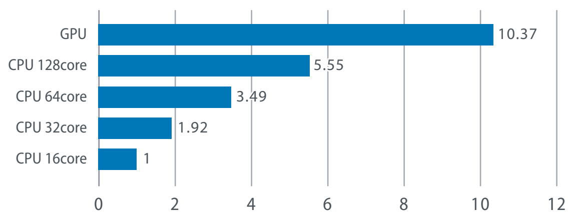



Model (number of particles: 4.6mil)

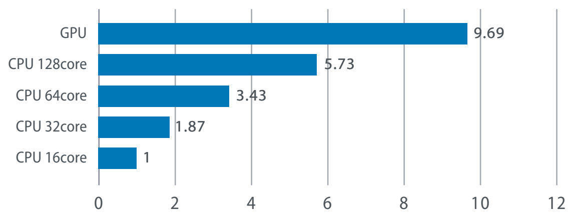
| Gear rotation speed | |
|---|---|
| Gear 1 | 500 rpm |
| Gear 2 | 1739 rpm |
| Gear 3 | 1771 rpm |

| Fluid property | |
|---|---|
| Density | 800 kg/m³ |
| Kinematic viscosity coefficient | $1\times10^{-5}$ m²/s |
| Surface tension coefficient | 0.03 N/m |

Computational speed ratio with 16 cores of CPU as 1 (Explicit)




Computational speed ratio with 16 cores of CPU as 1 (Implicit)




GPU: Tesla V100 CPU: Intel Xeon E5-2660v3

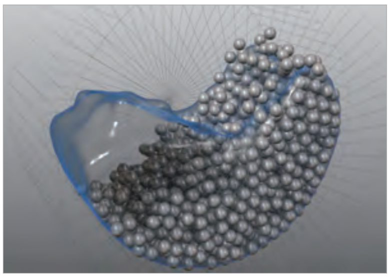
# Multiphysics Solution

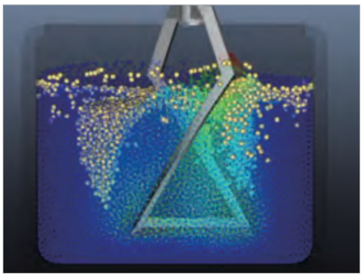
Particleworks can be coupled with Prometech's granular material simulator Granuleworks, and 3rd party software tools that analyze structural, crash, motion, fluid, and electro-magnetic behaviors. This allows users to evaluate the design to achieve more realistic physical phenomena.

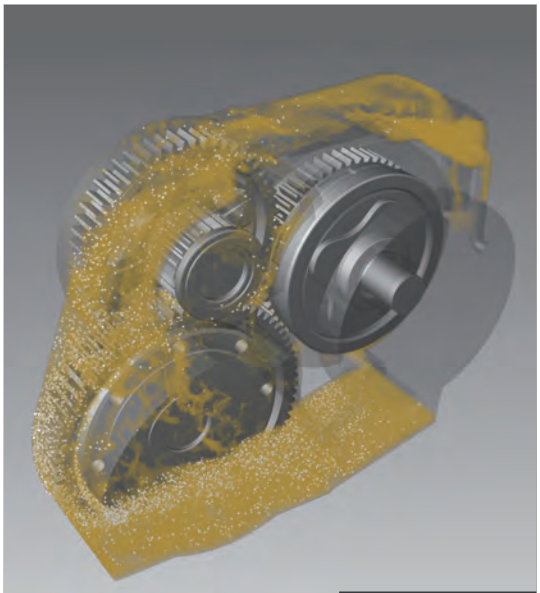
## Powder-Liquid Flow

By combining Granuleworks' DEM (Discrete Element Method) and Particleworks' MPS, you can simulate the motion of powder particles in fluids.
Aeration analysis, which can evaluate bubble behavior to predict engine oil behavior and chemical processes in stirring tanks, is available. This feature comes in handy when examining design issues related to bubbles. You can:

- Choose the size of simulated bubbles
- Calculate buoyancy force, wall force, drag force, bubble extinction, bubble coalescence, and bubble breakage
- View statistics for spatial distributions of bubbles based on size
- Visualize and spot issues related to bubble behavior



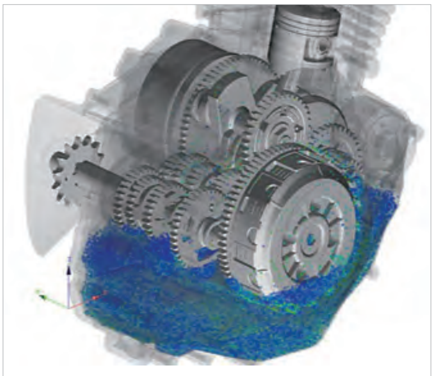




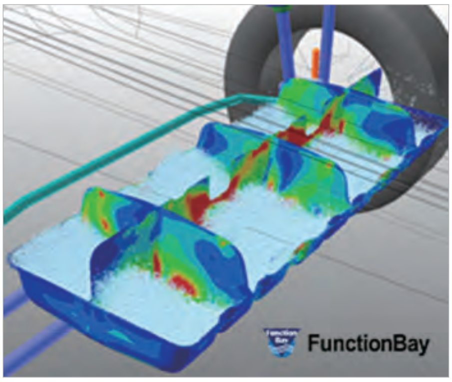
Aeration analysis

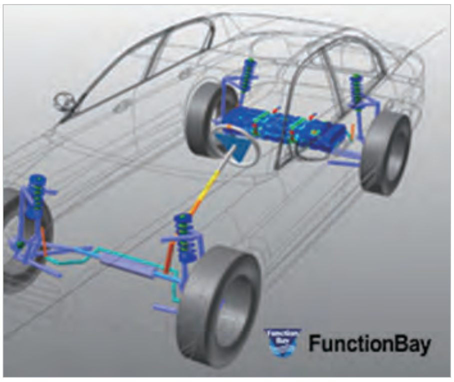
## Motion-Fluid

If Particleworks is used alone, the motion of bodies is defined by the user. Coupling Particleworks with RecurDyn, a multi-body dynamics simulation package developed by FunctionBay, Inc., allows for the bodies to move according to the laws of physics. In Particleworks x RecurDyn coupling, the physics of the fluid, computed by Particleworks, and the physics of the mechanical system, computed by RecurDyn, are completely, bidirectionally coupled, enabling accurate simulation of fluid-mechanism interaction.



Courtesy of Thomas Frevillier and FunctionBayK.K.



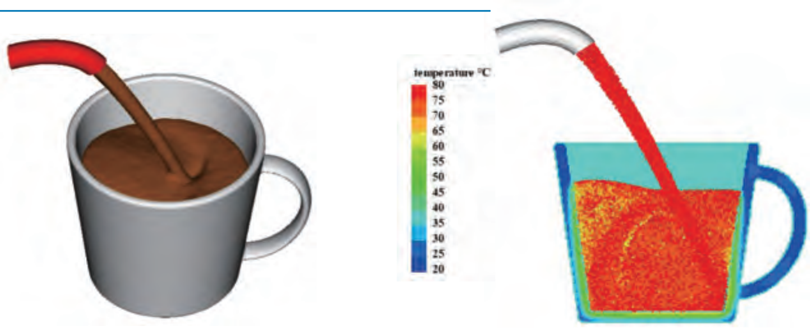




Simulation of baffle deformation and stress caused by pressure in oil tank while the car is in motion
Courtesy of FunctionBay, Inc.

RecurDyn supports fluid interaction with both rigid and flexible bodies. This permits fluid interaction with bodies experiencing large deformation, and it permits the calculation of deformation, stresses, and strains imparted to bodies by fluid forces.

## Fluid - Heat generation and temperature distribution prediction

Conjugate heat transfer analysis feature added in Particleworks 7.0 allows you to predict cooling by oil and water such as engine cylinder head cooling, motor cooling, steel plate cooling, etc.



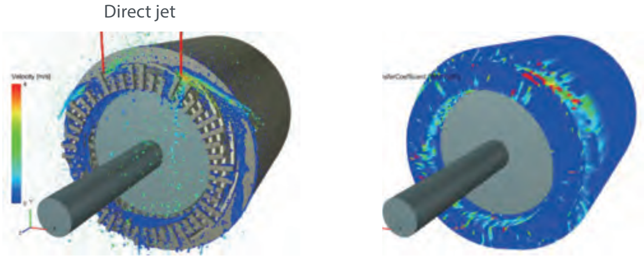

Heat transfer analysis between coffee (fluid) and a cup (solid)

## Coupling with various CAE tools

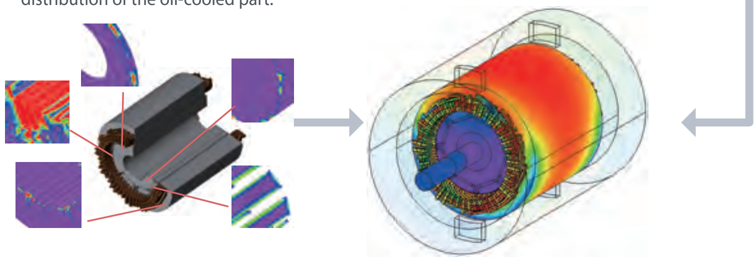
The results of the fluid simulation of Particleworks can be used in 3rd party CAE tools such as Abaqus, ANSYS, MSC/ Nastran, NX Nastran, LS-DYNA, and JMAG.

Physical quantity data (coordinates, pressure values, heat transfer coefficients, etc.) for Particleworks can be output as a CSV file and be converted into a format that allows you to use in other CAE tools, as the boundary conditions.

It can also be used with optimization software tools such as OPTIMUS or modeFRONTIER.



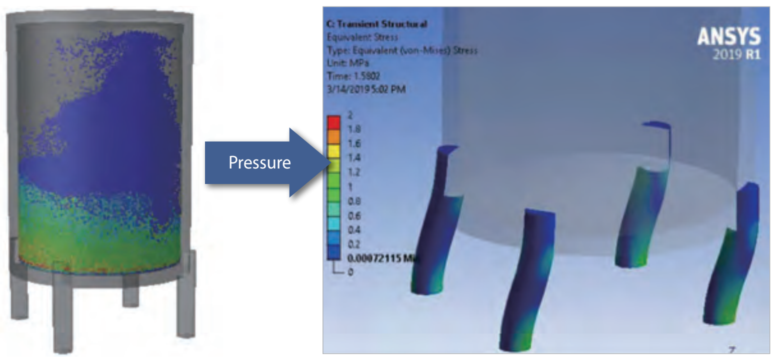

Particleworks simulates oil behavior and outputs the heat transfer coefficient distribution of the oil-cooled part.



Temperature distribution calculation considering the loss distribution of each part of the motor obtained by JMAG and the heat transfer coefficient output from Particleworks

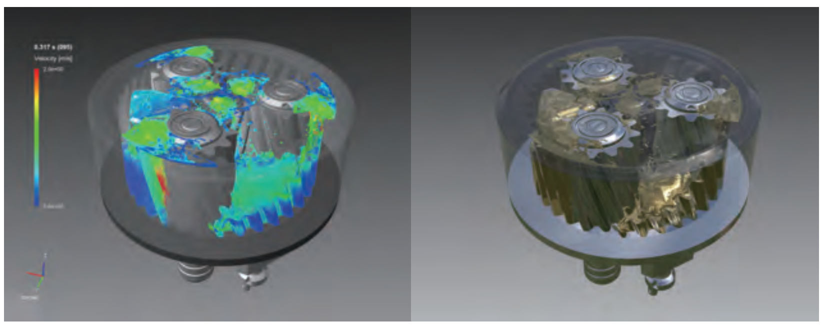
## Particleworks for ANSYS (option program)

Particleworks for ANSYS is an interface option for coupling analysis between Particleworks and ANSYS. By using this, Particleworks can be used in ANSYS Workbench environment and this makes it possible to simulate complicated multiphysics behaviors including liquid-structure, liquid-thermal, and gas-liquid flow easily.




## Visulization Option (option program)

Visualization Option converts the simulation results of Particleworks and Granuleworks into a general-purpose CG format and enables CG editing and rendering using CG software. It streamlines visualization from CAE simulation results and accelerates video creation and XR(VR/AR/MR) content development.



# Capabilities Chart

○: Supported　—: not supported

| | | | | CPU | GPU |
|---|---|---|---|---|---|
| Solver | Physical models・Features | Viscosity model | Newtonian fluid | ○ | ○ |
| | | | Non-Newtonian fluid (Bingham, Power law, Cross-Arrhenius, Data table input, User function) | ○ | ○ |
| | | Pressure term solution | Implicit / explicit method, Pressure oscillation suppression | ○ | ○ |
| | | | Negative pressure model | ○ | ○ |
| | | Viscosity term solution | Implicit / explicit method | ○ | ○ |
| | | Turbulence model | LES model + wall function model | ○ | ○ |
| | | Airflow | | ○ | ○ |
| | | Surface tension model | Potential model | ○ | ○ |
| | | | CSF model | ○ | ○ |
| | | Rigid body | Rigid-body motion, Contact, Fluid-rigid body coupling simulation | ○ | ○ |
| | | Thermal properties | Conjugate heat transfer analysis, Thermal conductivity analysis using rigid particles and wall particles<br>Thermal conduction, thermal viscosity, shear heat, thermal heat coefficient output | ○ | ○ |
| | | External force | Constant acceleration (gravity), Time-series data input | ○ | ○ |
| | | Aeration | Generation, rupture, wall force, coalescence, and drag force of bubbles with size distribution | ○ | ○ |
| | Boundary conditions | Particle wall | Forced motion (sloshing and mixing)<br>Adiabatic boundary, isothermal boundary, thermal calculation | ○ | ○ |
| | | Polygon wall | Forced motion (sloshing and mixing), Force & torque output<br>Adiabatic boundary, isothermal boundary,<br>Motion settings for structures that move periodically | ○ | ○ |
| | | Inflow boundary | Velocity, flow rate, flow volume, temperature input<br>Time-series input | ○ | ○ |
| | | Outflow boundary | Shape specified or region specified | ○ | ○ |
| | | Simulation domain | Periodic and moving boundary in orthogonal coordinate system Deletion of outflow particles from simulation domain | ○ | ○ |
| | | Pump | Region and velocity specified | ○ | ○ |
| | Parallel processing | | Inside-node parallel processing (SMP Multi-cores) | ○ | — |
| | | | Inter-node parallel processing (MPI) | ○ | — |
| | | | SMP and MPI hybrid parallel processing | ○ | — |
| | | | Inside-node parallel processing (multi-GPUs) | — | ○ |
| | | | Inter-node parallel processing (GPU cluster) | — | ○ |
| | Coupling-simulation | | Rigid and flexible of RecurDyn | ○ | ○ |
| | | | Interface for ANSYS "Particleworks for ANSYS"※ | — | — |
| | | | FMI interface | — | — |

| | | | |
|---|---|---|---|
| Pre / Postprocessing | Pre-Processing | Generator | Particle generation from geometry file (OBJ / STL) |
| | | | Particle generation from defined liquid level |
| | | | Boundary (distance function) generation from geometry file (OBJ / STL) |
| | | Geometry file format | STL (both ASCII and binary) |
| | | | OBJ |
| | | | Nastran file format |
| | | Other | Deletion of overlapping generation particles |
| | | | Adjustment of the number of fluid particles to specified volume |
| | | | Comparison between scenes (Unit system, Time step, Solution method, Physical model, Object, etc.) |
| | | | Copy and paste of object setting |
| | | | Snapshot function |
| | | | Python API support |
| | Post-Processing・Visualization | Multiple scene views | |
| | | Color map to particles (by group or physical quantities) | |
| | | Image / video output | |
| | | Cross-section creation | |
| | | Vector representation of physical quantities | |
| | | Particle path-line | |
| | | Examine physical quantities of an arbitrary particle | |
| | | Extraction of particles in specified region (Box probe) | |
| | | Mixing degree (probe) | |
| | | Transformation of coordinates to make results easily visible (Coordinate transformation) | |
| | | Reference frame | |
| | | Interpolation from particle data to geometry data (Mapping) | |
| | | Estimation of physical quantities in an arbitrary coordinate (Point probe) | |
| | | Interpolation from particle data to grid data (Grid) | |
| | | Streamline | |
| | | Isosurface / Isoline | |
| | | Surface mesh generation from particle data | |
| | | ASCII conversion of results | |
| | | Flow rate measurement | |
| | | Measurement of geometry data and particles | |
| | | Visualization of simulation results「Visualization Option」※ | |

※ GPU computing, Visualization Option, and Particleworks for ANSYS require optional licenses.
※ Aeration function requires a DEM optional license and a MPS-DEM interface optional license.

| | | |
|---|---|---|
| Operation environment / Requirements | OS | Windows 8.1／Windows 10／RedHat Enterprise Linux WS 6.x／WS 7.x |
| | OpenGL | 4.0 or later |
| | CPU | Intel, AMD, x86 compatible ≧2GHz |
| | GPU(computing) | NVIDIA GeForce GTX TITAN V / TITAN X, NVIDIA Tesla P100 / V100, NVIDIA Quadro GP100 / GV100 |
| | Memory | ≧4GB |
| | HDD | ≧5GB |

※ Installation of CUDA 9.1 is required to use GPU computing. ※Particleworks requires 64-bit operating system.
※ Particleworks for ANSYS is only supported on Windows10 64bit.

# MPS – The Moving Particle Simulation Method

The MPS method was originally proposed by Prof. Koshizuka at the University of Tokyo, who is a co-founder of Prometech Software. Flexible modeling, capability of simulating complex moving boundary problems and multi-physics are the main assets of the MPS Method.

## Conventional grid method

Grid methods (e.g. FDM, FEM, FVM) use computational grids called meshes to calculate physical properties (e.g. pressure, flow velocity). Meshes need to be set in advance both in the region where fluid is present in the initial state and in the region where fluid may flow. The fluid flows from one mesh to the adjacent mesh. This is where the inflow and outflow of mass and momentum of each mesh are calculated. The following obstacles are raised with regard to such a grid method.

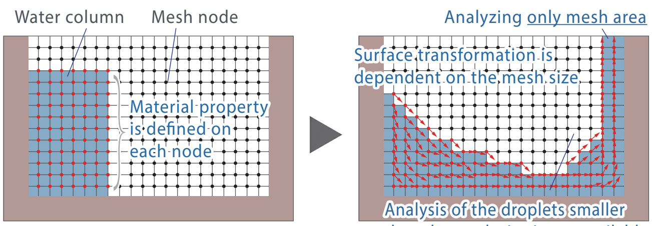
- Meshes need to be created by predicting the fluid flow areas. Calculations will fail when the mesh is not defined, or when the fluid flows in unexpected directions. Unnecessary calculations will perform when meshes are not used.
- Mesh generation for complex shapes require a lot of manhours. (e.g. automotive gearboxes, air conditioning heat exchangers.
- The mesh arrangement and density need to be set in consideration of the flow direction and flow velocity distribution etc. due to the mesh dependency in fluid analysis. Adequate knowledge and experience is required in order to carry out highly accurate calculations.
- Collapsing meshes and abnormal calculation terminations are common with analyses with large mesh deformations.

## MPS

MPS allows the fluid itself to be modeled with particles without using a grid determining space. These "particles" are "calculation points" for flow velocity and pressure that correspond to grid points for grid methods. They do not represent substances such as water droplets. These are some of the benefits of the Particle Method that reduce analysis man-hours in both design and development.

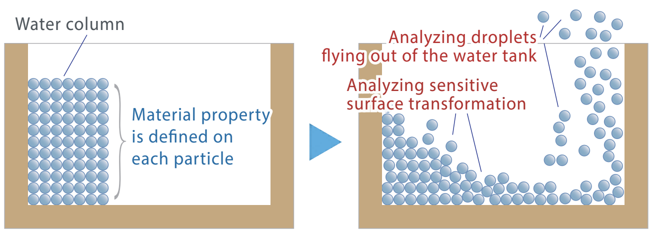
- No need to set the analysis region in advance. The particles themselves represent the flow of fluid. This is optimal for tracking conditions where fluid droplets are widely scattered. The free surface is naturally visualized by particle distribution.
- Easily modeling of the fluid parts with CAD data of the walls, even for containers and piping of complex shapes. Only the initial particle spacing needs to be specified. Particles are evenly distributed according to the specified spacing. This results in a drastically reduced preparation process of the analysis model and allows engineers to focus on analyses and result validation.
- No abnormal termination and mesh collapsing issues.

Conventional grid method




Meshes are created in the entire analysis area

MPS




Particle are created only in the water area

## References



| | | |
|---|---|---|
| **Moving Particle Semi-implicit Method: A Meshfree Particle Method for Fluid Dynamics** | Authors: Seiichi Koshizuka, Kazuya Shibata, Masahiro Kondo, Takuya Matsunaga | Publisher: ELSEVIER May/2018 |

PROMETECH.

[ Developer, Main Domestic / Global Dealer ]

## Prometech Software, Inc.

[HQ]
Hongo Dai-ichi Building, 8F, 34-3, Hongo 3-chome, Bunkyo-ku, Tokyo 113-0033, Japan
Tel: +81-(0)3-5842-4812  Fax: +81-(0)3-5842-4123

https://www.prometech.co.jp/home_en.html    E-mail: kaidai@prometech.co.jp



TENSOR
... passion 4 engineering

Bucharest, ROMANIA

Tel: +4021 444 23 78
office@tensor.ro
http://www.tensor.ro

2020.06