

USER MANUAL | REGISTERED USER

SHOCK VML Portal

Date: 29th April 2014

© 2014 Sprinx Systems, a.s.



CONTENTS

INTRODUCTION.....	3
PURPOSE OF THIS DOCUMENT	3
SW REQUIREMENTS.....	3
WEB INTERFACE	4
MAIN PAGE	4
REGISTRATION AND LOG IN	5
PROJECTS AND SIMULATIONS	7
VISUALISATION WIZARD	10
BASIC OVERVIEW	12
WORKING WITH HELPER.....	13
VISUALIZATION STEP BY STEP.....	13
FAVOURITES	23
VISUALIZATION OUTPUTS	24
SINGLE TIME	24
TIME RANGE.....	26
DATA DOWNLOAD	29
DATA UPLOAD & CONVERSIONS	31

INTRODUCTION

PURPOSE OF THIS DOCUMENT

This document is intended for users who want to use the web portal **Virtual Mission Laboratory** (hereinafter only **VML**). It includes instructions, use cases and describes all features of the portal from the viewpoint of a **registered user**.

SW REQUIREMENTS

For proper and full functionality of VML portal, you need to use a web browser with JavaScript and HTML5.

Recommended web browsers:

- Mozilla Firefox (version 19 or higher)
- Chrome (version 25 or higher)
- Internet Explorer (version 10 or higher)

WEB INTERFACE

MAIN PAGE

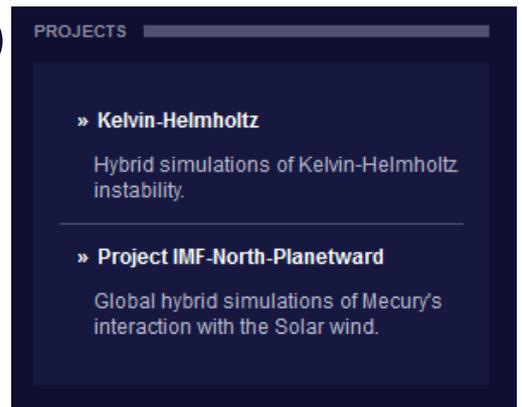
Main page contains 4 basic elements:

1. **Main menu** for basic navigation on web page
2. **List of projects** includes all projects in the VML portal and their short description
3. **Contact | RSS feed** for the possibility to contact us or subscribe for news about new projects and simulations on the portal
4. **List of new simulations** includes 5 newly added simulations

1



2



3



4



REGISTRATION AND LOG IN

To log in you need to have a user account. User account is free and available for everyone. The procedure for registration is as follows.

1



2

A screenshot of the login form. At the top right is a 'LOG IN' button. Below it, the text reads: 'Please, provide your login and password. You will need cookies enabled to log in successfully.' Below this is the text: 'If you don't have an account GO HERE TO REGISTER.' A hand cursor is pointing to the 'REGISTER' link. There are two input fields: 'Login' and 'Password'. Below the 'Password' field is a checkbox labeled 'Remember me for 1 month'. At the bottom is a blue 'Log In' button.

Note: If you already have a user account, simply fill in your credentials (**e-mail** and **password**) and click the **Log In** button.

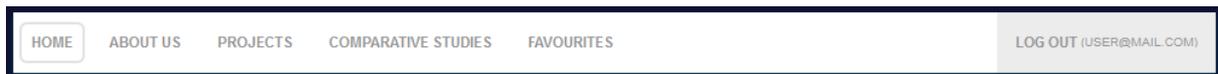
In this case skip the following step.

3

Fill in information about you. When you finish click the **Register** button. Then you will be automatically logged in.

A screenshot of the registration form. At the top left is a 'REGISTER' button. Below it, the text reads: 'Please, provide your registration information. You will need cookies enabled to log in successfully.' Below this is the text: 'If you already have an account [go here to log in.](#)' There are four input fields: 'Name: *' with 'Username', 'Email: *' with 'user@mail.com', 'Password: *' with '.....', and 'Institution: *' with 'Sprinx Systems'. Below the fields is the text: '* indicates a required field'. At the bottom is a blue 'Register' button. A hand cursor is pointing to the 'Register' button.

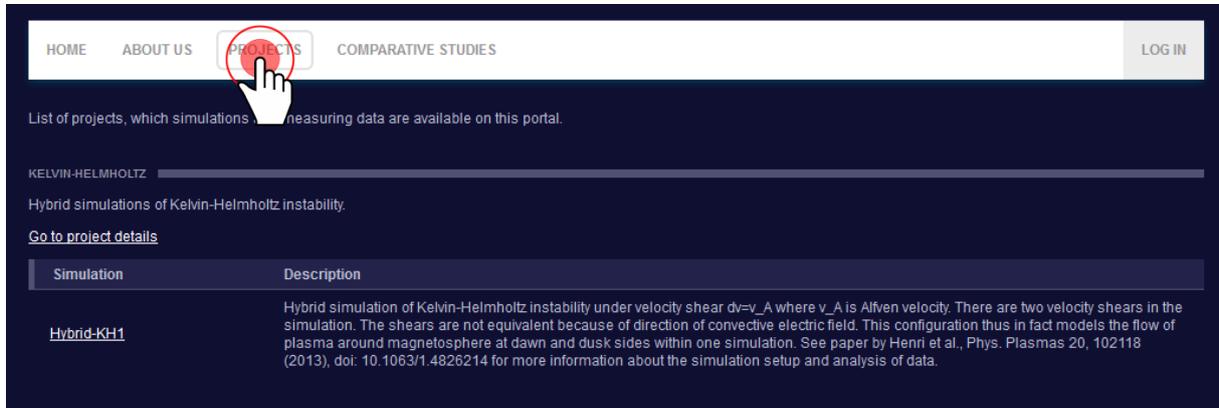
If your registration is successful main menu will change to the following form.



Note: Some of the following procedures and figures are shown from the perspective of an unlogged user. They are the same for both unlogged and logged user.

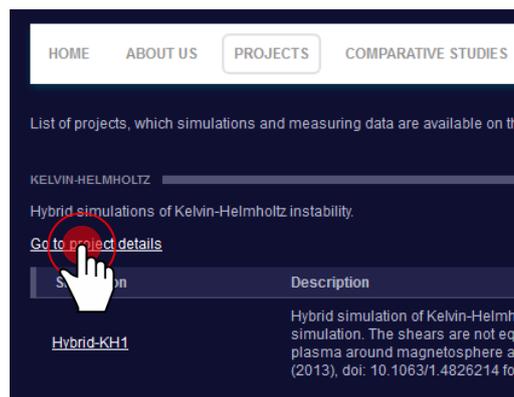
PROJECTS AND SIMULATIONS

An overview of projects and simulations that VML portal contains can be found in the **Project** tab in main menu. It includes all projects and their simulation including detailed descriptions.



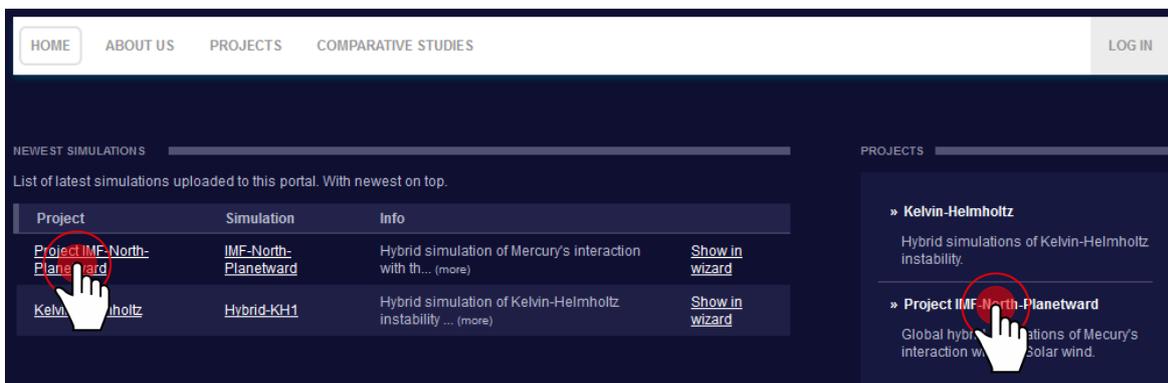
If you want to go **project details**, use one of the following ways:

1 From **Projects** tab



Or

2 From **Home** tab



Page with **project details** shows all information about project, figures and list of their simulations including basic information and available products.

The screenshot displays the SHOCK Virtual Mission Laboratory Portal interface. At the top, the SHOCK logo is accompanied by the text 'Virtual Mission Laboratory Portal'. A navigation bar includes links for HOME, ABOUT US, PROJECTS (highlighted), and COMPARATIVE STUDIES, along with a LOG IN button. The main heading is 'Project IMF-North-Planetward'. Below this, there are two tabs: 'DETAILED DESCRIPTION' and 'PROJECTS'. The 'DETAILED DESCRIPTION' tab is active, showing a text block about three-dimensional hybrid simulations of Mercury's magnetosphere. To the right, under the 'PROJECTS' tab, there are two buttons: 'Kelvin-Helmholtz' and 'Project IMF-North-Planetward'. Below the description is a 'PROJECT IMAGES' section containing three visualizations: a 3D model of Mercury with its magnetosphere, a 2D heatmap of the magnetosphere's cross-section, and a line graph showing the ratio B_x/B_0 versus X/R_{H_0} . At the bottom, a 'LIST OF SIMULATIONS' section features a card for 'IMF-North-Planetward' with a plus sign icon on the right. The card text includes: 'Hybrid simulation of Mercury's interaction with the solar wind under northward-planetward interplanetary magnetic field.' and 'Available products: B, B_x, B_y, B_z, Density'. The footer contains the author's name, contact information, and logos for various partner organizations including ESA, Queen Mary University of London, CNRS, and Sprinx Systems.

For more detailed information about chosen simulation click on the drop down element.

This is a close-up view of the 'LIST OF SIMULATIONS' section. It shows a single card for the 'IMF-North-Planetward' simulation. The card contains the same text as seen in the previous screenshot: 'IMF-North-Planetward', 'Hybrid simulation of Mercury's interaction with the solar wind under northward-planetward interplanetary magnetic field.', and 'Available products: B, B_x, B_y, B_z, Density'. A red circle highlights a plus sign icon on the right side of the card, with a hand cursor pointing to it, indicating that clicking this icon will expand the simulation details.

Then you can find information about all available products, coordinate system, initial conditions and parameters of the simulation. There are also predefined wizard settings (**simulation presets**). Presets usually are interesting settings defined by administrator of the project.

LIST OF SIMULATIONS

IMF-North-Planetward

Hybrid simulation of Mercury's interaction with the solar wind under northward-planetward interplanetary magnetic field.

Available products: B, B_x, B_y, B_z, Density

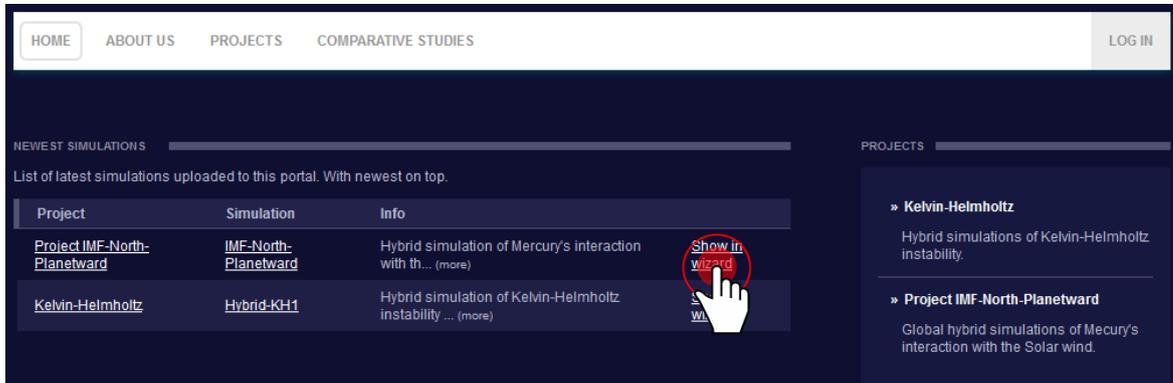
<p>COORDINATES</p> <p>Coordinate system is centered in Mercury's center and unit of length is Mercury's radius. Axis X is parallel to solar wind flow direction; axis Z is parallel to Mercury's dipole axis; axis Y completes right-handed system.</p>	<p>INITIAL CONDITIONS</p> <p>Plasma conditions in (background) unperturbed solar wind are as follows: Magnetic field is northward-planetward, $B=(0.94, 0, 0, 0.34)$ in simulation units. Plasma flow is super-Alfvenic, $v=(4v_A, 0, 0)$. Proton kinetic to magnetic pressure ratio is $\beta_p=0.5$.</p>	<p>PARAMETERS</p> <p>Grid size: $N_x=594, N_y=N_z=286$; Time step: $dt=0.01$ in units of inversed proton gyrofrequency. Cell size: $dx=0.4, dy=dz=1$ in units of proton inertial length; Mercury's radius: $R=15.9$ in units of proton inertial length.</p>
--	--	--

<p>SIMULATION PRESETS</p> <ul style="list-style-type: none"> ■ Density - example 2 ■ Magnetic field - example 1 	<p>PRODUCTS IN SIMULATION</p> <ul style="list-style-type: none"> ■ B - Magnitude of magnetic field ■ B_x - Magnetic field component in the direction of the solar wind flow ■ B_y - Magnetic field component in the direction of Mercury's orbital motion ■ B_z - Magnetic field component in the direction of Mercury's dipole axis ■ Density - Proton charge density
--	--

VISUALISATION WIZARD

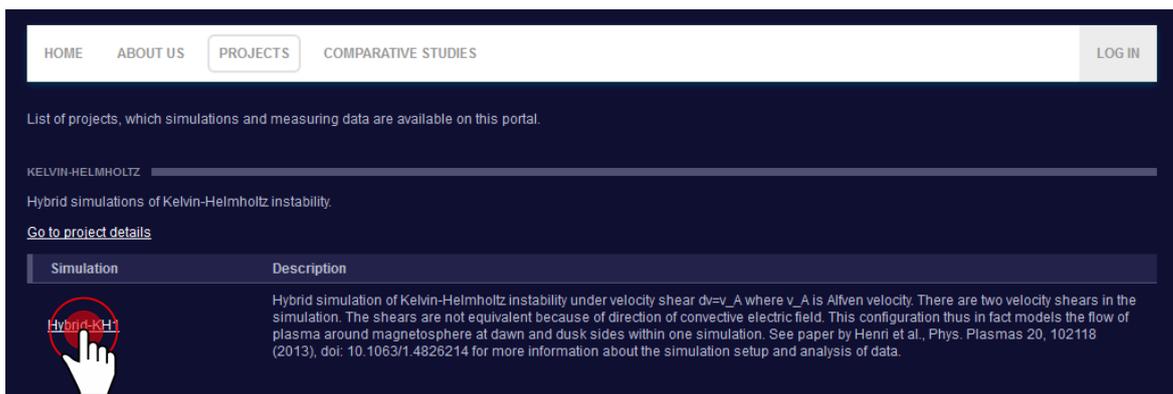
There are 4 ways how to get into visualization wizard of the simulation:

1 From Home tab



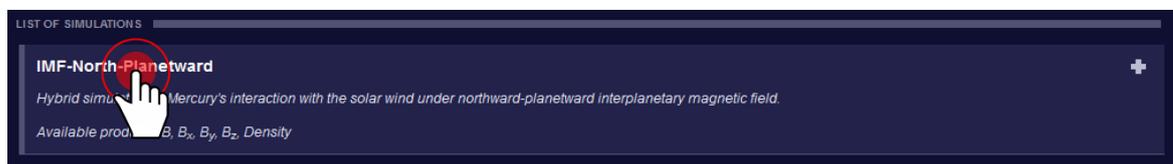
Or

2 From Projects tab



Or

3 From Simulation list in the project details page



Or

- 4 From unrolled **Simulation details** in the project details page
You can choose one of the simulation preset.

LIST OF SIMULATIONS

IMF-North-Planetward

Hybrid simulation of Mercury's interaction with the solar wind under northward-planetward interplanetary magnetic field.

Available products: B, B_x, B_y, B_z, Density

<p>COORDINATES</p> <p>Coordinate system is centered in Mercury's center and unit of length is Mercury's radius. Axis X is parallel to solar wind flow direction; axis Z is parallel to Mercury's dipole axis; axis Y completes right-handed system.</p>	<p>INITIAL CONDITIONS</p> <p>Plasma conditions in (background) unperturbed solar wind are as follows: Magnetic field is northward-planetward, B=(0.94,0.0,0.34) in simulation units. Plasma flow is super-Alfvenic, v=(4v_A,0,0). Proton kinetic to magnetic pressure ratio is beta_p=0.5.</p>	<p>PARAMETERS</p> <p>Grid size: Nx=594, Ny=Nz=286; Time step: dt=0.01 in units of inversed proton gyrofrequency. Cell size: dx=0.4, dy=dz=1 in units of proton inertial length; Mercury's radius: R=15.9 in units of proton inertial length.</p>
<p>SIMULATION PRESETS</p> <ul style="list-style-type: none"> ■ Density - example 2 ■ Magnetic field - example 1 	<p>PRODUCTS IN SIMULATION</p> <ul style="list-style-type: none"> ■ B - Magnitude of magnetic field ■ B_x - Magnetic field component in the direction of the solar wind flow ■ B_y - Magnetic field component in the direction of Mercury's orbital motion ■ B_z - Magnetic field component in the direction of Mercury's dipole axis ■ Density - Proton charge density 	

BASIC OVERVIEW

Information about the simulation

Allow/Disable helper

Product selection

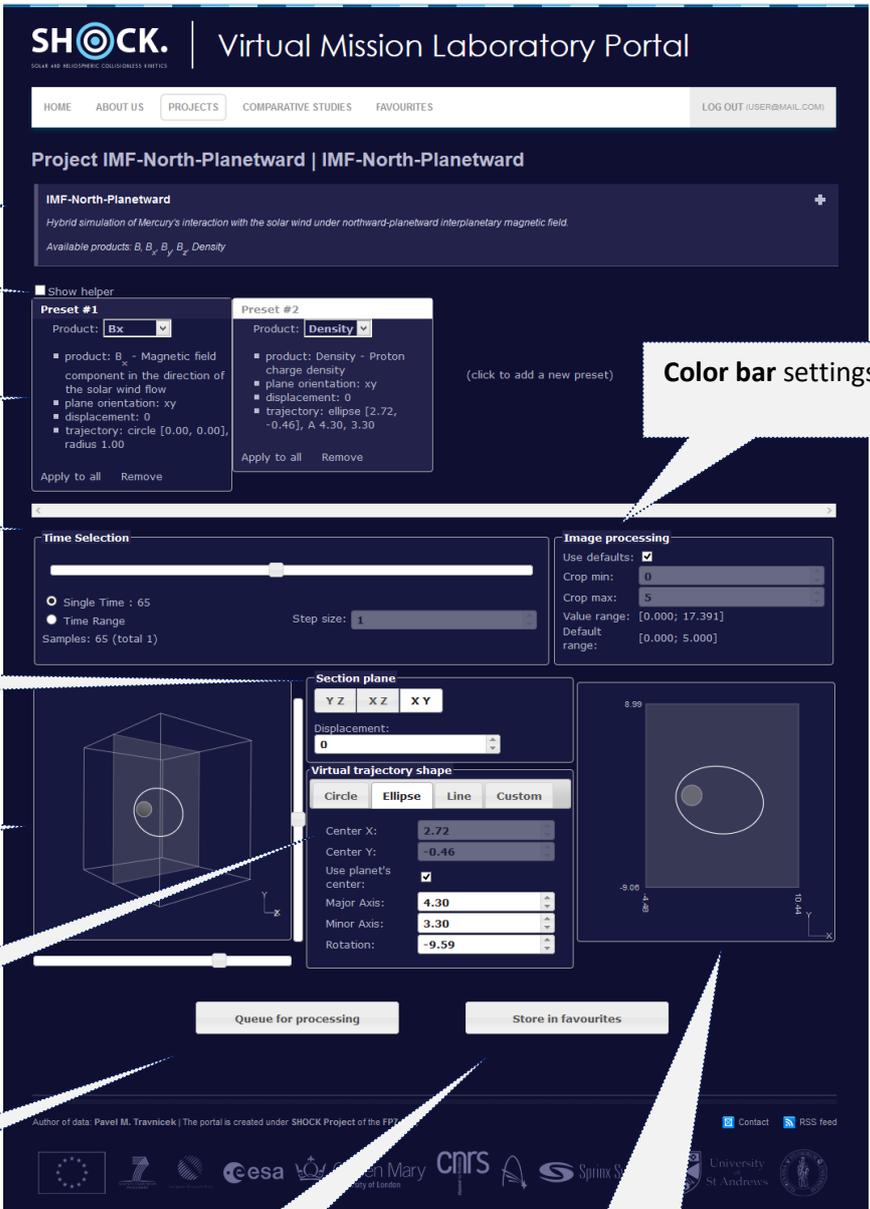
Time selection

Plane selection

Three-dimensional situational picture

Trajectory shape and parameters

Queue for processing button



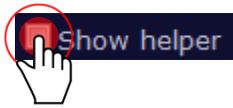
Color bar settings

Store in favourites button

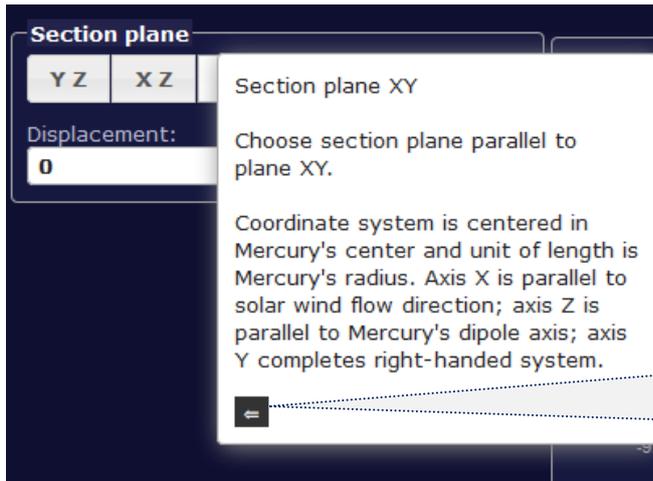
Two-dimensional situational picture

WORKING WITH HELPER

For new users we highly recommend to **allow helper**.



After clicking on the element help guide pops up with additional information and what to do in this step (similarly as shown below).



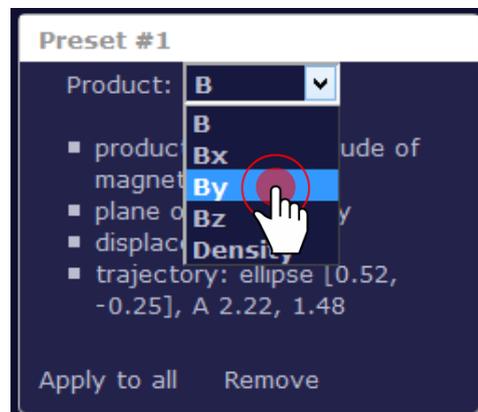
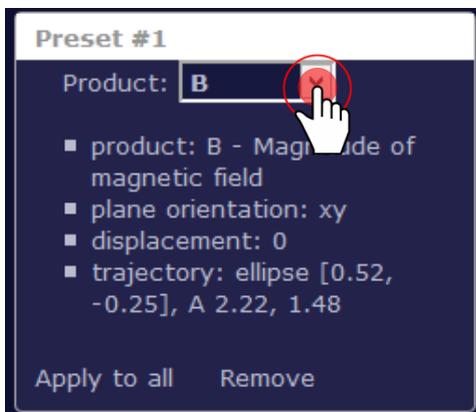
Note: If there is a button in the helper window you can change settings by click on them or move to the next step.

VISUALIZATION STEP BY STEP

1

Product selection

Choose required **product** from the list. The term **product** means a physical quantity that you want to visualize.



Preset #1

Product: **By** ▼

- product: B_y - Magnetic field component in the direction of Mercury's orbital motion
- plane orientation: xy
- displacement: 0
- trajectory: ellipse [0.52, -0.25], A 2.22, 1.48

Apply to all Remove

(click to add a new preset)

Here you can see information about product and chosen parameters.

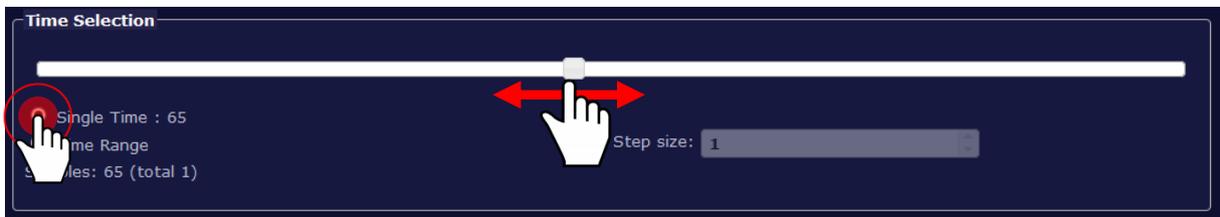
This button applies the settings of this preset to all others. It means plane orientation, displacement, trajectory shape and parameters and color bar settings.

For removing product from visualization use this button.

If you want to visualize more than one product, click here.

2 Time selection

Here you have 2 options. First one is to select only one time. Then all the products will be shown in this **single time** (static figure). Select time by moving with the time cursor.



Or

The second option is to set **time range**. Then all the products will be shown in this time range (animation). Select the time by moving start time and end time cursor.



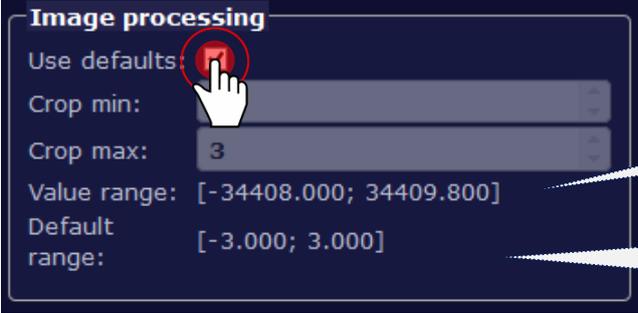
Selected time range and available samples

If there are too many samples in the dataset you can choose to calculate every n^{th} sample.

3 Color bar settings

In this step you select settings for color range of the resulting visualization. This means that you select a range of values that can be interpreted by different colors. Values outside this range will be interpreted by "outer" colors.

You can use **defaults setting** that were set by the administrator of the simulation.



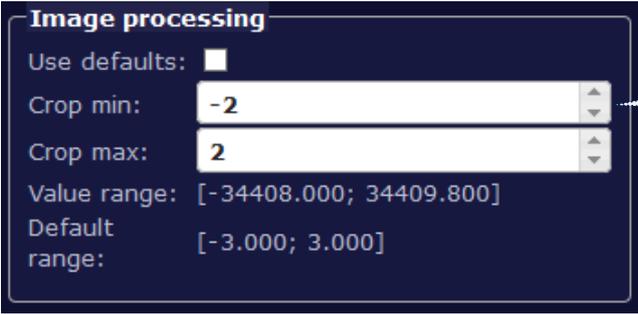
The screenshot shows the 'Image processing' settings panel. A hand icon is pointing to the 'Use defaults' checkbox, which is checked. The 'Crop min' field is empty, and the 'Crop max' field contains the value '3'. The 'Value range' is displayed as [-34408.000; 34409.800] and the 'Default range' is [-3.000; 3.000].

Value range of the actual dataset

Default range set by the administrator of the simulation (will be used in case of default settings)

Or

You can use your own color bar settings.



The screenshot shows the 'Image processing' settings panel. The 'Use defaults' checkbox is unchecked. The 'Crop min' field contains the value '-2' and the 'Crop max' field contains the value '2'. The 'Value range' is displayed as [-34408.000; 34409.800] and the 'Default range' is [-3.000; 3.000].

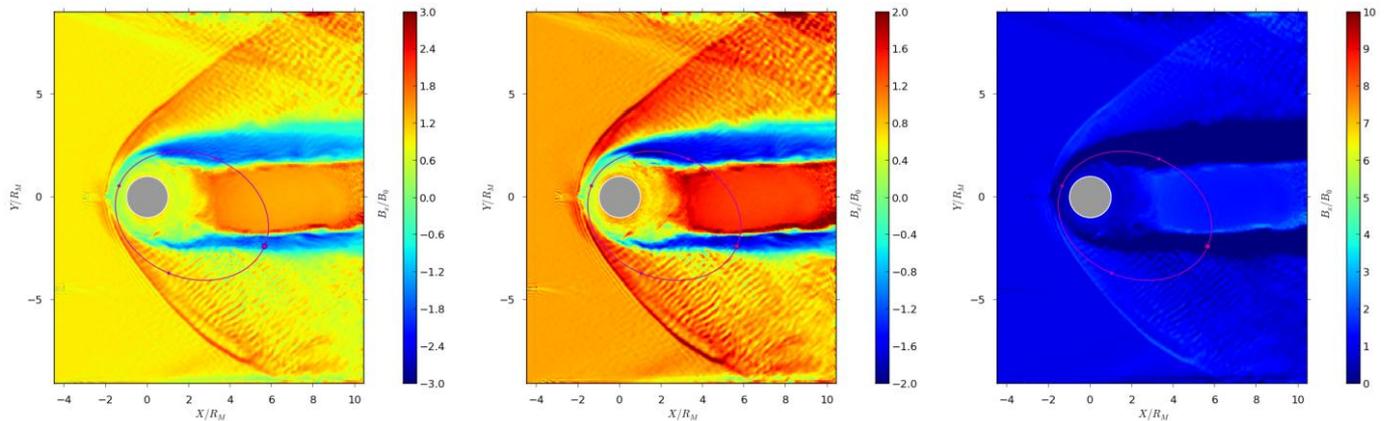
Select color bar range by writing a value or by clicking on increase/decrease buttons.

Note: The effect of this adjustment on the resulting visualization can be seen in the following examples.

Crop min: -3
Crop max: 3

Crop min: -2
Crop max: 2

Crop min: 0
Crop max: 10



4 Plane selection

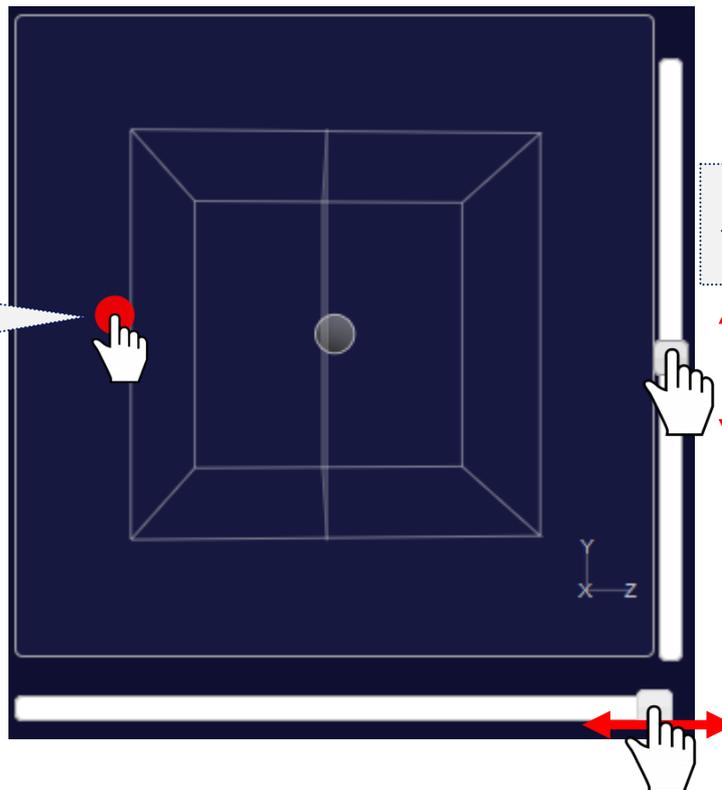
Note: This step is present only in case of three-dimensional simulation. Naturally, there is no plane selection in two-dimensional simulation.

In this step you select parameters of the plane you want to visualize. There are 3 options of an **orientation of the plane**. First choose one of the following options – XY | XZ | YZ. Secondly select **displacement** of the plane in space.



Displacement you can select by writing a value or by clicking on increase/decrease buttons.

For better imagination what you set up you can watch **situational figures**.



Tip: Instead of sliders you can do **left-click** into the area and rotate it directly.

Use sliders to rotate with the figure.

5 Trajectory shape and parameters

Wizard offers the opportunity to enter 4 different types of trajectory shape. Pick one of them and fill in trajectory parameters.

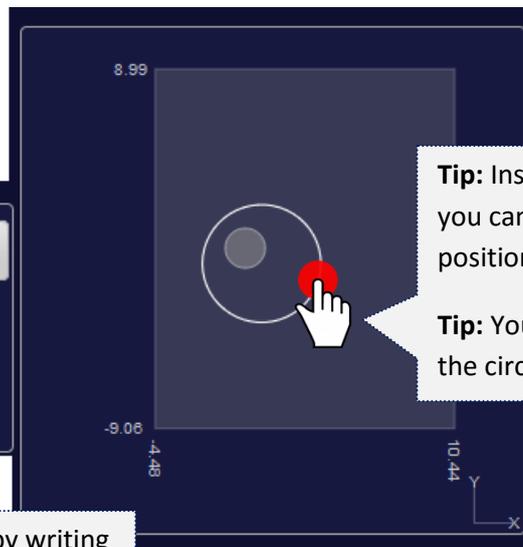
Circle trajectory

Parameters:

Center X – coordinate X of the circle center

Center Y – coordinate Y of the circle center

Radius - size of the circle radius in simulation units



Tip: Instead of writing values you can change the circle position by **left-click & move**.

Tip: You can change radius of the circle by **right-click & move**.

Trajectory parameters can be set by writing a value or by clicking on increase/decrease buttons.

Ellipse trajectory

Parameters:

Center X – coordinate X of the ellipse center

Center Y – coordinate Y of the ellipse center

Major Axis - size of diameter of the ellipse in x-direction (at 0° rotation)

Minor Axis - size of diameter of the ellipse in y-direction (at 0° rotation)

Rotation - rotation angle compared to the x-axis in degrees

Virtual trajectory shape

Circle **Ellipse** Line Custom

Center X: 2.66

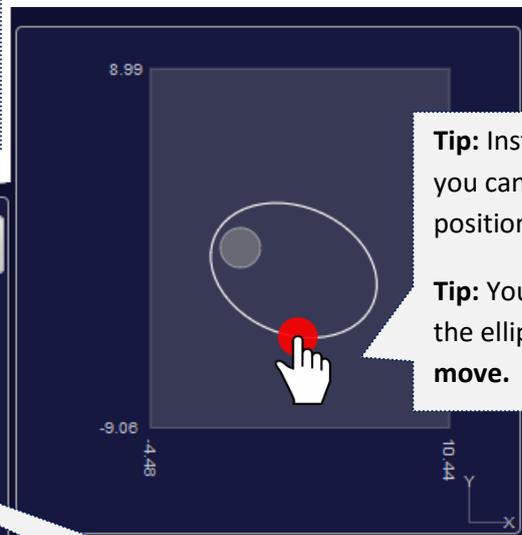
Center Y: -1.14

Use planet's center:

Major Axis: 4.30

Minor Axis: 3.20

Rotation: -23.00



Tip: Instead of writing values you can change the ellipse position by **left-click & move**.

Tip: You can change rotation of the ellipse by **right-click & move**.

Trajectory parameters can be set by writing a value or by clicking on increase/decrease buttons.

Use planet's center – this option means that focus of the ellipse is placed in the center of the planet. Thus, the parameters Center X and Center Y will be calculated automatically.

Line trajectory

Parameters:

Start X – coordinate X of the line starting point

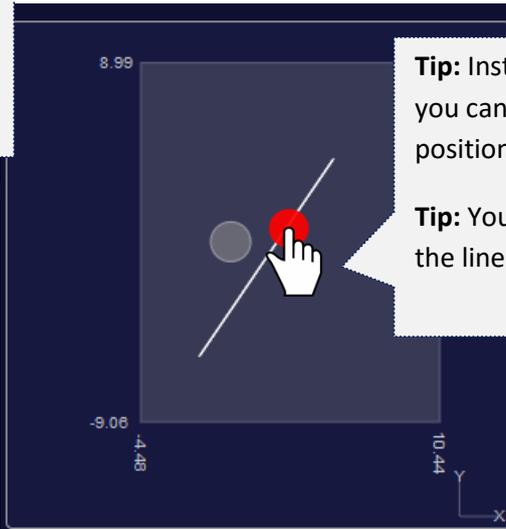
Start Y – coordinate Y of the line starting point

Rotation - rotation angle compared to the x-axis in degrees

Length – distance between starting and ending points

End X – coordinate X of the line ending point

End Y – coordinate Y of the line ending point



Tip: Instead of writing values you can change the line position by **left-click & move**.

Tip: You can change rotation of the line by **right-click & move**.

Trajectory parameters can be set by writing a value or by clicking on increase/decrease buttons.

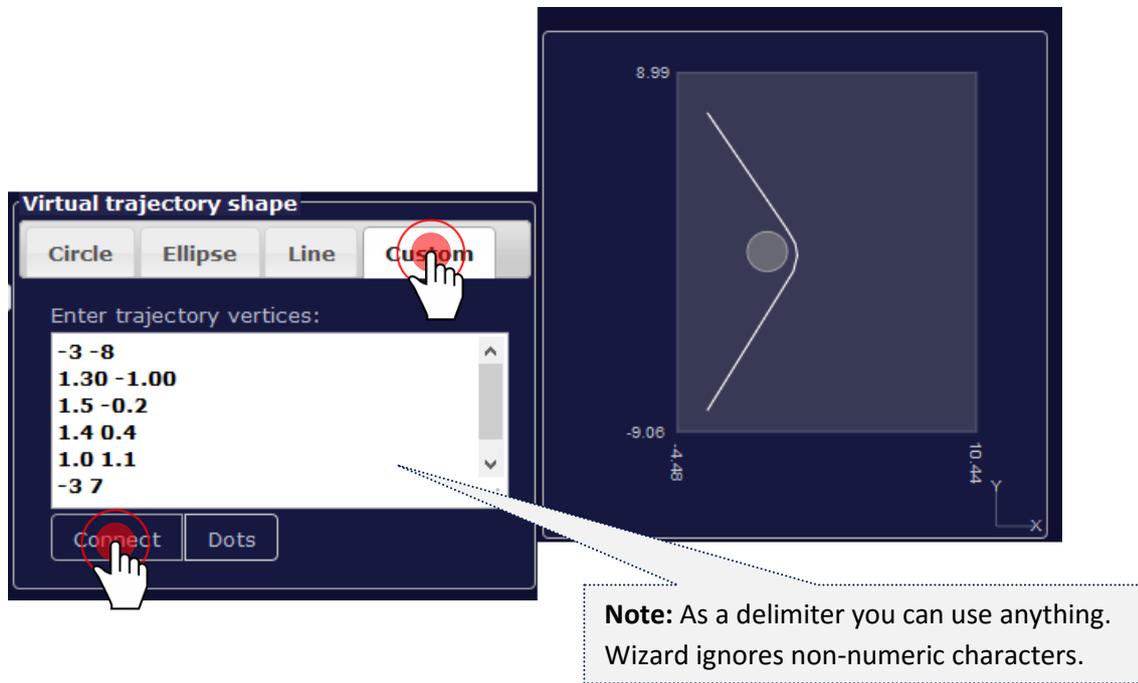
Option 1: Set coordinates of the starting point, rotation and length parameters. The coordinates of the ending point will be calculated automatically.

Or

Option 2: You can set coordinates of the ending point directly.

Custom trajectory

Option 1: Enter the coordinates of points from which you want to create your path and **click the connect button**.

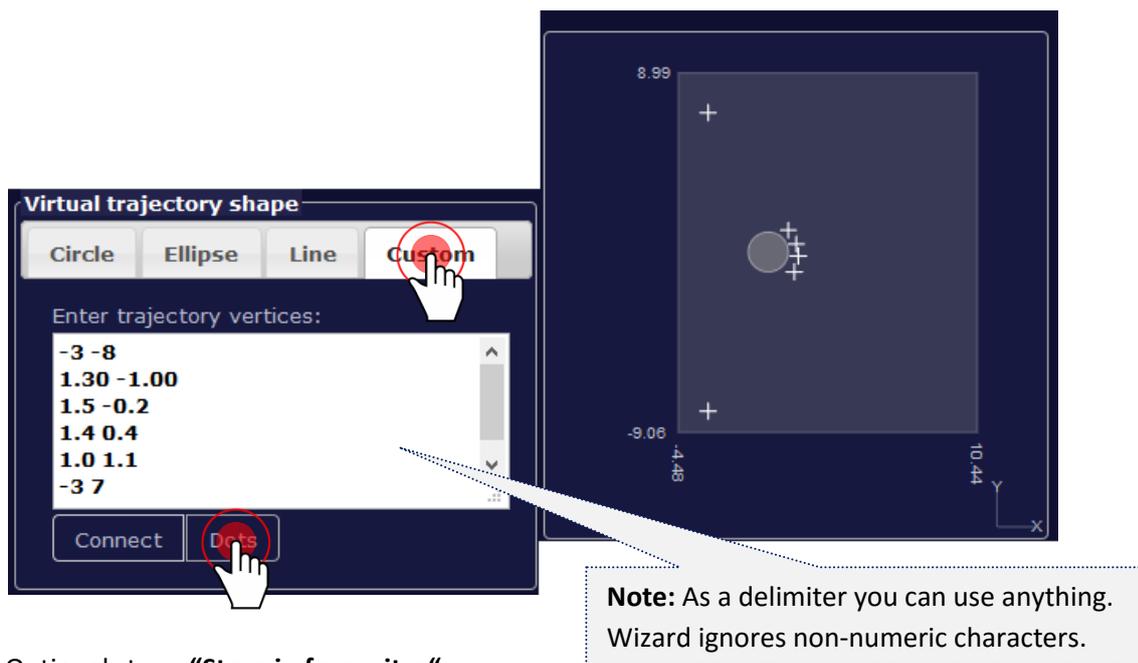


Note: As a delimiter you can use anything. Wizard ignores non-numeric characters.

Or

Option 2: Enter the coordinates of points and **click the dots button**.

Then it will be a special kind of visualization. The points will be taken as a single (stationary) trajectory. This kind of setting is used primarily in the case of comparing values at specified points in time.



Note: As a delimiter you can use anything. Wizard ignores non-numeric characters.

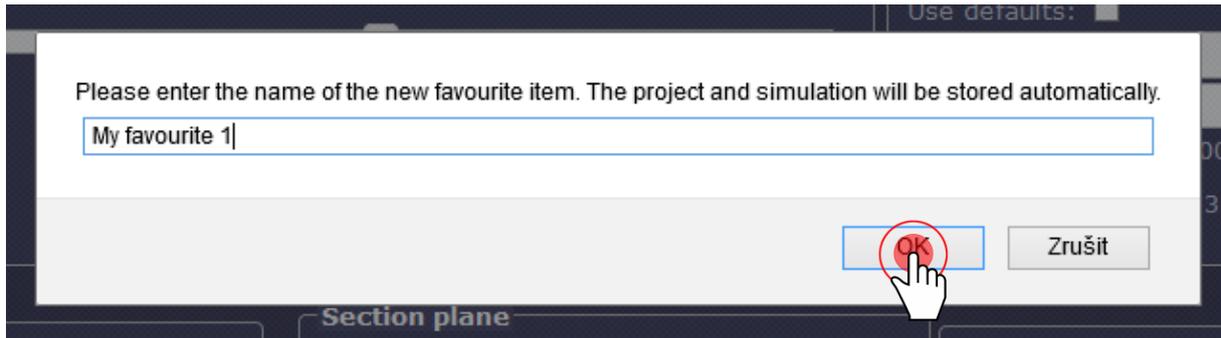
6

Optional step - "Store in favourites"

Wizard offers the option to save the current settings for later reopening. This is done by clicking on "Store in favorites" button.



Then will pop up a window where you fill in the name of your item.



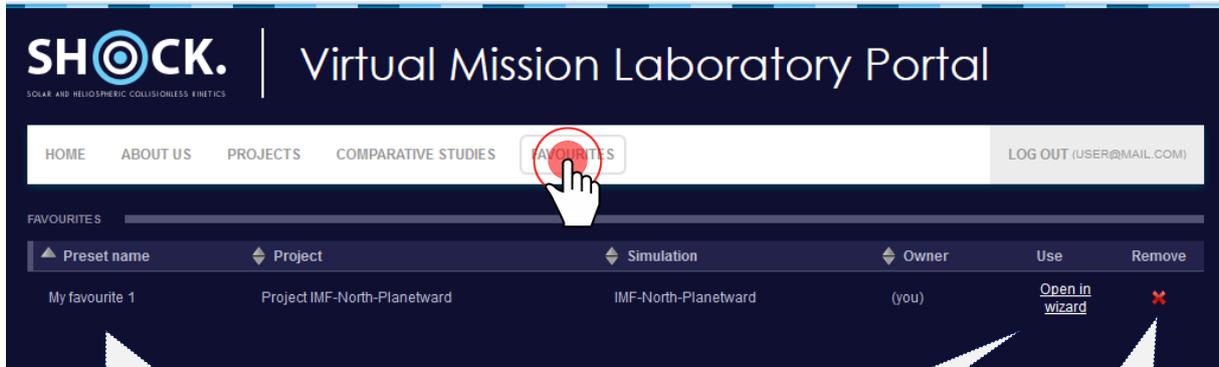
6 Final step – "Queue for processing"

If you are satisfied with your settings click the "Queue for processing" button. Then your job will be moved into the queue for processing. When your job is finished you'll see the visualization output.



FAVOURITES

VML portal offers the option to reopen your stored wizard settings. This functionality is located in the **Favourites** tab.

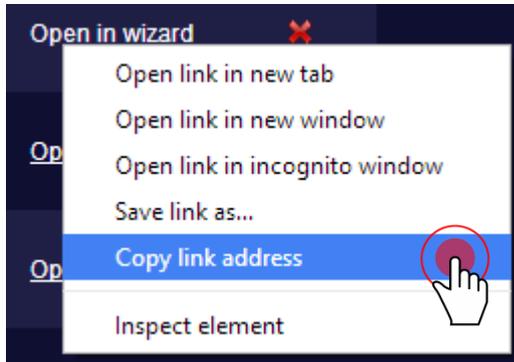


If you want to **change** the item name, click here and write new name.

Click here to **open** this item in wizard or **right-click** to **copy link address**.

Click here to **delete** this item

If you want to **share your favourite preset** follow these steps:



- 1 Right-click on "Open in wizard"
- 2 Choose the option "Copy link address"
- 3 Paste link e.g. into the email message.

Link address example: <http://amalka.asu.cas.cz/esa/index.php/projects/10064/10059/1000171>

VISUALIZATION OUTPUTS

SINGLE TIME

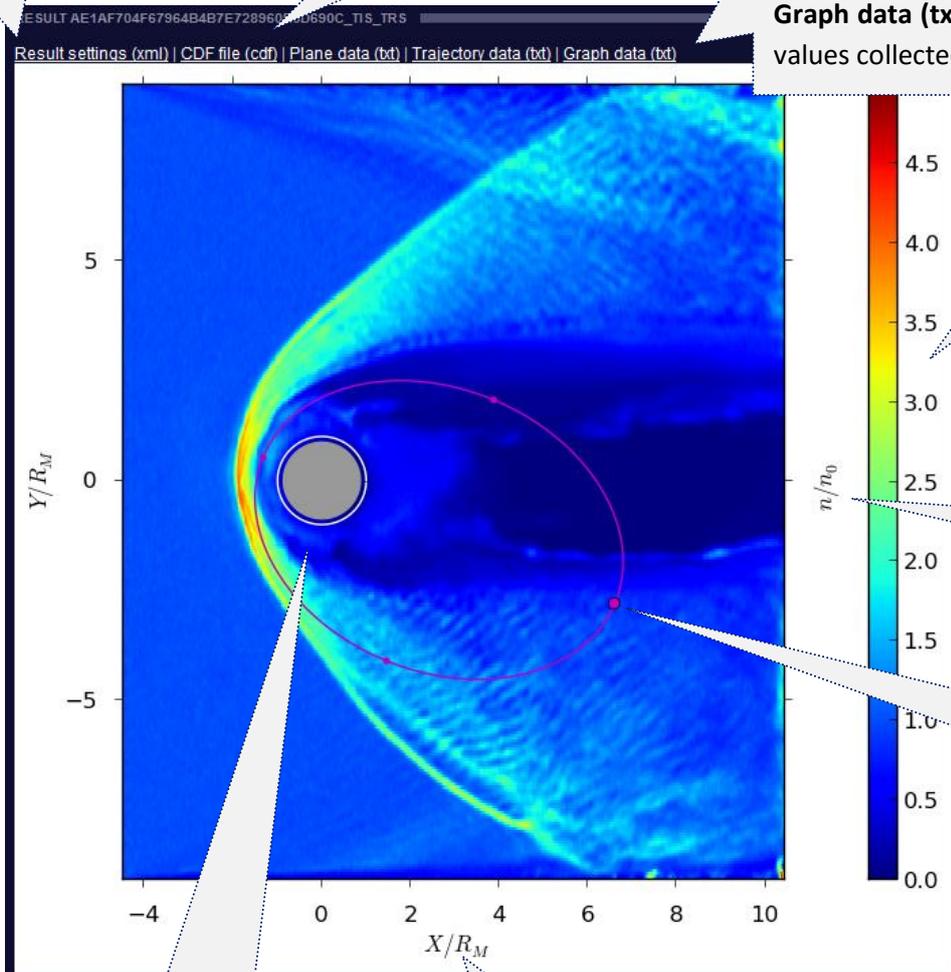
Result settings (xml) is the output where you can find all selected settings in xml form. **It is for information purposes.**

CDF file (cdf) is the output containing all the numerical values of selected plane in CDF file form.

Plane data (txt) is the output containing all the numerical values of selected plane in textual form.

Trajectory data (txt) is textual output that contains values of trajectory coordinates.

Graph data (txt) is textual output that contains values collected along the trajectory.



Color bar for the correct interpretation of the values in the figure

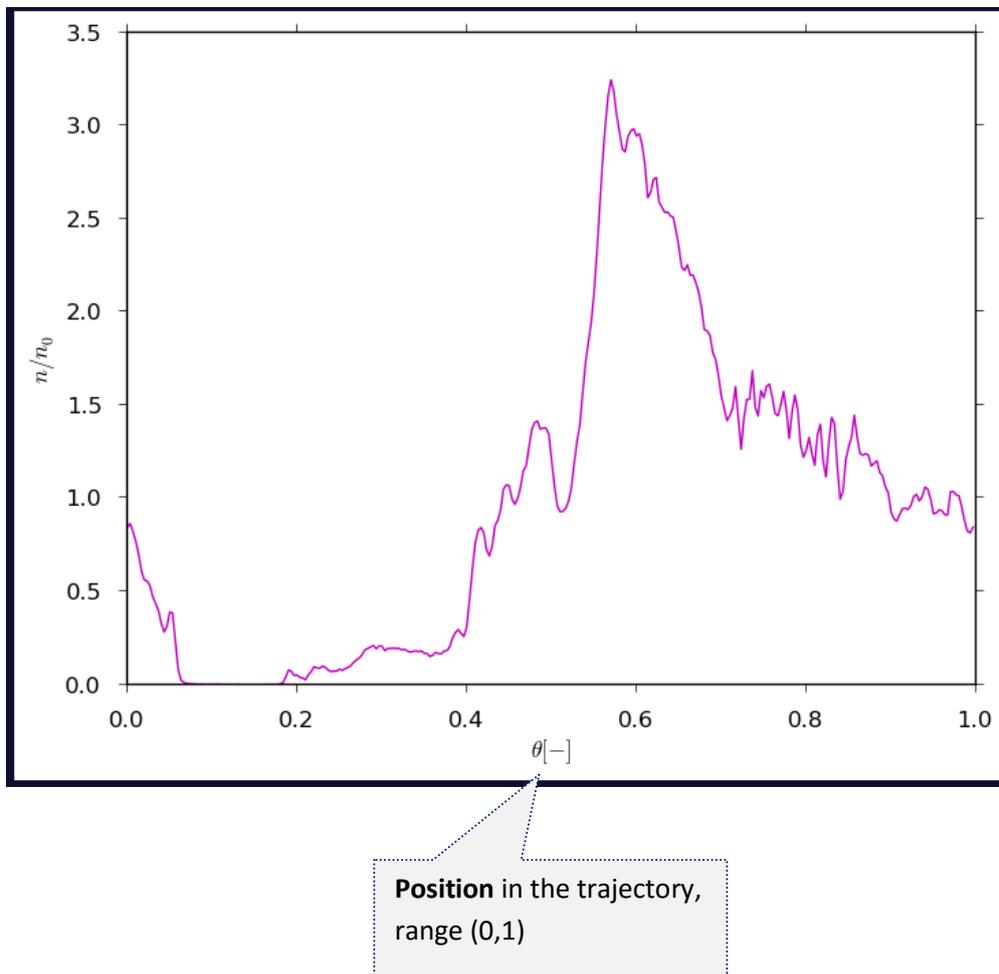
Product name with units

Trajectory has highlighted points at the start, 1/4, 1/2 and 3/4 of its length.

Illustration of the planet
 The **gray color** indicates cross section the planet.
 The **white color** indicates contour the planet.

Axis label with units

The graph shows the value measured along the chosen trajectory. The x-axis indicates the position of the trajectory. For example, a value of 0 indicates the beginning of the trajectory, 0.5 the middle and 1 its end.



TIME RANGE

The time range output is similar to the single time output. Description of output elements you can find in the previous chapter.

Note: Outputs (plane data) in the CDF and textual file format are not listed because of their potential size. If you want to get the output in this format, please select option - single time (step 2).

RESULT 3E2438562B4620AC80CFA18D7C1098B6_TIA_TR5
Result settings (xml) | Time dependence data (txt) | Trajectory data (txt)

Simulation time = 85.0

Y/R_M

X/R_M

n/n_0

5.0
5.0
4.0
3.5
3.0
2.5
2.0
1.5
1.0
0.5
0.0

Time dependence data (txt) is the output containing values collected along the trajectory for in the whole time range.

Actual **simulation time**

Tip: You can use the slider to browse frames.

Switch between animation in **gif** format and **frame** format.
See the note below

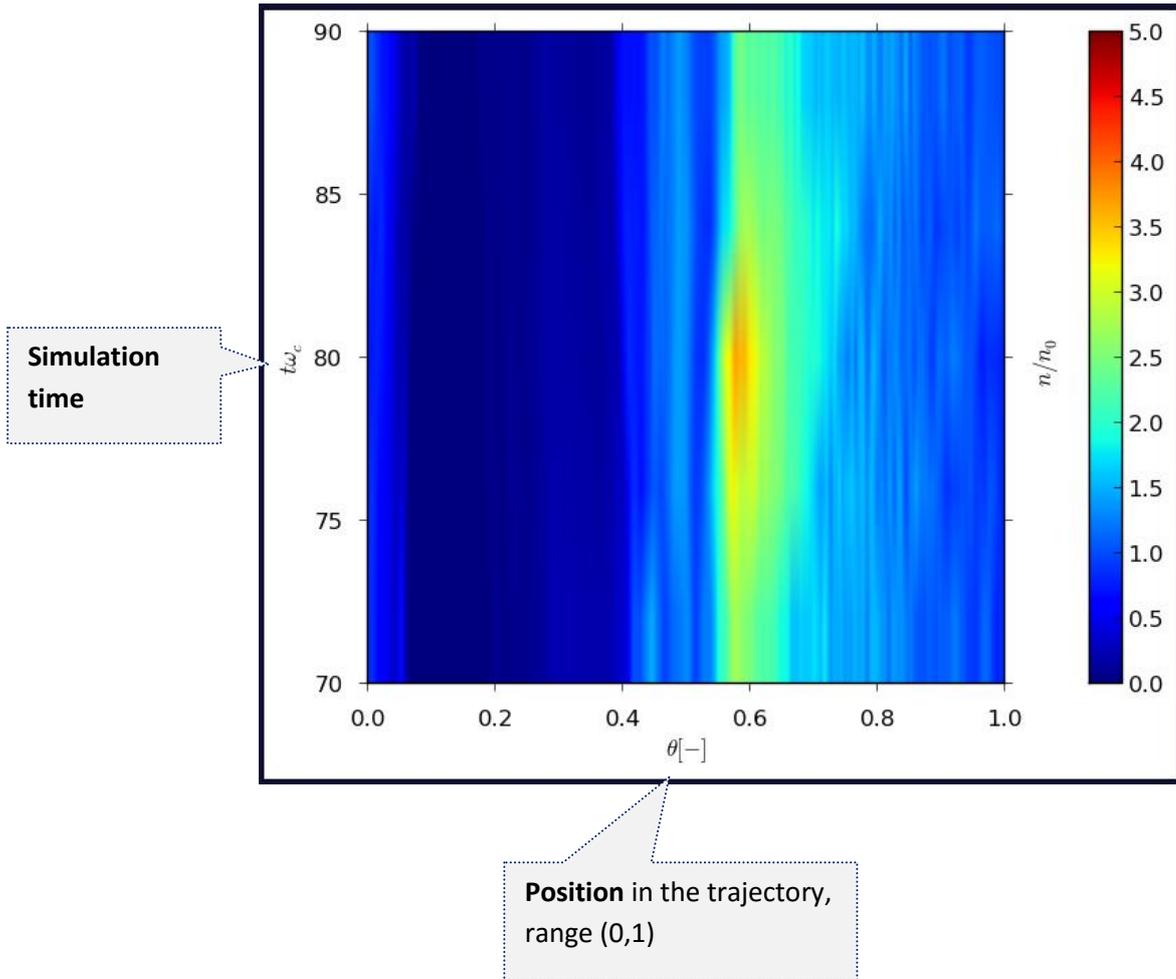
Next frame

Previous frame

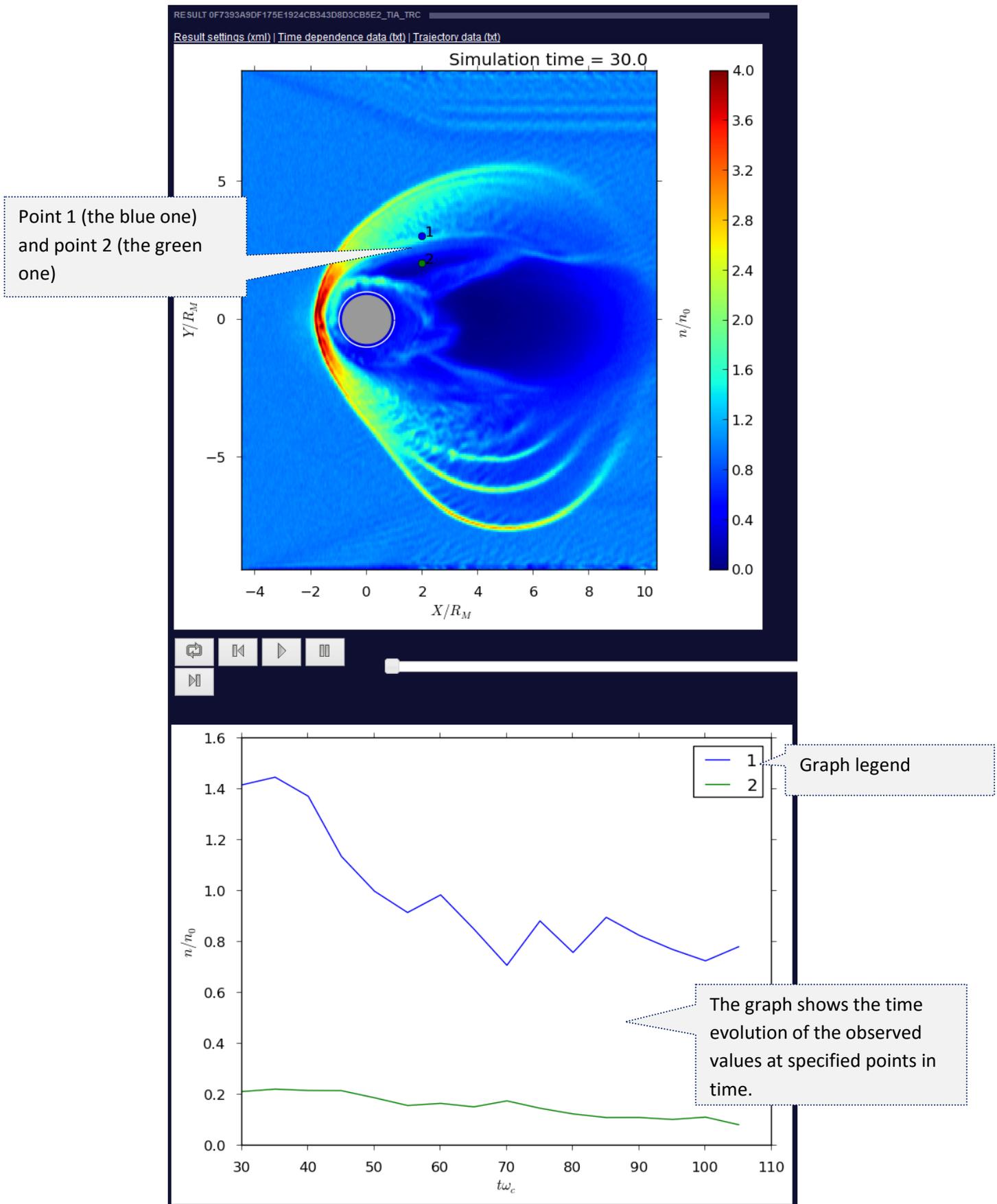
Play the animation

Pause the animation

Note: There are two animation formats (gif and frame format). Gif format is here for the possibility saving animation. In that case use **right-click** and choose option “**save image as**”. Please note that switching between formats may take a few seconds.



Time range visualization output example – case of custom “point” trajectory



DATA DOWNLOAD

There are 2 ways how to **download data results** from visualization request.

1 Click the link above the visualization

Note: For download click the right mouse button and choose option „save as“.



Or

2 Download via **SFTP** account

Every registered user has his own **SFTP** account. At this account you can find data results from your visualization requests **for last 7 days**.

After successful registration you should receive an e-mail with your **SFTP credentials**. We highly recommend you to use some **SFTP client** (for example **FileZilla**).

Here fill in **hostname** that you have received via e-mail.
(probably **amalka.asu.cas.cz**)

Here fill in **username** that you have received via e-mail.

Here fill in your **password** (the same you log in with)

Service **SFTP** runs on port **22**

Here you have a list of your visualization requests.

If you have the **incoming** folder, you can use it as your working directory for data conversion or as a channel for sending data to the administrator.

The screenshot shows the WinSCP SFTP client interface. At the top, there are input fields for Host (amalka.asu.cas.cz), Username (vml-1603), Password (masked with dots), and Port (22). A 'Quickconnect' button is highlighted with a red circle and a hand cursor. Below the input fields, a status window shows the following log:

```

Status: Directory listing successful
Status: Retrieving directory listing...
Command: cd "/1775"
Response: New directory is: "/1775"
Command: ls
Status: Listing directory /1775
Status: Directory listing successful
    
```

The interface is split into two panes. The left pane shows the local file system with drives C:, D:, and E: visible. The right pane shows the remote site directory structure:

Filename	Filesize	Filetype	Last modified
..			
1773		File folder	22/04/2014 15:20:00
1775		File folder	22/04/2014 15:22:00
incoming		File folder	17/04/2014 09:32:00

At the bottom, there is a table for 'Server/Local file' with columns for Direction, Remote file, Size, Priority, and Status. Below that are tabs for 'Queued files', 'Failed transfers', and 'Successful transfers'. The status bar at the bottom right shows 'Queue: empty'.

DATA UPLOAD & CONVERSIONS

This chapter is intended for users who produce data and would like to upload them to VML portal. For proper functionality of the VML portal, you need to follow these steps to load the modelled data.

1 Prepare CDF data files

Create CDF data files according to the **document VML Toolkit** (available for download here: <http://amalka.asu.cas.cz/esa/examples>). It is necessary to satisfy mandatory attributes and variables including their type.

Python template for creating a CDF file can be downloaded here <http://amalka.asu.cas.cz/esa/examples/template-cdf.py>

Note: Because each data producer keeps its data in a different format, it is not possible to provide universal script for conversion into CDF format.

Tip: If you don't want to prepare the environment for data conversion on your computer, you can use already prepared environment on VML portal. Click the tab **Convertor**.



For safety reasons you can work with files and directories in your **/incoming** directory only. You have full access to this directory. It means you can create files and directories here. For uploading data to the **/incoming** use your **SFTP account** (for more information please see the [chapter Data Download](#)).

Important note: Please notice that all data on your SFTP account are **periodically deleted**. It means that all older than 10 days will be deleted.

SHOCK. | Virtual Mission Laboratory Portal
SOLAR AND HELIOSPHERIC COLLISIONLESS FINETICS

HOME PROJECTS ABOUT US FAVOURITES MANAGE USERS IMPORTS CONVERTOR

CONVERT

You can convert your source data by uploading a Python script and then executing it. In the script you can only access your own upload

CONVERSION SCRIPT

```
# -*- coding: utf-8 -*-

def create_cdf_test(product,runname,time,loaddirectory,savedirectory):

    simdesc = 'Hybrid simulation of Io\'s interaction with plasma torus'
    axunit = 'R_I'

    proddesc = 'Magnetic field component B_x'
    produnit = 'B_0'
    produnitdesc = 'Magnitude of ambient magnetic field'

    import numpy as np
    from spacepy import pycdf

    print '\t Reading data...'
    simdata = rdgzip3d(loaddirectory+'/' +product+runname+'t'+str(time)+'.gz')
    #from load import file_3D_bin
    #simdata =file_3D_bin(loaddirectory+'/' +product+spec+runname+'t'+str(time)+'.bin.gz')

    dx = 0.05
    dy = 0.05
    dz = 0.05
    xcenter = 0.4
    ycenter = 0.5
    zcenter = 0.5

    dim = simdata.shape
    ncx = dim[2]
    ncy = dim[1]
    ncz = dim[0]

    print '\t Computing axis values...'
    xval = np.empty(ncx) # values on x-axis in planet-centered coordinate system
    for ix in range(len(xval)):
        xval[ix] = (ix*dx - xcenter*ncx*dx)

    yval = np.empty(ncy) # values on y-axis in planet-centered coordinate system
    for iy in range(len(yval)):
        yval[iy] = (iy*dy - ycenter*ncy*dy)
```

MY FILES

Click on a file to insert its path into the conversion script.

- loaddir/
 - Bzio3dt200.gz (129.2MB)
 - Bxio3dt200.gz (136.4MB)
 - Byio3dt200.gz (135.8MB)
- Pillow-2.4.0-py2.6-linux-i686.egg-tmp/
 - PIL/
 - (empty)
- savendir/
 - Bzio3dt200.cdf (170.89MB)
 - Bxio3dt200.cdf (170.89MB)
 - tex_cache/
 - (empty)

Start code

Press **Start code** when your code is ready to run

OUTPUT

```
SpacePy is released under license. See __licence__ and __citation__ for details, and help() for HTML help.
Reading data...
Computing axis values...
Opening cdf Bzio3dt200.cdf
saving global attributes
saving variables
Writing simulation data...
Closing cdf...
```

Copy&Paste your python script into the **Code** area

List of your files (in **/incoming** directory)

Tip: Click on the filename. It adds path to file into the code automatically.

Here you can see **standard output** (prints, errors, ...)

(content is refreshed periodically)

2 Create project and simulation folders

All projects should be located in separate folders. If you want create new project, create new folder `/[project_name]`

Simulations always are under the project as a subfolder. If you want to add new simulation to the project, create folder under the project `/[project_name]/[simulation_name]`.

3 Copy CDF files into the simulation folder

Copy CDF data files into the simulation folder `/[project_name]/[simulation_name]/...`

4 Prepare configuration .xml files (project.xml, simulation.xml)

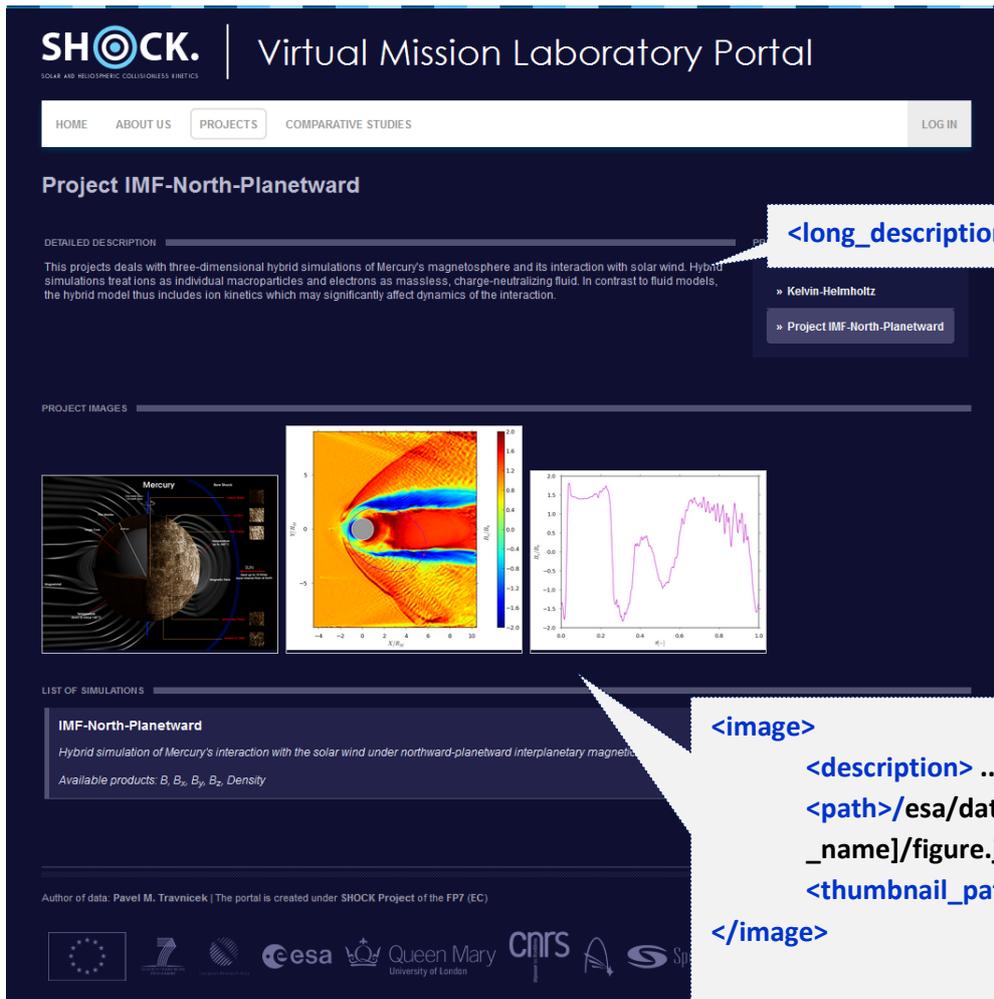
XML files are used to specify various settings and enter the content that appears on the website.

- An example of **project.xml** is available on <http://amalka.asu.cas.cz/esa/examples/XML-templates/project.xml>
- An example of **simulation.xml** is available on <http://amalka.asu.cas.cz/esa/examples/XML-templates/simulation.xml>

Content from XML files appears on the website in this way:

project.xml

The screenshot shows a website interface with a navigation menu (HOME, ABOUT US, PROJECTS, COMPARATIVE STUDIES) and a list of projects. The first project is 'KELVIN-HELMHOLTZ' with a short description 'Hybrid simulations of Kelvin-Helmholtz instability.' and a link 'Go to project details'. Below this is a table with columns 'Simulation' and 'Description'. The first row in the table is 'Hybrid-KH1' with a detailed description: 'Hybrid simulation of Kelvin-Helmholtz instability. The shears are not equal in the plasma around magnetosphere at (2013), doi: 10.1063/1.4826214 for...'. Annotations with dashed boxes point to the XML tags used in the XML files: `<name> ... </name>` points to the project name, and `<short_description>...</short_description>` points to the short description.



`<long_description>... </long_description>`

`<image>`
`<description> ... </description>`
`<path>/esa/data/DB_simdata/[project_name]/figure.jpg</path>`
`<thumbnail_path></thumbnail_path>`
`</image>`

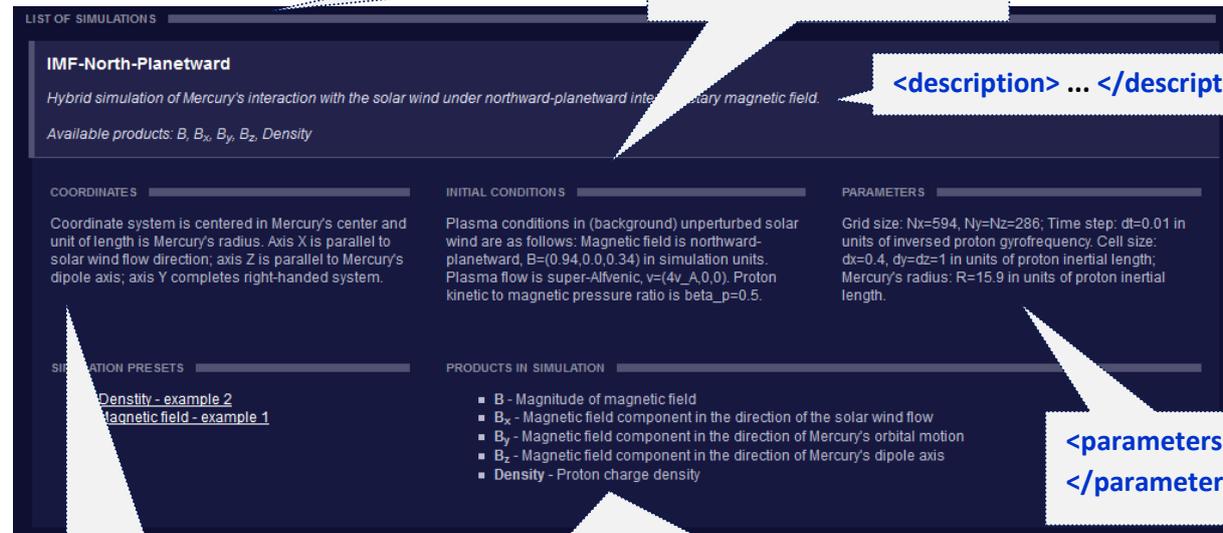
Tip: Use **html** for text adjustment in the element `<long_description>`

There are enabled html tags in the element `<long_description>`. User can add tags to make the long description more clear. Because it is necessary to add these tags to the xml configuration file the characters "`<`" and "`>`" must be replaced with "`<`" and "`>`,"

Examples:

- **End of line (instead of "enter")** - `
`
`"
"`
- Make some text **bold** or **italics** - `...text...` or `<i>...text...</i>`
`"...text..."` or `"<i>...text...</i>"`
- **Add link to the text** - `Google`
`"Google"`

simulation.xml



`<name> ... </name>`

`<initials_condition> ... </initials_condition>`

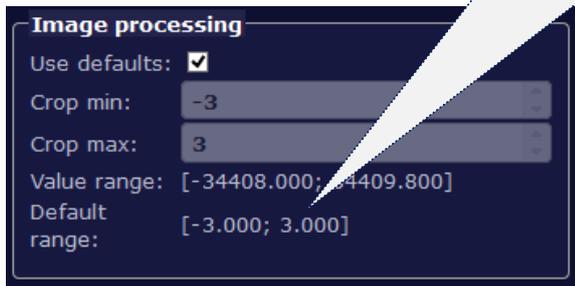
`<description> ... </description>`

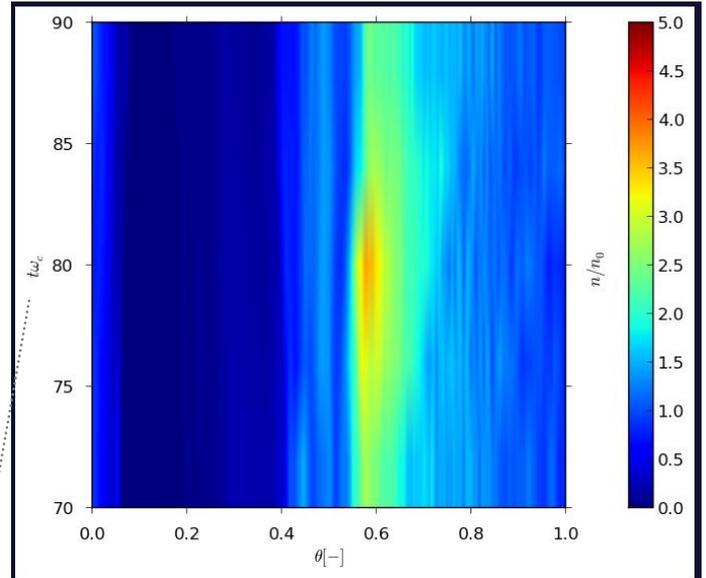
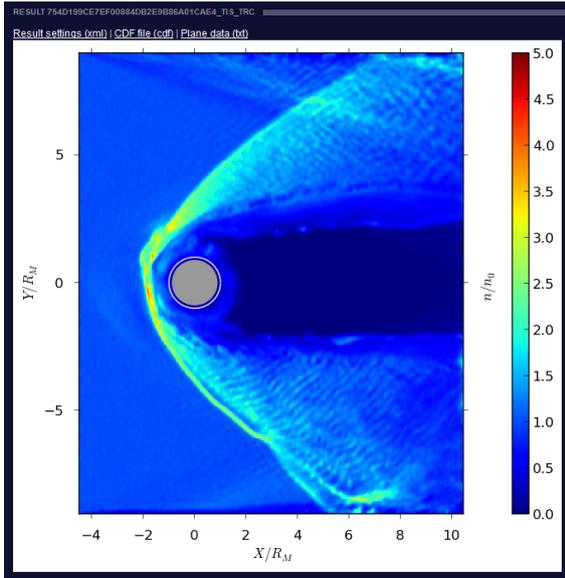
`<parameters> ... </parameters>`

`<coordinate_system> ... </coordinate_system>`

`<product>`
`<name>Bx</name>`
`<display_name>B_x</display_name>`
`<caption>B_{x}/B_{0}</caption>`
`<description> Magnetic field component... </description>`
`<parent_product>B</parent_product>`

`<default_color_range>`
`<min>-3.0</min>`
`<max>3.0</max>`
`</default_color_range>`
`</product>`





<captions>

<x_caption> X/R_M </x_caption>

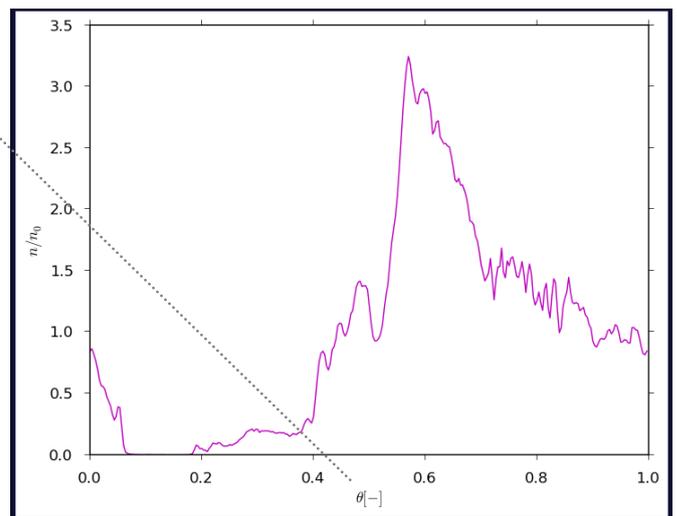
<y_caption> Y/R_M </y_caption>

<z_caption> Z/R_M </z_caption>

<time_caption> $t\omega_c$ </time_caption>

<trajectory_caption> $\theta[-]$ </trajectory_caption>

</captions>



5 Copy .xml files (project.xml, simulation.xml) to the project/simulation folders

In every project folder must be **project.xml** and in every simulation folder must be **simulation.xml**. In another case project/simulation will be ignored by importer tool.

Note: If you prepare project and simulation folders with CDF and xml files on your own computer, it is necessary to upload them via **SFTP** to your **/incoming** folder. How to use SFTP please see the [chapter Data Download](#).

You can upload other files too (figures and other files). Importer tool works with CDF files, project.xml and simulation.xml files only.

6 Contact the administrator

Contact the administrator that your data is prepared and ready for import to the VML portal. For this purpose you can use link located on every page of VML.



Important: In the message, please include your email with which you registered on the portal or your SFTP username.