11th SPINE meeting , ESA/ESTEC, 04/2007

Spacecraft Plasma Interaction System
Course and Practical Work
Part 1: Introduction

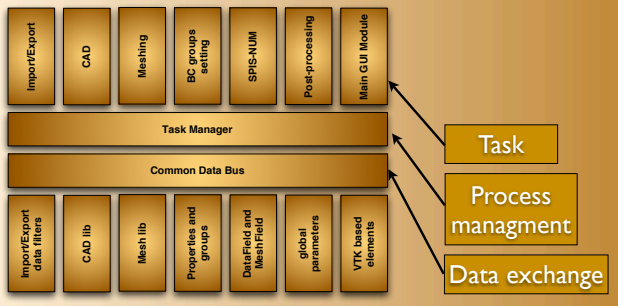
J.Forest(1), S.Jourdain(1), S. Bagnier(1),
contact@artenum.com
(1) Artenum, France;

This lecture is given in parallel of the Practical Work: Open the project "SPINE_labWork1.spis"

For further information, please see the
SPIS Technical Note 01: USER MANUAL & QUICK START

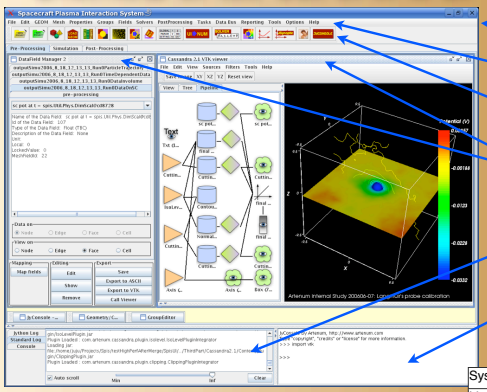
Global structure of SPIS

- SPIS-UI is a global and modular modelling framework (IME)
  - Controlling SPIS-NUM as simulation kernel
  - Offering / controlling several pre/post-processing tools
  - Transfer / convert data



We will focus here on the use in interactive mode (GUI).

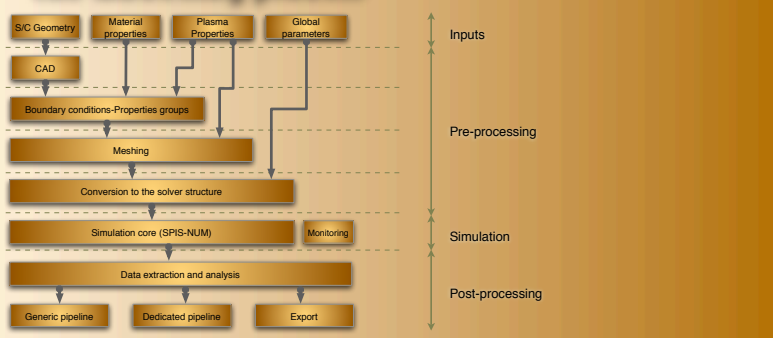
Overview of the GUI of SPIS-UI



To launch the framework:
1) go into the SPIS_ROOT/SpisUI directory
2) Execute the launching script:

System	Launching script
Linux	spis_tasks_gui.sh
Apple MAC OS-X	spis_tasks_guiOSX.sh
MS Windows NT/2000/XP	spis_tasks_gui.bat

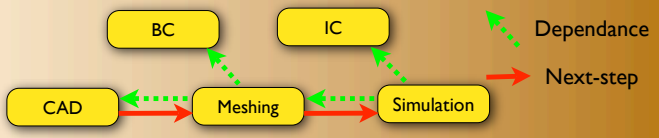
The modelling process



The Tool Bar order follows the most common modelling process!

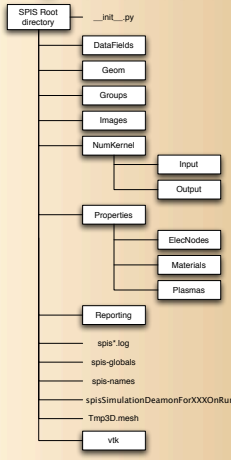
Task Manager and orchestration

- SPIS-UI includes an orchestration module called "TaskManager", that:
  - Check the dependencies of a task and perform them before
  - Run the framework according to scripts called "scenarios"
- Quite similar behaviour than Make



CAUTION! Normally the dependance tree is performed only one time, after what the framework perform only the requested action. If you wish re-init the dependance tree, please select the Tasks->Reset Done Tasks menu.

Project oriented approach and structure

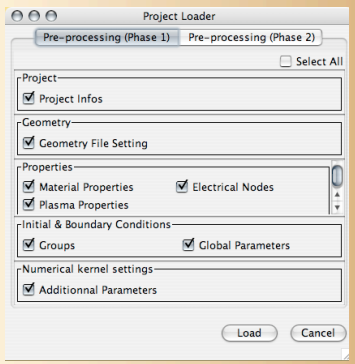


- All data are stored in a "project directory"
  - A modular structure with sub-directories for each type of data
  - Most of the data can be save/loaded independently
  - Most of the data saved as Python scripts
- Create a new project or open an existing one should be the first thing to do.

➡ To create a new project just click on

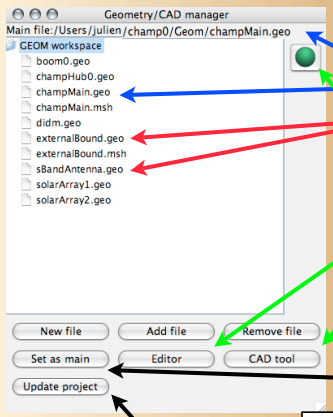
➡ To open an existing project click on (and select the directory)

Select data to load



- Data can be loaded in two phases
  - Phase 1: Minimum set to define a complete model
    - CAD
    - Properties
    - Groups
    - Gobal parameters
  - Phase 2: Pre-processed model
    - Mesh
    - Pre-processed DataFields
- Data can loaded independently

CAD module and Geometric Manager (1)



- The geometrical model can be defined by several files
  - One main file
  - Several sub-systems
- Possibility to edit files through
  - Using pre-defined shape generators
  - Text editor
  - Using Gmsh
- With SPIS 3.7 and later, the Geometry Manager can be use as standalone application

1) Don't forget to set your main file!!

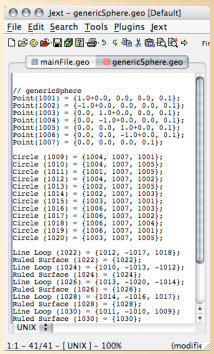
2) Don't forget to update your project regularly!!

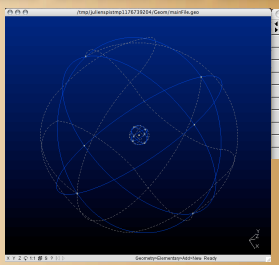
3) Don't forget to load your main CAD file into the framework!!

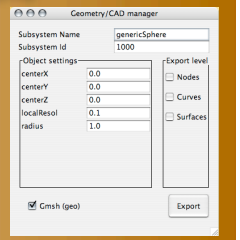
LOAD GEOM

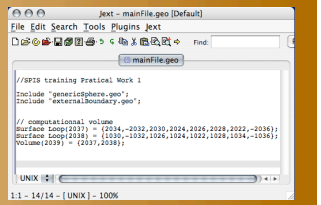
SAVE PROJECT

CAD module and Geometric Manager (2)





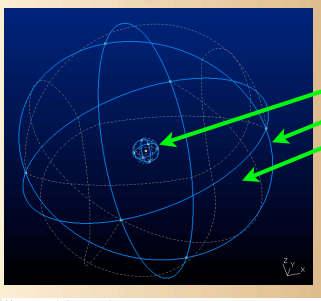




CAD done with Gmsh (.geo files)

BREP based approach

Minimum geometrical system



- The geometrical model must at least have:
  - One inner boundary (i.e S/C)
  - One external boundary (i.e far plasma)
  - One computational volume
- And the corresponding Groups (*Physicals* for Gmsh):
  - One surface group of the S/C
  - One surface group for the external boundary
  - One group of volume for the computational domain

```
//SPIS training Pratical Work 1

Include "genericSphere.geo";
Include "externalBoundary.geo";


// computationnal volume
Surface Loop(2037) = {2034,-2032,2030,2024,2026,2028,2022,-2036};
Surface Loop(2038) = {1030,-1032,1026,1024,1022,1028,1034,-1036};
Volume(2039) = {2037,2038};


Physical Surface(2040) = {1032,1034,1026,1028,1024,1030,1036};
Physical Surface(2041) = {2028,2026,2034,2032,2030,2036,2024,2022};
Physical Volume(2042) = {2039};
```

See Gmsh documentation to define geo elements and groups

The Physical of Volume should be defined manually with the text editor!!

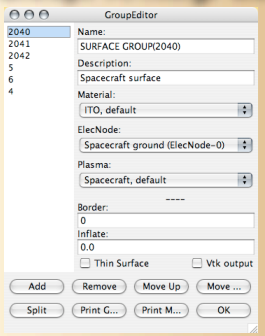
Meshing rules and tricks

```
// genericSphere
Point(1001) = {1.0+0.0, 0.0, 0.0, 0.1};
Point(1002) = {-1.0+0.0, 0.0, 0.0, 0.1};
Point(1003) = {0.0, 1.0+0.0, 0.0, 0.1};

resLocal1 = 0.2;

Point(1004) = {0.0, -1.0+0.0, 0.0, resLocal1};
Point(1005) = {0.0, 0.0, 1.0+0.0, resLocal1};
Point(1006) = {0.0, 0.0, -1.0+0.0, resLocal1};
Point(1007) = {0.0, 0.0, 0.0, resLocal1};
```

- Unstructured mesh:
  - Tethraedrons based
  - Mesh size can be adapted locally, according to the local resolution
- Typical practical grid between 10^3 and 4.10^3 cells
- Stability of the mesher may be sensitive to the local mesh size
  - Try progressive refinements
- Choice of the mesh size depends on the choice of the model, see SPIS-NUM documentation

1st try outside the framework, in Gmsh in GUI mode only.

Groups settings and properties attribution



- First load the properties
  - From the default catalogue
  - Or from you own properties files
- Load the Group Editor
- Attribute properties to each group

Take care to the priority order !! Volume and default groups should have the lowest priority! Use the up/down buttons to set the priority.

Take care to set the right properties to each group!! See table below.

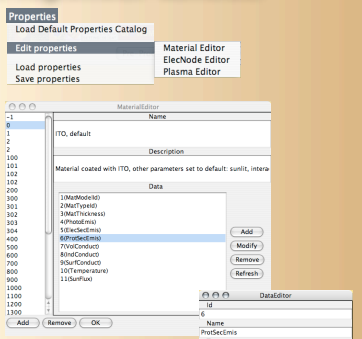
Object or zone to be marked	*Type of physicals*
Spacecraft surface	Surface
External boundary of the computational volume	Surface
Computational volume	Volume

Physical	*Material*	*Electric*	*Plasma*
Spacecraft surface	ITO	ElecNode-0	Spacecraft, default
External boundary	None	None	Boundary default
Computational volume	None	None	Plasma Model in Volume, default

SAVE PROJECT

To see the groups: Select the Groups->Show groups menu

Properties Edition

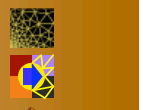


- Each property can be modified
- Three main types of properties
  - **Material**, that defines material properties
  - **ElecNode**, that defines nodes for the internal electrical balance
  - **Plasma**, that defines numerical properties needed by the plasma model (e.g. flags)
- Edited properties can be saved with the project or individually.

SAVE PROJECT

Meshing, groups convertion and DataFields mapping

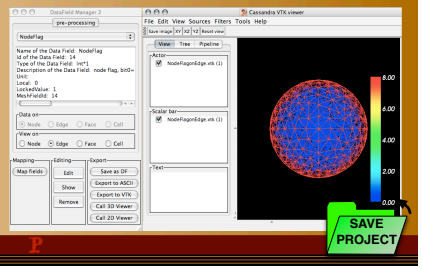
- Local properties need to be deployed on the grid according the groups definitions. For this you should:
  - Mesh the system and load it into the framework
  - Convert the Geometrical Groups into Mesh Groups
  - "Map" or deployed the local properties on the grid and build the corresponding fields or *DataFields*.



At this level the whole pre-processing phase is done for local parameters and the geometrical setting.

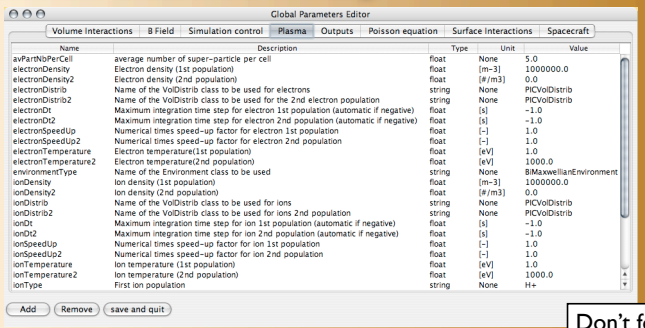
To see the mesh, select the Mesh->View Mesh menu

To see the deployed pre-processing fields, use the DataFields Manager with the following icon



SAVE PROJECT

Global Parameters Setting



Name	Description	Type	Unit	Value
avPartNbPerCell	average number of super-particle per cell	float	None	5.0
electronDensity	Electron density (1st population)	float	[m-3]	1000000.0
electronDensity2	Electron density (2nd population)	float	[#/m3]	0.0
electronDistrib	Name of the VolDistrib class to be used for electrons	string	None	PICVolDistrib
electronDistrib2	Name of the VolDistrib class to be used for the 2nd electron population	string	None	PICVolDistrib
electronDt	Maximum integration time step for electron 1st population (automatic if negative)	float	[s]	-1.0
electronDt2	Maximum integration time step for electron 2nd population (automatic if negative)	float	[s]	-1.0
electronSpeedUp	Numerical times speed-up factor for electron 1st population	float	[-]	1.0
electronSpeedUp2	Numerical times speed-up factor for electron 2nd population	float	[-]	1.0
electronTemperature	Electron temperature(1st population)	float	[eV]	1.0
electronTemperature2	Electron temperature(2nd population)	float	[eV]	1000.0
environmentType	Name of the Environment class to be used	string	None	BiMaxwellianEnvironment
ionDensity	Ion density (1st population)	float	[m-3]	1000000.0
ionDensity2	Ion density (2nd population)	float	[#/m3]	0.0
ionDistrib	Name of the VolDistrib class to be used for ions	string	None	PICVolDistrib
ionDistrib2	Name of the VolDistrib class to be used for ions 2nd population	string	None	PICVolDistrib
ionDt	Maximum integration time step for ion 1st population (automatic if negative)	float	[s]	-1.0
ionDt2	Maximum integration time step for ion 2nd population (automatic if negative)	float	[s]	-1.0
ionSpeedUp	Numerical times speed-up factor for ion 1st population	float	[-]	1.0
ionSpeedUp2	Numerical times speed-up factor for ion 2nd population	float	[-]	1.0
ionTemperature	Ion temperature (1st population)	float	[eV]	1.0
ionTemperature2	Ion temperature (2nd population)	float	[eV]	1000.0
ionType	First ion population	string	None	H+

Don't forget to save and quit!

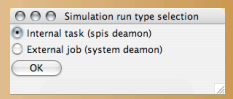
- Define the global (i.e. unlocalised properties)
- See the "How-to control NUM from UI" for the meaning of each parameter

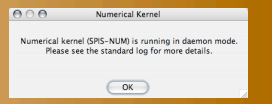
SAVE PROJECT

At this level,the whole pre-processing is done.

Simulation Control

- SPIS-UI is a generic framework, data need to be converted into the SPIS-NUM structure by clicking on UI NUM
- The SOLVER $f(x, y, z, t) = 0$ icon launch the simulation process.
- Simulation can be done in several mode:
  - Iterative mode (default)
  - batch mode





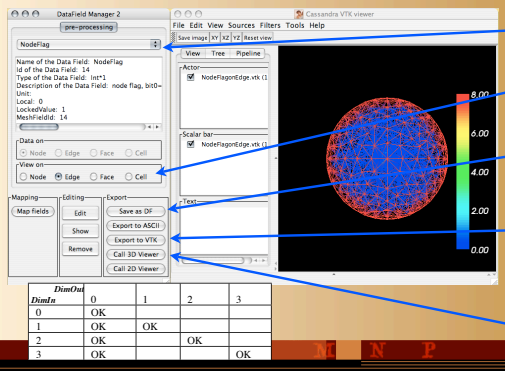
Several information and dialogue boxes should appears.

Check the Jython and the standard logs and take care to possible errors messages.

You can extract data during the simulation by selecting the Solver->SPIS-NUM->Solver:Extract Data menu.

Data extraction and analysis

- End of the simulation, output data are returned to the framework.
- Use the DataField Manager to extract and convert them
  - Location of data may require to be converted
  - Several viewers: Cassandra, Paraview for 3D, 2DPlot and JSynoptic for 2D.



Select your DF

Select the cell conversion

For 3D data export to VTK

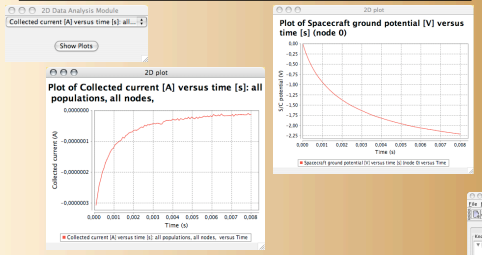
For 3D data call the default viewer (Cassandra)

For 2D select the 2D plot tool

SAVE PROJECT

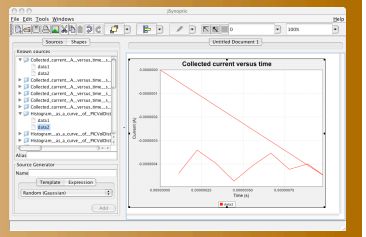
Post-processing tools: 2D data

The simple 2D viewer is called by clicking on



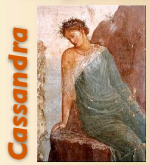
1) Select the data plot
2) Click on Show Plots button
3) On the plot, crt+click leads to a contextual menu to:
   - set the plot properties
   - print the plot
   -...

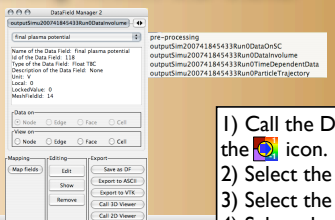
JSynoptic can be called by clicking on JSynoptic



JSynoptic is an advanced 2D plot facility. Please see the JySynotpic documentation for further details.

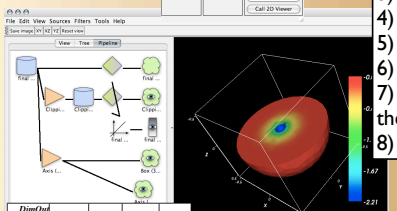
Post-processing tools: 3D data





See the Cassandra's Tutorial for further information.

1) Call the DataField Manager by clicking on the icon.
2) Select the type of DataField (e.g post-pro)
3) Select the DataField to display
4) Select the type of cells conversion
5) Convert the data into VTK data set.
6) Call Cassandra, by clicking on the icon.
7) In Cassandra, with File-> open menu, select the right file
8) Perform the post-processing (e.g cutting)



DimOut / DimIn	0	1	2	3
0	OK			
1	OK	OK		
2	OK		OK	
3	OK			OK

You can also use Paraview, with the Postprocessing->3D View-> Paraview menu.

ARTENUM

End of the first part

P N M N P