

# The Tool of Automatic Aerodynamics and Stress Analysis for Radiolocation Systems Based On Open-Source Codes

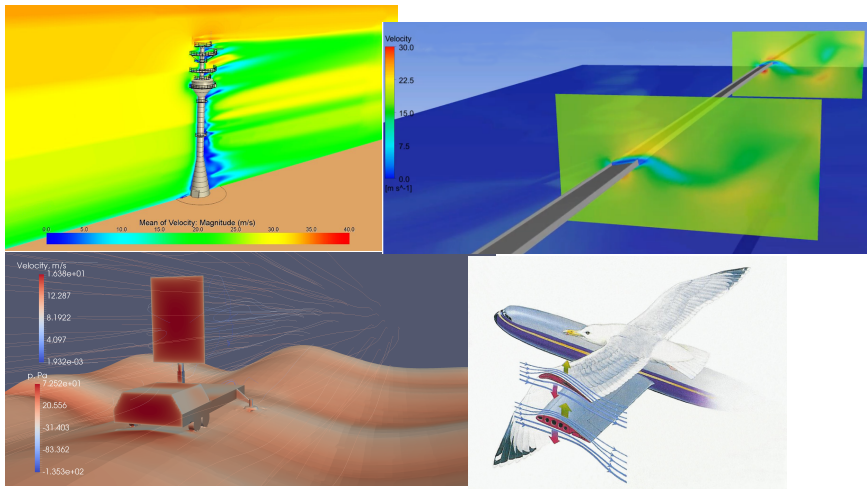
Victoria Korchagova, Matvey Kraposhin, Daniil Ryazanov, Sergei Strijhak, Kirill Vatutin, Valeriia Melnikova, Artem Konovalchik, Oleg Plaksenko, Maxim Konopelkin, Maxim Kudrov, Ivan Martynov

Ivannikov ISP RAS Open Conference

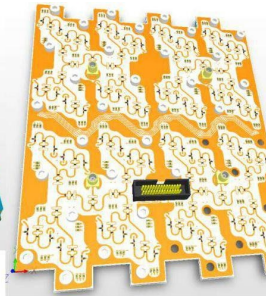
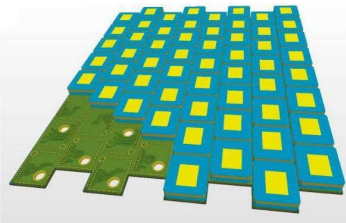
6 December, 2019

- Introduction
  - Example
  - Conditions
- Tools of Automatic Design for Radar
  - Goals
  - Capabilities
  - Composition
- Exploitation
  - Markup
  - Calculation Parameters
- Visualization
  - Multiparameter Calculation
  - Calculation on Cluster
  - Restart
- Implementation
  - Preparing
  - Formation of FE mesh
  - Formation of FE mesh
  - Aerodynamic Calculation
  - Heat Transfer Calculation
  - Strain Calculation
  - Virtualization

# Coupled Problems



# Radar



# Conditions



# Requirements

- User-friendly interface
- Usability
- Diagnostic tools
- Control of intermediate results of computations
- Simulations for various constructions:
  - aerodynamics;
  - heat state;
  - strain-stress state
- Fast computations with supercomputers
- Cross-platforming



AERO



THERMO

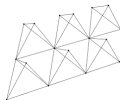


Stress



# Additional goals

- Automatic data transmission (e.g. workstation – cluster)
- Coupled work of strongly different tools
- Search of optimal algorithms for mesh building and numerical modeling
- Work process: from operations with geometry to analysis of results



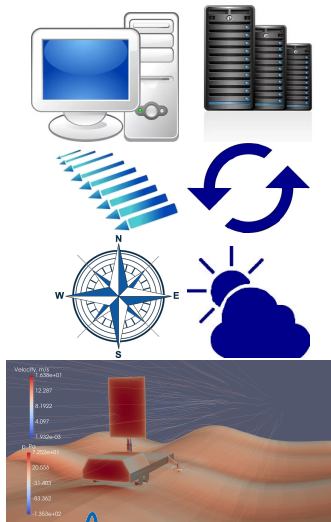
**Press**



**To Win!**

# Capabilities

- Computations both on workstation and cluster
- Simultaneous simulations for different conditions
- Restart of computations in case of input data correction
- Various set of conditions:
  - wind speed and direction according to atmospheric logarithm profile;
  - geographical position;
  - sun intensity;
  - terrain relief
- Possibility for simulations for set of positions of different rotating parts of model
- Joint usage of 1D, 2D, 3D elements in the computational model
- Flexible visualisation of results





# Composition

SALOME:	geometry
OpenFOAM:	aerodynamics
Code_Aster:	heat, strain-stress
Paraview:	visualisation
VirtualBox:	cross-platforming
PyFoam:	interface between modules
PyQt:	graphical interface
C++:	radiation
Python libs:	interface, parallelization-by-tasks

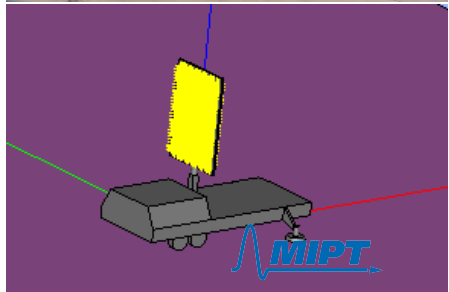


OpenFOAM

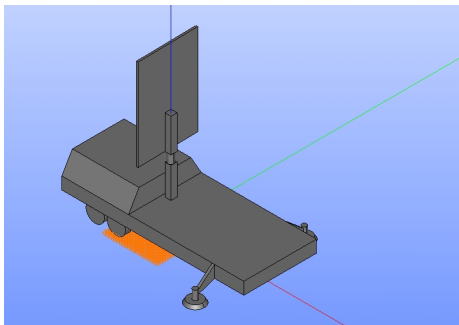


# Order of Exploitation

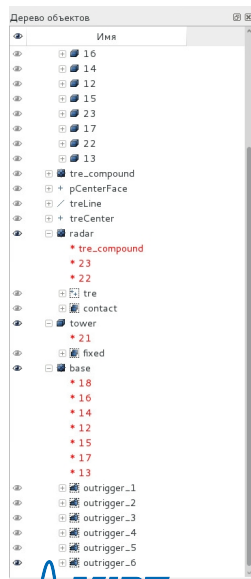
- prepare the geometry:
  - draw your own model or simplify an imported one
  - mark up the model;
- set up physical properties of the construction;
- set up the operational conditions (weather, geography...)
- set up mesh parameters;
- set up parameters for multicomputations;
- set the number of cores and the machine for computations.



# Markup

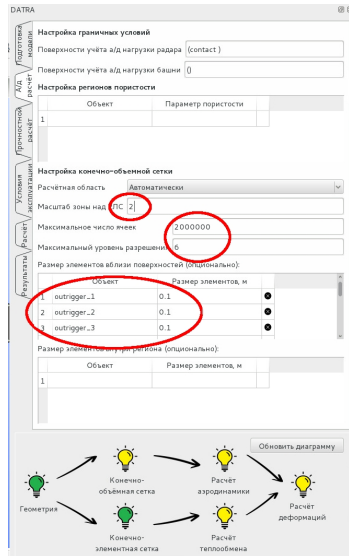


- Key parts of model (ground, TRE...)
- Rotating and static parts
- Volume groups for definition of materials
- Surface and nodal groups for boundary conditions
- Additional groups for mesh refinement

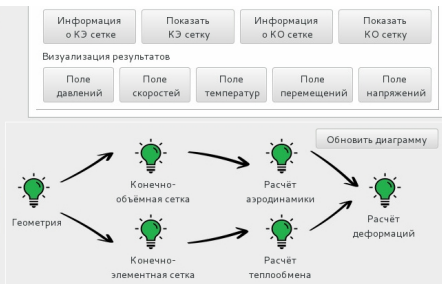


# Calculation Parameters

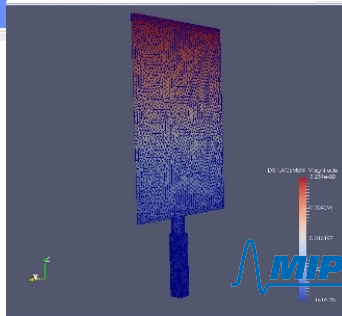
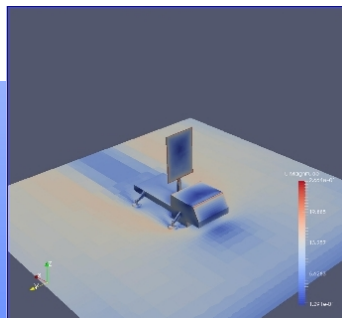
- Parameters of FEM and FVM meshes
- Operational conditions:
  - date and time (Grinwich)
  - longitude
  - wind velocity (magnitude, wind, direction)
  - cloudiness
- Boundary conditions for marked groups



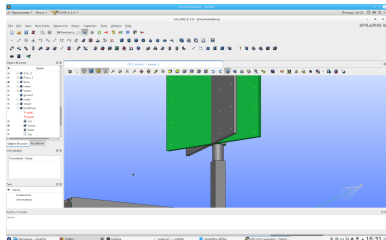
# Visualization



- Pressure field
- Velocity field and streamlines
- Temperature
- Stress
- Displacement



# Multiparameter Calculation



Combination of parameters:

- positions of rotating parts;
- wind parameters;
- sun parameters.

Глоб. м

Угол поворота антенны (15 30 45 60 75 90)

А/Д

Горизонтальный расчёт

----- 0

----- 0

Сформировать варианты расчёта

Условия эксплуатации

Расчёт

Результаты

Таблица расчётных случаев

ол направлен рости ветра, г	одуль скорост ветра, м/с	/гол поворота антенны, град	Облачность
0 0.0	8.0	0	0
1 0.0	8.0	15	0
2 0.0	8.0	30	0
3 0.0	8.0	45	0
4 0.0	8.0	60	0
5 0.0	8.0	75	0
6 0.0	8.0	90	0

Запуск расчёта

Место расчёта Рабочая станция

Запуск базового расчёта

Число ядер 8

Рестарт

Запуск  
вариантных  
расчётов

# Calculation Using Cluster

Commands: via ssh

File exchange: scp

- On workstation:
  - Preparation of computational cases
  - FEM mesh generation
  - Computation of heat flows through radar surface
- On cluster:
  - FVM mesh generation (MPI)
  - Computation of aerodynamics (MPI)
  - Computing of heat transfer inside radar (tasks)
  - Computing of stress-strain state (tasks)



# Restart of Calculations

**Запуск расчёта**

Место расчёта: Рабочая станция

Число ядер: 8

Запуск базового расчёта

Рестарт

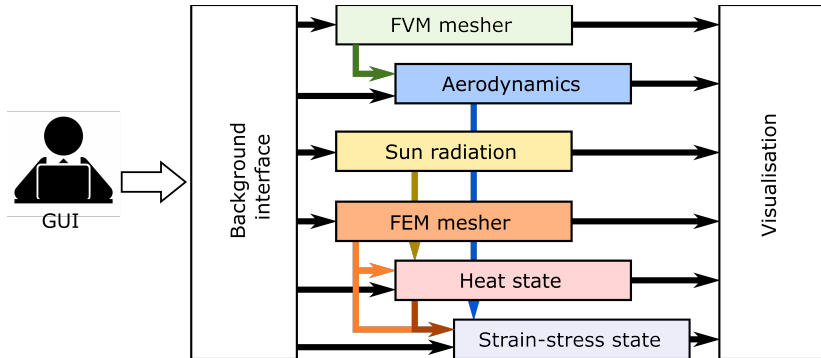
Запуск вариантных расчётов

Обновить диаграмму

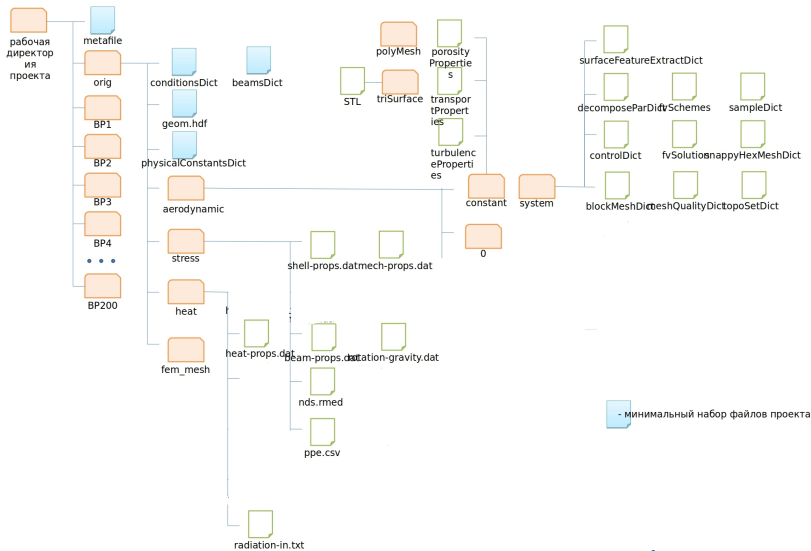
```
graph LR; G[Геометрия] --> CO[Конечно-объёмная сетка]; G --> CE[Конечно-элементная сетка]; CO --> RA[Расчёт аэродинамики]; CE --> RT[Расчёт теплообмена]; RA --> RD[Расчёт деформаций]; RT --> RD;
```



# Scheme of Data Flows



# Preparatory stage



# Finite Element mesh formation



The screenshot displays the SALOME 8.3.0 software interface. The main window shows a 3D model of a mechanical part with a blue finite element mesh. The interface includes a top menu bar, a toolbar, and a Pipeline Browser on the left. On the right, the DATRA (Data Transfer) panel is visible, containing the following text:

Ход расчёта  
Пор. программы:  
fader (required) found  
tre (required) found  
tower (required) found  
ground (required) found  
LocalCS (required) found  
base (optional) found

Идет расчёт № \_\_\_ из \_\_\_     

Результаты расчёта  
Номер варианта расчёта для визуализации: 0

Визуализация результатов

Diagram illustrating the calculation workflow:

```
graph TD; G[Геометрия] --> CO[Конечно-объёмная сетка]; G --> CK[Конечно-элементная сетка]; CO --> AV[Расчёт аэродинамики]; CK --> TD[Расчёт теплообмена]; AV --> CD[Расчёт деформаций]; TD --> CD;
```

# Finite Volume mesh formation



The screenshot displays the SALOME 6.3.0 software interface. The main window shows a 3D wireframe mesh of a room with a central object. The interface includes a toolbar at the top, a left sidebar with 'OpenFOAMReader', and a right sidebar with a 'DATA' panel. The 'DATA' panel shows the calculation progress and options for visualization.

**DATA**

Ход расчёта  
Лог программы:  
gadi (required) found  
tre (required) found  
tome (required) found  
ground (required) found  
LocalCS (required) found  
base (optional) found

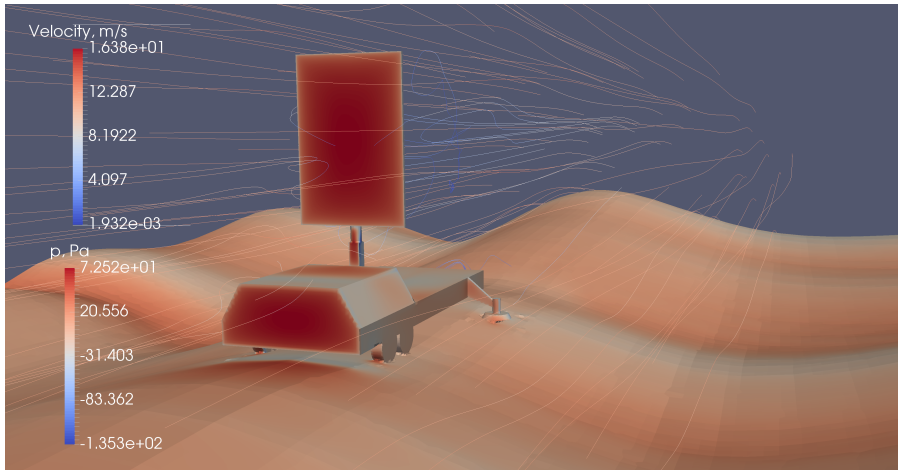
Идет расчёт № \_\_\_\_ из \_\_\_\_

Результаты расчёта  
Номер варианта расчёта для визуализации: 0

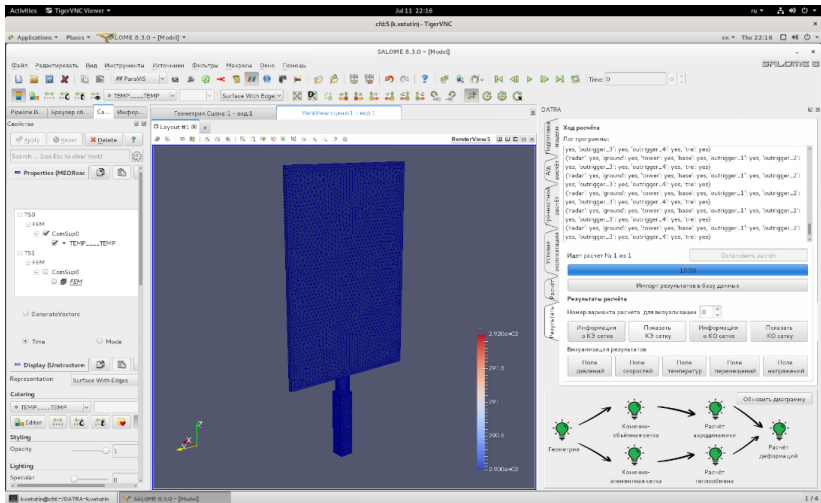
Визуализация результатов

Геометрия → Конечная элементная сетка → Расчёт элементов сетки → Конечная сетка → Расчёт температуры → Расчёт деформаций

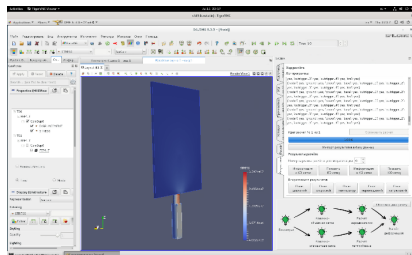
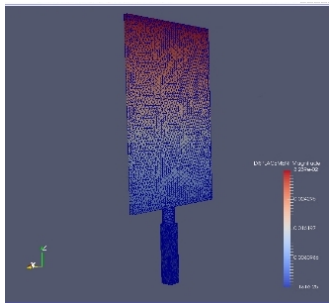
# Aerodynamic Calculation



# Heat Transfer Calculation



# Strain Calculation



- GUI
  - Navigation
  - Simulation control
  - Notices, advises, tips
- Simulation on cluster
- Multiparameter Calculation
  - Interpolation
- Coupled work of strongly different tools