

# **REGIONS Gamma-Gamma** User Guide

Copyright © Sergey L. Gladkiy  
lrndlrnd@mail.ru

## Contents

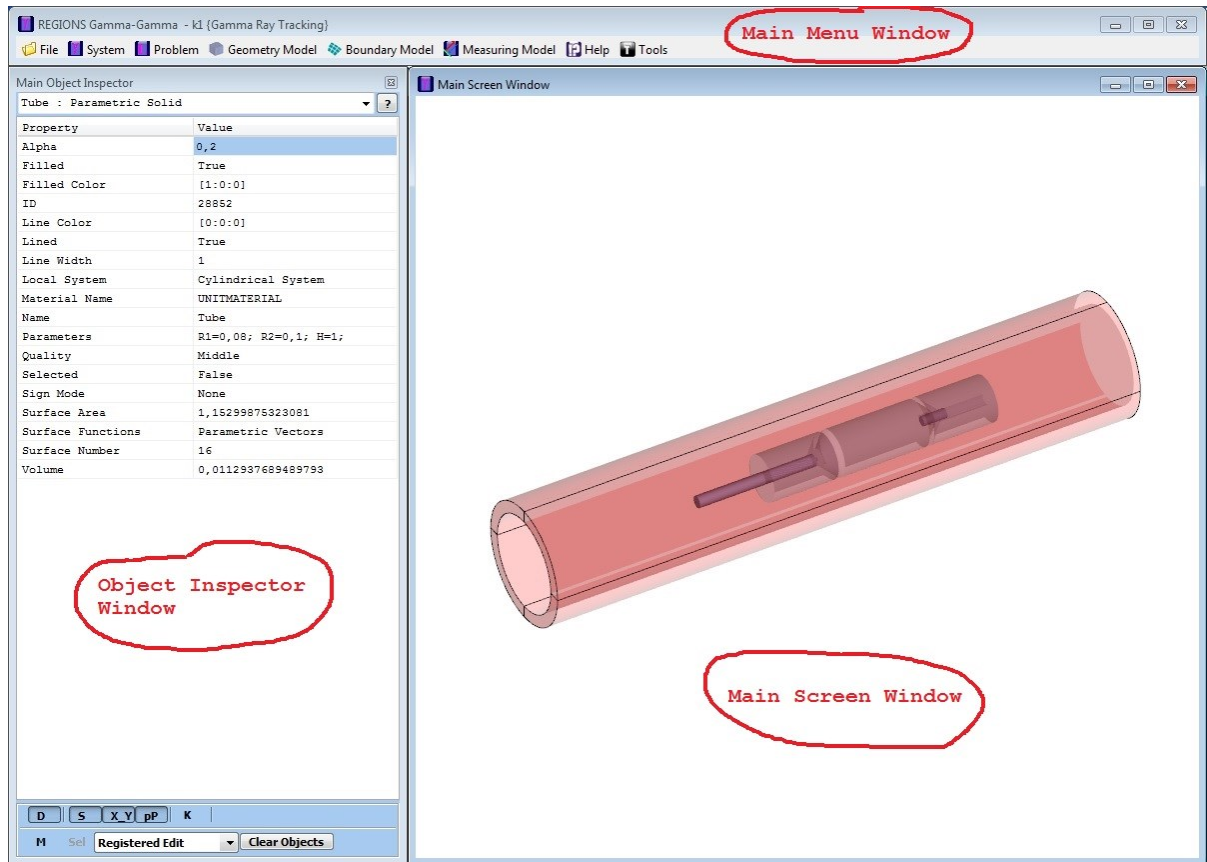
Introduction .....	3
User Interface .....	4
Problem Solution Procedure .....	5
Creating new problem .....	5
Problem structure .....	5
Creating Geometry model .....	6
Creating Points .....	6
Creating Lines .....	6
Creating Surfaces .....	8
Creating Solids .....	10
Creating Boundary model .....	12
Creating Boundary objects .....	12
Creating Measuring model .....	13
Setting Source and Detector properties .....	14
Solving Problem .....	15
Results and postprocessing .....	16
Solving Test Problems .....	19

## **Introduction**

REGIONS Gamma-Gamma system is intended to model space-energy distribution of gamma-quanta in physical continuums. The program is based on Monte-Carlo simulation algorithm.

## User Interface

The REGIONS Gamma-Gamma program user interface consists of four main windows (picture 1).



Picture 1. REGIONS Gamma-Gamma program user interface.

The **Main Menu** window contains menu strip to execute all program operations.

The **Main Screen** window displays current graphical information and allows user to manipulate objects (select, unselect, get object information and so on) by mouse. Screen view management includes the following operations:

Zoom In, Zoom Out: Mouse Wheel Down, Up

Pan: Mouse Left Down -> Mouse Move

Rotate (about global coordinate system center): Mouse Right Down -> Mouse Move

Zoom by window: Ctrl+Mouse Left Down -> Mouse Move -> Mouse Left Up

(to escape Zoom by window while Mouse Moving just press Escape).

If the **Main Screen** window is not visible - use main menu command *System -> Graphics -> Main Screen Window* to show the screen. **Main Screen** window has Popup menu that can be opened by pressing Mouse Right button on the window.

The **Main Object Inspector** window allows user to change properties of objects. To set new property value just input (or select from list) new value and press Enter. Some complex properties (such as Colors, Vectors and so on) have their own Property Editors those can be executed by pressing button on the right of the property. Some properties are Read Only (such as ID of the object, Area of the surface and so on) and cannot be edited (but they are displayed by Object Inspector). The **Main Object Inspector** window is commonly shown when user executes *Edit* operations from the main menu.

The **Log** window (is not shown on the picture) displays text information about user action results, solution progress and results, errors and hints. To make the Log window visible use main menu command *System -> Log -> Show Information Log Window*.

## Problem Solution Procedure

Problem Solution procedure includes several stages which should be (commonly) implemented in the order as described below.

### Creating new problem

First of all you need to create new problem. Use main menu command *Problem -> New Problem*. In the displayed form input the Name of the problem, select working directory (press button to use Directory Browser), select Problem Analysis from the list and press Apply button. New problem will be created and Geometry Model will be loaded. If problem file with the same name exists in the same directory - problem data information will be loaded from the file.

### Problem structure

A REGIONS Gamma-Gamma Problem is a set of Models. A Model is again a set of objects of some Base type. For an example, Geometry model is a set of geometry objects - points, lines, surfaces and solids. Objects in model can be connected with other objects in the same model and in the other models. REGIONS Gamma-Gamma Problem consists of the following models:

- Geometry model.
- Material model.
- Boundary model.
- Measuring model.
- Result model.

With each model user can make the following operations (menu strip **Problem**):

**Load model** - model data is loaded into program memory and is ready to manipulate with the model objects (if the model file exists on the disk in the problem directory - model data is loaded from the file).

**Load model from** - model data is loaded from the selected by user file and is ready to manipulate with the model objects.

**Unload model** - model data is saved into model file in the problem directory and destroyed in the problem memory (so it is not accessible to manipulate with).

**Save model as** - model data is saved into selected by user (so it is still accessible to manipulate with).

**Save model** - model data is saved into the model file in the problem directory (so it is still accessible to manipulate with).

All problem data can be saved into files with the main menu command *File -> Save Problem*. With the main menu command *File -> Save Problem as...* all problem data saved into files with selected name in selected directory. If some model data is not loaded into the program memory and the model file exists - the file is just copied with the new name. Saving problem with the new name causes the name of the problem and the working directory changing.

## Creating Geometry model

Geometry model describes the geometry of the Boundary Value problem. Geometry model consists of a set of geometry objects – points, lines, surfaces and solids.

The REGIONS Gamma-Gamma program uses Boundary Representation (BREP) technique, so to create a solid body a user must create all surfaces the body consists of. A surface can be created by existing lines or as new surface with its own parameters (for an example – parametric surface). A line can be created by using existing points or as new line with its own parameters (for an example – parametric curve). A point can be created by its coordinates or can be created on existing object (for an example – on a line).

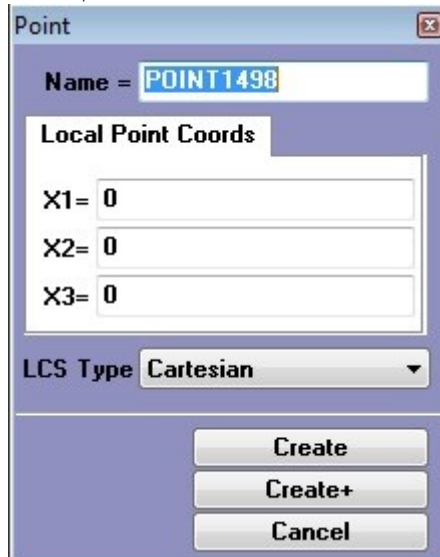
All created geometry objects can be changed in any moment of geometry modeling. Use main menu command *Geometry -> Edit ->...* to change geometry object properties. Because (in common) geometry objects are connected with each other (for an example a straight line created by end points), changing one geometry object leads to changing the other, connected with it.

Each geometry object has its own Local Coordinate System (LCS). The parameters of Local Coordinate systems (translation, rotation and so on) are described relatively to the Global Coordinate system. Local Coordinate systems can be changed via translation, rotation and other operations. So the geometry objects can be changed via LCS manipulations too.

All geometry objects are created in the current Working Space. Working Space is simply an object representing a Local Coordinate System. To show Working Space use main menu command *Problem -> Options* (page Graphics). To manipulate Working Space use main menu command *Problem -> Edit Working Space*. Using Working Space for geometry object creation means just using the initial Local Coordinate System parameters for the object.

## Creating Points

Use main menu command *Geometry Model -> Create -> Point* to create a point by coordinates. Point window will be shown (picture 2).

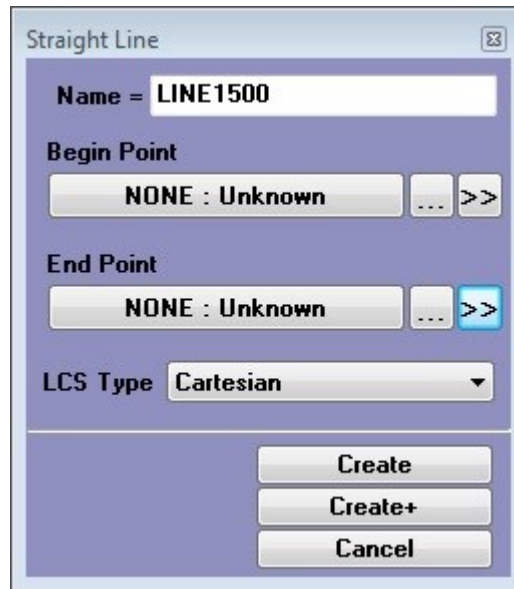


Picture 2. Point window.

Input coordinates of the point in Local Coordinate System (the type of the system can be selected from the list). While inputting coordinates – the preview of the point is shown on the Main Screen. Press Create or Create+ button to create the point.

## Creating Lines

Use main menu command *Geometry Model -> Create -> Straight Line* to create a line by two end points. Straight Line window will be shown (picture 3).



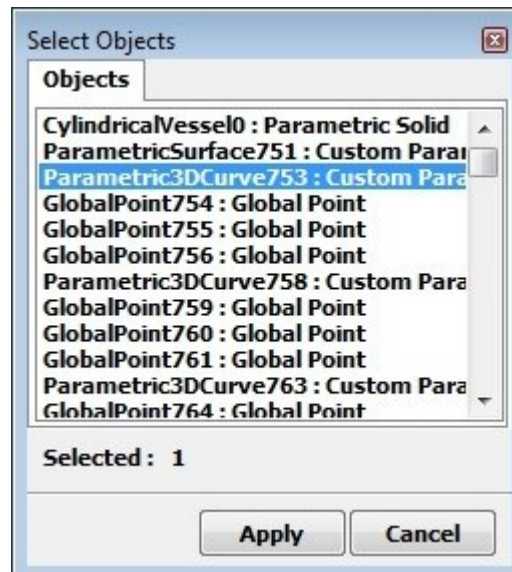
Picture 3. Straight Line window.

Two points must be chosen to create Straight Line (so at least two points must be created before). Use Object Choice interface (picture 4) to choose points.



Picture 4. Object Choice interface.

Object Choice interface allows two ways to get object. Use >> button to choose object with Object Browser (picture 5).



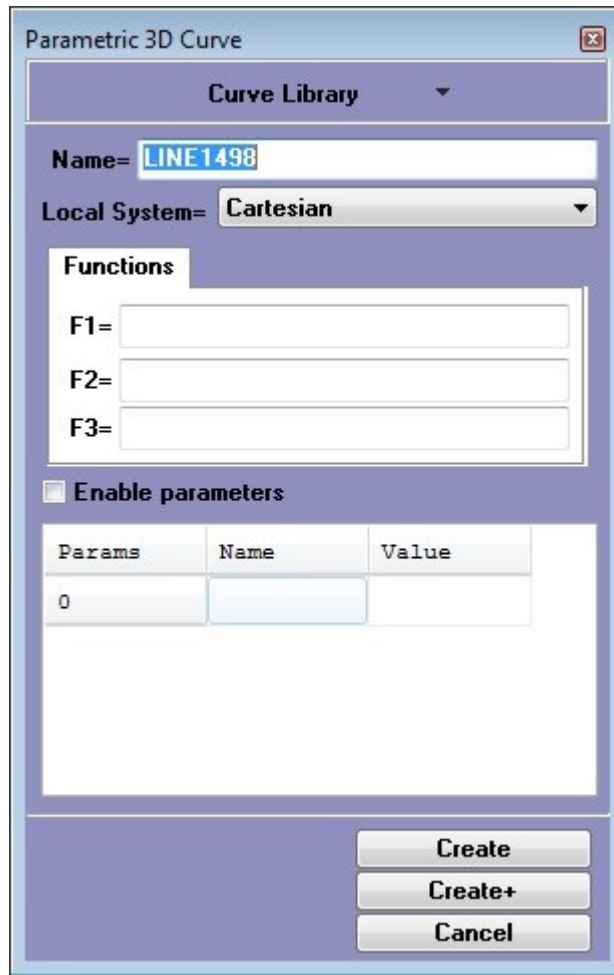
Picture 5. Object Browser window.

When an object selected in the Object Browser it is highlighted on the Main Screen. Press Apply button to choose the selected object.

Use ... button to activate Object Picking by mouse. Object Picking will be activated and object can be chosen by mouse on the Main Screen. When Object Picking is activated the current object is highlighted on the Main screen while mouse moving.

When both end points are chosen - the preview of the line is shown on the Main Screen. Press Create or Create+ button to create the line.

Use main menu command *Geometry Model* -> *Create* -> *Parametric Curve* to create a freeform line by equations. Parametric Curve window will be shown (picture 6).



Picture 6. Parametric Curve window.

Parametric Curve is a freeform line that determined by three equations

$$\begin{cases} x_1 = F_1(l) \\ x_2 = F_2(l) \\ x_3 = F_3(l) \end{cases} \quad (1)$$

were  $x_1, x_2, x_3$  - coordinates of the line point (in Local Coordinate System),  $F_1(l), F_2(l), F_3(l)$  - functions of Line Parameter  $l$ . The Line Parameter  $l$  changes from 0 to 1.0.

So, three functions of the variable  $l$  must be given to create a Parametric Curve. User can input the function equations himself or can use Curve Library menu to select existing line equations. Curve Library menu contains also operations to add line equations into library, save equations in text files and read equations from text files. There are two libraries - Standard Library contains simple line equations (such as arc, elliptic arc and so on), Additional library contains more complicated line equations.

When all three equations set - the line preview is shown on the main screen. If there are errors in equations - hints are shown in the Log window. Press Create or Create+ button to create the line.

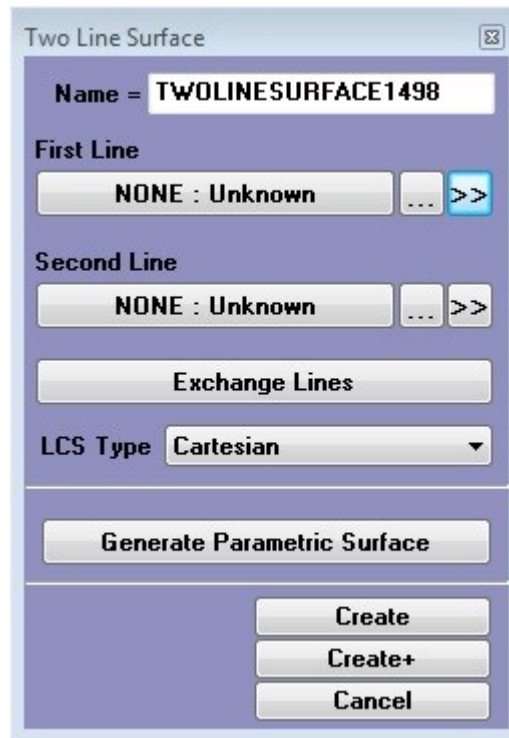
### Creating Surfaces

Use main menu command *Geometry Model -> Create -> Two Line Surface* to create a surface that starts with the first line and ends with the second one (so called ruled surface). Two Line Surface window will be shown (picture 7).

Two end lines must be chosen to create a ruled surface. Use >> buttons to choose lines with Object Browser or ... buttons to activate Object Picking by mouse on the Main Screen.

When two lines are chosen - the preview of the surface is shown on the Main Screen. Press Create or Create+ button to create the surface.

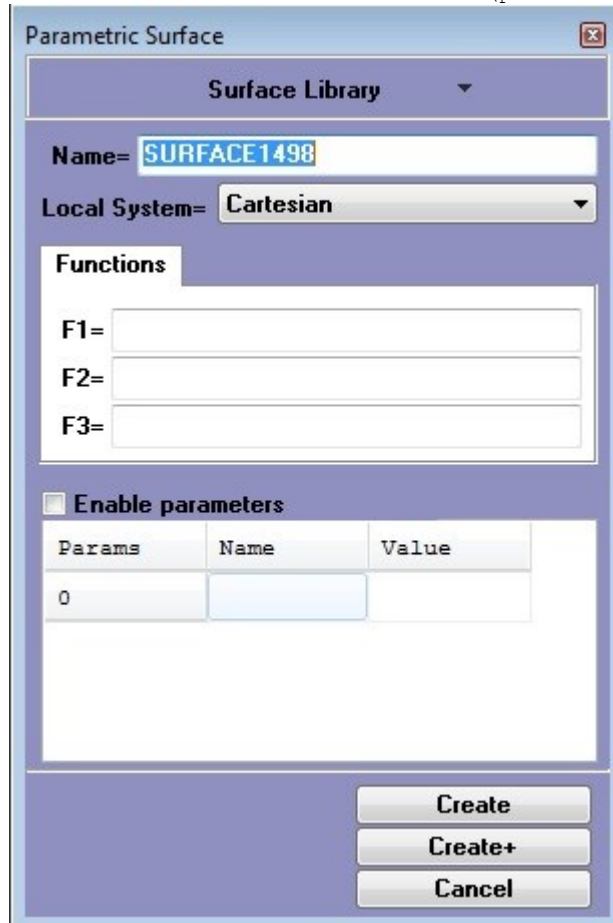




Picture 7. Two Line Surface window.

NOTE about surfaces: any surface has outer face and inner face (the former displayed on the screen with surface color and the last is darkened). The outer direction is where the normal to the surface points to, and the inner is inverse direction. When a preview of a surface is shown - the normal direction is displayed as an arrow. The surface direction can be changed to the inverse one by changing the property Inverse Direction value to the True (use command *Geometry Model -> Edit -> Edit Surfaces* to edit surface properties).

Use main menu command *Geometry Model -> Create -> Parametric Surface* to create a freeform surface by equations. Parametric Surface window will be shown (picture 8).



Picture 8. Parametric Surface window.

Parametric Surface is a freeform surface that determined by three equations

$$\begin{cases} x_1 = F_1(u, v) \\ x_2 = F_2(u, v) \\ x_3 = F_3(u, v) \end{cases} \quad (2)$$

were  $x_1, x_2, x_3$  - coordinates of the surface point (in Local Coordinate System),  $F_1(u, v), F_2(u, v), F_3(u, v)$  - functions of Surface Parameters  $(u, v)$ . The Surface Parameters  $u$  and  $v$  changes from 0 to 1.0.

In the REGIONS X program surface parameters denoted as  $L[1]$  and  $L[2]$ . So, three equations of variables  $L[1]$  and  $L[2]$  must be given to create a parametric surface. A user can write his own surface equations or get a predefined ones form Surface Library.

When three valid surface functions are given - the preview of the surface is shown on the Main Screen. If there are errors in formulae - hints are displayed in the Log window. Use Create or Create+ button to create the surface.

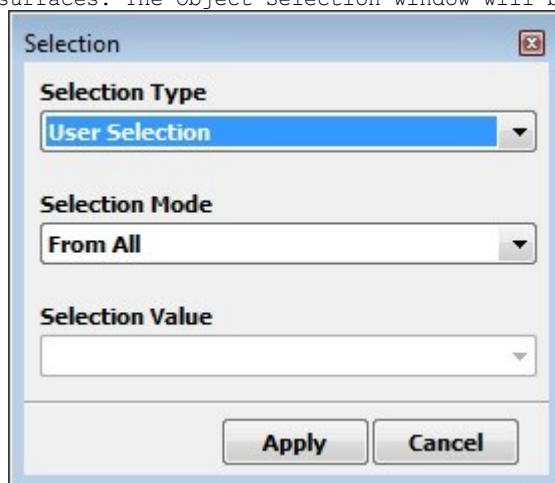
### Creating Solids

Use main menu command *Geometry Model -> Create -> Solid Body* to create a solid by existing surfaces. Solid Body window will be shown (picture 9).



Picture 9. Solid Body window.

Surfaces representing the boundary of the body must be selected to create a solid. Press Select button to select existing surfaces. The Object Selection window will be shown (picture 10).



Picture 10. Object Selection window.

With the Object Selection interface different selection methods can be used. Selection Type determines how objects will be selected. For an example, User Selection type will activate Object Browsing interface. By Name type allows object selection by their names and so on. Selection Mode determines how the selection is applied to the objects. For an example, From All mode means that selection is applied to objects directly. Add mode means that chosen objects are added to the already selected and so on. Surface selection by mouse can be activated by main menu command *Geometry Model -> Select -> Select Surfaces by Pick*.

Selected objects are always displayed on the screen with Red color. When all required surfaces are selected - press Create or Create+ button to create the solid.

NOTE about solids: REGIONS Gamma-Gamma program modeling functions do not check the validness (connection of the surface edges, surface normal directions and so on) of the solids created. So, it is all user responsibility to check the solid validness.

Use main menu command *Geometry Model -> Create -> Parametric Solid* to create a totally parameterized solid. Parametric Solid window will be shown (picture 11).

**Parametric Solid**

**Solid Library** ▼

**Name =** SOLID1498

**Material Name :** UNITMATERIAL ▼

**Local System Type =** Cartesian ▼

**Parameters :**

Params	Name	Value
0		

**Surface functions :**

Functions	F1	F2	F3
0			

**Create**  
**Create+**  
**Cancel**

Picture 11. Parametric Solid window.

Parametric solid is a solid the boundary of which totally consists of parametric surfaces. All boundary parametric surfaces have the same parameters but different surface functions. So, to create a parametric solid three functions for each boundary surface must be given. A user can input his own surface equations or get predefined ones from the Solid Library.

When valid surface functions are given - the preview of the solid is displayed on the main screen. If errors exist - the hints are shown in the Log window. Press Create or Create+ button to create the solid.

## Creating Boundary model

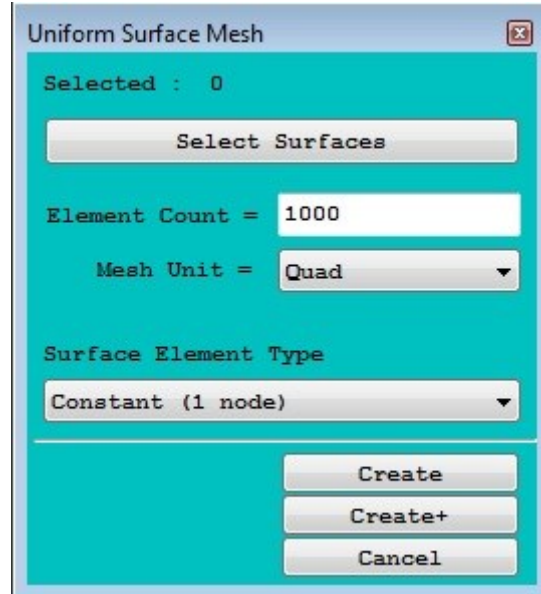
Boundary Model is a set of boundary objects those represent the boundary of the problem. Actually, boundary object is a collection of boundary elements. Boundary elements are mainly used to implement Particle Tracking for Monte-Carlo simulation algorithm.

Boundary model creation includes two procedures – creation of boundary objects (mesh the boundary) and setting boundary conditions. Before begin operation with Boundary Model it must be loaded into program memory. Use main menu command *Problem -> Load Boundary Model* to get it loaded.

## Creating Boundary objects

The simplest and fastest way to create boundary objects (mesh) is to create uniform mesh and then change the number of boundary elements for individual boundary objects.

Use main menu command *Boundary Model -> Create -> Uniform Surface Mesh* to create uniform mesh for a set of selected surfaces. Uniform Surface Mesh window will be shown (picture 12).



Picture 12. Uniform Surface Mesh window.

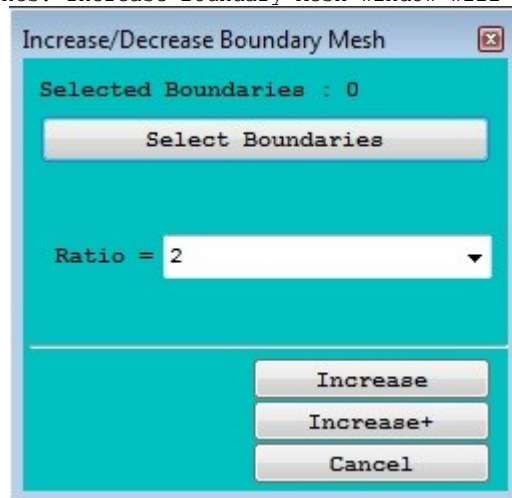
Press the Select Surfaces button to select surfaces with the Object selection interface. When surfaces selected, input the number of boundary elements, select the form of elements (quads or triangles) and the type of elements. Press Create or Create+ button to create the mesh.

If mesh operation completes successfully – created boundary objects will be displayed on the Main Screen (to switch between model views – use Main Screen popup menu command *Plot -> ...*).

NOTE about uniform mesh: it is not always possible to produce really uniform mesh for any geometry object because of its natural “nonuniformness”, so the uniform mesh tool does not always produce the desired result.

Use main menu command *Boundary Model -> Create -> Uniform Surface Mesh for Solid* to create uniform mesh for all surfaces of selected solids.

Use main menu command *Boundary Model -> Edit -> Increase/Decrease Boundary mesh* to change the number of boundary elements. Increase Boundary Mesh window will be shown (picture 13).



Picture 13. Increase Boundary Mesh window.

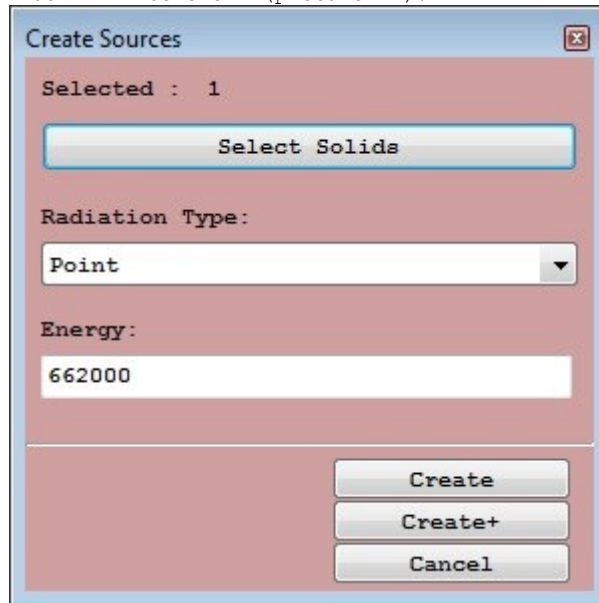
Press **Select Boundaries** button to select boundaries with the **Object Selection** interface. Selection by mouse picking can be activated by main menu command *Boundary Model -> Select -> Select by Pick*. After boundary selection input (or choose) the ratio of mesh increasing and press **Increase** or **Increase+** button to remesh objects.

NOTE about boundary objects: geometry model objects can be edited in any moment of Problem Solution procedure, even when the boundary objects created for some geometry objects. But boundary model is not updated automatically when the target geometry is changed. To update boundary objects use main menu command *Boundary Model -> Remove/Refresh -> ...*.

## Creating Measuring model

Measuring Model is a set of Sources and Detectors. Sources are objects radiating the gamma quanta. Detectors are the objects fixing gamma quanta and registering their energy.

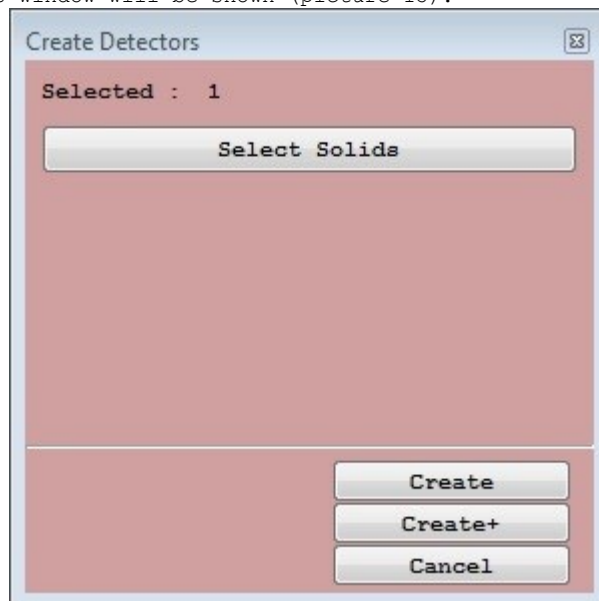
Use main menu command *Measuring Model -> Sources -> Add Sources* to create new source objects. Create Sources window will be shown (picture 14).



Picture 14. Create Sources window.

Press the **Select Solids** button to select solid geometry objects with the **Object selection** interface. When some solids selected, input the energy of radiating gamma quanta. Press **Create** or **Create+** button to create the sources.

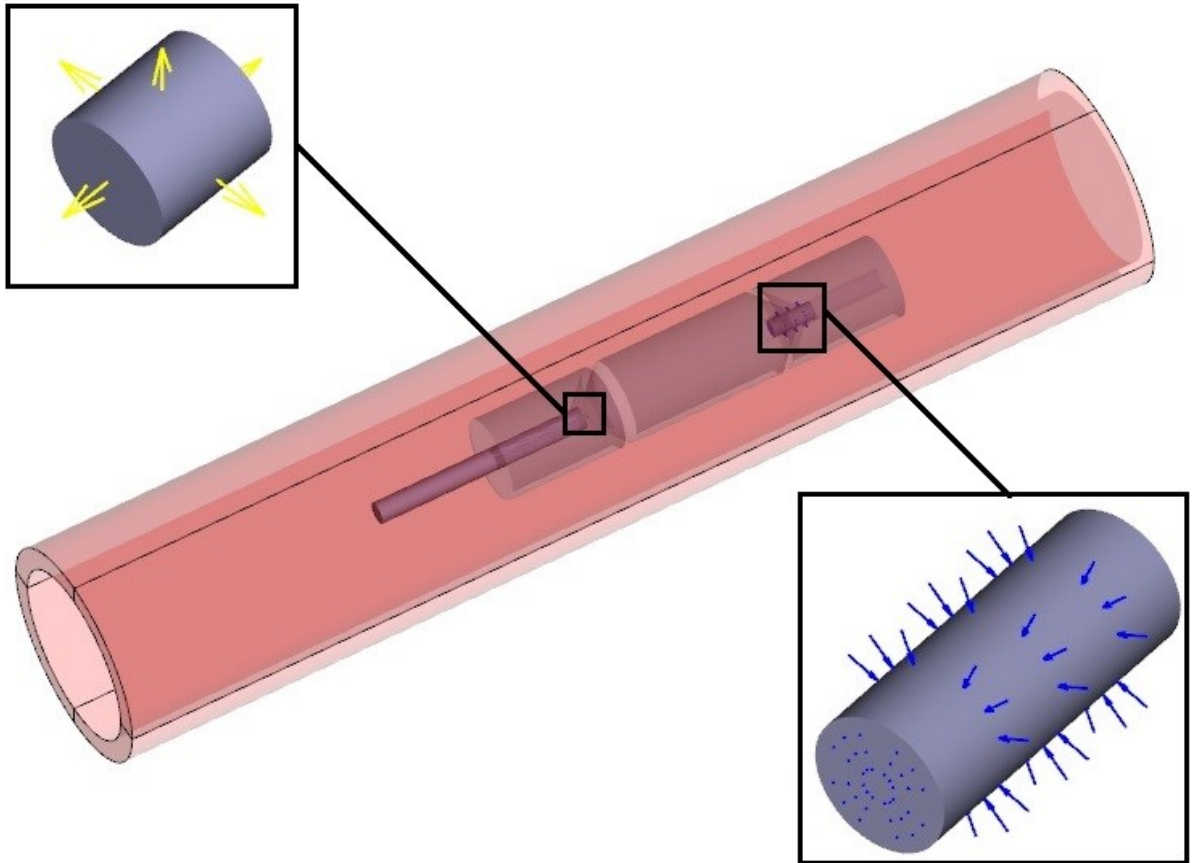
Use main menu command *Measuring Model -> Detectors -> Add Detectors* to create new detector objects. Create Detectors window will be shown (picture 15).



Picture 15. Create Detectors window.

Press the **Select Solids** button to select solid geometry objects with the **Object selection** interface. When some solids selected, press **Create** or **Create+** button to create the detector objects.

When some measuring objects (sources or detectors) are created, they can be shown on the Main Screen Window (using the window's Pop-Up menu *Plot->Geometry/Measuring*). An example of Measuring Model with one source and one detector is shown on picture 16.

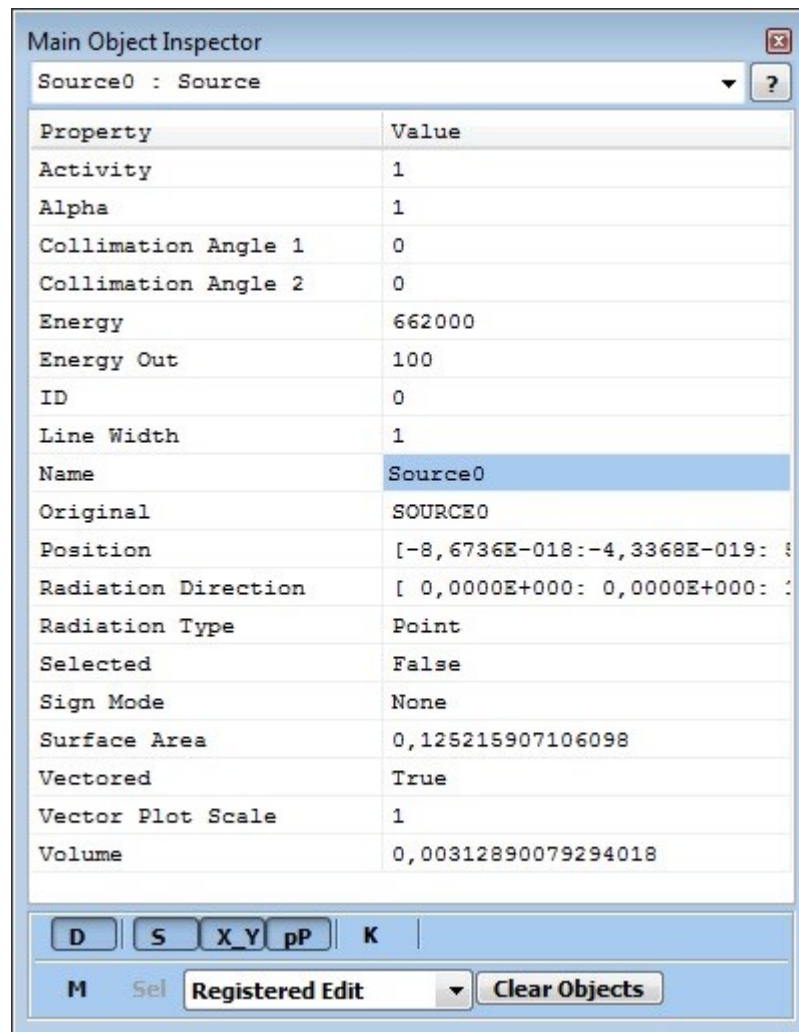


Picture 16. An example of measuring model view.

### Setting Source and Detector properties

When creating Sources and Detectors, only main properties assigned to the created objects. To edit other properties use Main Menu commands Measuring Model->Sources->Edit Sources or Measuring Model->Detectors->Edit Detectors. The Main Object Inspector window will be shown (picture 17).

The Object Inspector allows editing all source and detector properties defining the gamma quantum distribution modeling process (for an example, source energy, source activity and so on). Also, the object's view properties can be edited to display the measuring objects properly.



Picture 17. Main Object Inspector window.

## Solving Problem

Solving process can be executed by *Solution->Solve* main menu command. The solution progress can be viewed in the Log window.

Before solving the problem use main menu command *Problem->Option* (page Solution) to set appropriate solution option - total gamma quantum runs and so on. Solution options window is shown on the picture 18.

Main options are:

- Total gamma count - the number of gamma quanta to be run during entire solution procedure.
- Step gamma count - the number of gamma quanta to be run during one solution step.
- Randomizing mode - if is on, the random number generator will start with different value.
- Result for each step - if is on, the trajectories for each step will be saved in different result.
- Postsolve each step - if is on, the trajectory statistic and spectrum data of all detector object will be shown for each step.
- Result trajectories - the type of trajectories to save in the Result Model:
  - No Trajectories - no trajectory saved in result;
  - Detected Only - only trajectories caught by a detector saved in result;
  - Interacted Only - only trajectories having an interaction saved in result;
  - All Trajectories - all trajectories saved in result;
  - Failed - only failed trajectories saved in result.
- Analyse Duplicated Trajectories - if is on the solver will do analysis of duplicated trajectories and mark them with 'Bad' state.

Other options are recommended to keep default values.

NOTE: It is not recommended to set 'All Trajectories' with Total gamma count higher than 1000000 because it will take much memory and will lead to slow result plotting.



**Problem Options**

Descriptions Solution Graphics

**Presolver options**

Automatically update boundary ☒

Automatically update measuring ☒

**Solver options**

Total Gamma Count

Step Gamma Count

Point Inside Method =

Space Step Distance

Parallel Threads =

Randomizing mode ☒

**Postsolver options**

Result Trajectories =

Analyse Duplicated Trajectories ☐

Result for Each Step ☐

Postsolve each step ☐

Apply Cancel

Picture 18. Problem solution options window.

## Results and postprocessing

After Solution procedure successfully completed, the calculated result added to the Result model. Spectrum data for all detector objects are also shown in the Log window and can be saved as text file.

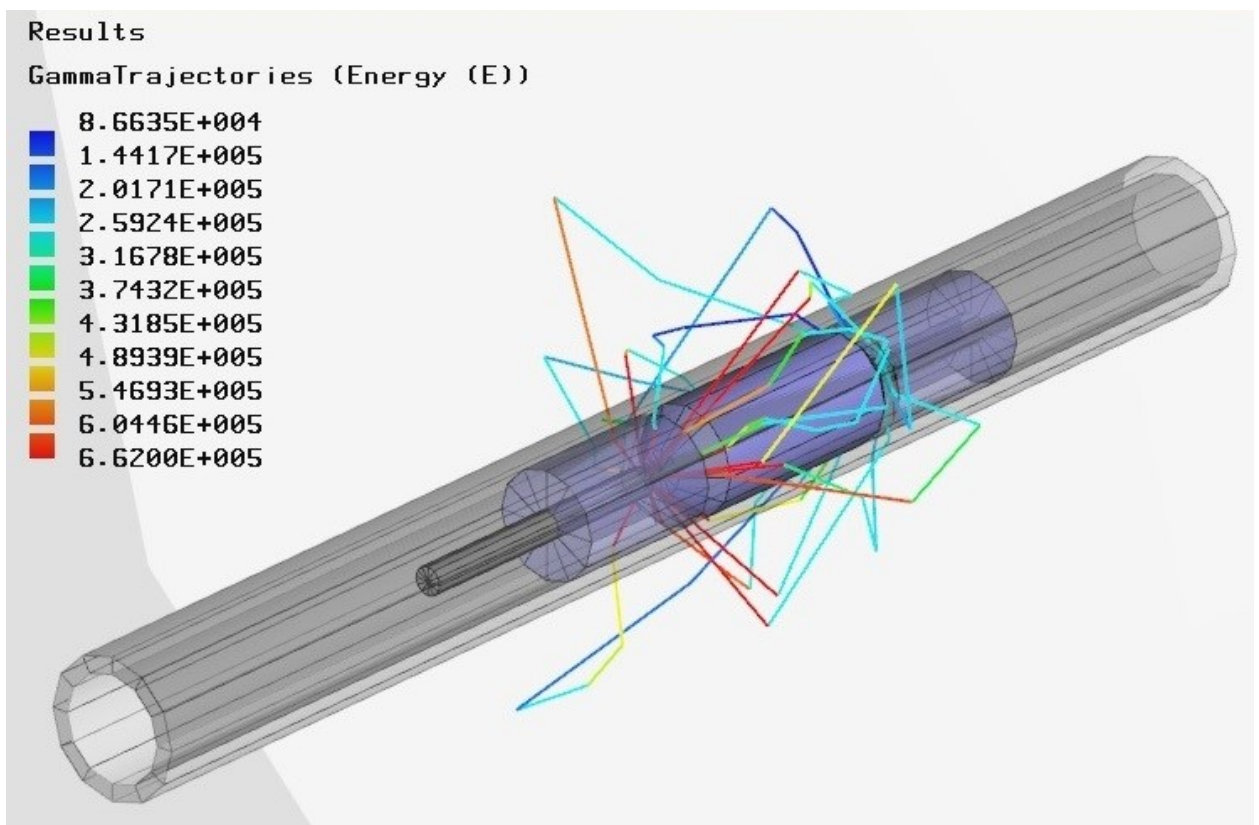
Calculated gamma quantum trajectories can be viewed by executing command *Result Model-> Plot Results*. The Plot Results form will be shown (picture 19). Select the result for plotting and press **Apply** button - the result will be shown on the **Main Screen**. The result plot style and boundary plot style can also be selected to plot result in appropriate manner.

An example of gamma trajectories result is shown on picture 20.





Picture 19. Plot result window.



Picture 20. Example of gamma trajectories result plot.

The result trajectories can be analyzed and managed using command *Solution->Calculate Results->Analyze Trajectories*. The *Trajectory Analysis* window is shown on the picture 21. Select results from the list and choose appropriate options. The following actions can be done:

- Finding duplicated trajectories in results.
- Changing trajectory state in results.
- Delete trajectories with specified state.
- Analyze nodes of trajectories (distances, directions).

After analysis completion the summary is shown in the *Log Window* or report shown in new text window.

### Trajectory Analysis x

**Results :**  

G-Trajectories1  
G-Trajectories2  
G-Trajectories3  
G-Trajectories4  
G-Trajectories5

**Duplicated Trajectory Analysis**  
Analyze Trajectories with State Detected ▼  
Weak trajectory comparison ☒  
Mark duplicated trajectories as Bad ▼ ☒  

Analyze Duplicated

**Trajectory Management**  
Particle State = Not Run ▼  
Include state in result trajectories ☒  

Manage State  
Delete by State

**Trajectory Geometry Analysis**  
Analyze Nodes with State Interacted ▼  
Analysis Coordinate System Cartesian (...)  
Geometry Analysis Type Distance ▼  

Analyze Geometry

Picture 21. Trajectory Analysis window.

Let us consider an example of trajectory geometry analysis. First of all - select the results you need to analyze in the result list. Then, select the node state to be included in the analysis - let it is Interacted. Next select the local system for the analysis (all results will be calculated in the Local coordinates of the system), let it is Cylindrical system. And finally - select the analysis type, let it is Distance. Press the *Analyze Geometry* button and the report will be shown. The part of report is presented on picture 22, where Values - the gamma energy; States - the gamma state at the trajectory node; r, phi, z - cylindrical coordinates of the node position.

The additional analysis of the report can be done by external standard statistical tools. For an example, as we analyzed distances in Cylindrical coordinate system, we can create the histogram of distribution of node distance from the Z axis by analyzing local r coordinate (because here the r is actually the distance from Z axis).

Values	States	r	phi	z
2.0134E+005	[Interacted,Detected]	3.5384E-001	-2.3634E+000	1.1024E+001
0.0000E+000	[Interacted,Dead,Detected]	8.6440E-001	-1.7706E+000	1.0275E+001
3.1589E+005	[Interacted]	1.6806E+000	1.8429E+000	5.3164E+000
2.4574E+005	[Interacted]	7.7099E-001	5.5043E-001	6.0548E+000
1.3644E+005	[Interacted]	6.8448E-001	2.3232E-001	7.2811E+000
9.5521E+004	[Interacted]	1.4804E+000	1.0159E+000	6.6285E+000
9.5453E+004	[Interacted]	1.4184E+000	1.0249E+000	7.0485E+000
0.0000E+000	[Interacted,Dead,Detected]	1.0299E+000	9.3886E-001	9.0711E+000
5.5445E+005	[Interacted,Detected]	2.4918E-001	2.9548E-001	1.0483E+001
2.9385E+005	[Interacted,Detected]	1.4355E-001	-1.6606E-001	1.0715E+001
5.3395E+005	[Interacted,Detected]	6.0729E-001	-1.8305E+000	9.9966E+000
2.2090E+005	[Interacted,Detected]	9.7384E-001	-6.8550E-001	8.9223E+000
2.5265E+005	[Interacted,Detected]	4.6492E-001	-2.3025E+000	1.0588E+001
2.2110E+005	[Interacted,Detected]	6.7196E-001	-2.3997E+000	1.0523E+001
1.9507E+005	[Interacted]	4.0327E+000	-1.9482E+000	7.3384E+000
5.7091E+005	[Interacted]	1.4242E+000	-2.9408E+000	5.6836E+000
5.0140E+005	[Interacted]	1.5568E+000	3.0204E+000	6.6531E+000
4.3707E+005	[Interacted,Detected]	1.1929E+000	2.9052E+000	9.3336E+000
2.3717E+005	[Interacted,Detected]	9.0936E-001	2.4268E+000	1.0023E+001
2.0820E+005	[Interacted,Detected]	6.2171E-001	1.7833E+000	9.6487E+000
1.8255E+005	[Interacted]	1.6960E+000	1.2821E+000	7.3793E+000
1.0986E+005	[Interacted]	2.0841E+000	1.3327E+000	7.2503E+000
2.1451E+005	[Interacted,Detected]	1.0789E+000	1.8178E+000	9.5030E+000
2.1025E+005	[Interacted]	3.2958E+000	9.3278E-001	7.1866E+000
1.5946E+005	[Interacted]	4.4450E+000	7.2608E-001	6.0645E+000
1.0471E+005	[Interacted]	4.7841E+000	7.5975E-001	6.1910E+000
6.4654E+005	[Interacted]	1.7584E-001	-9.2577E-001	5.9931E+000
2.1771E+005	[Interacted,Detected]	7.7819E-001	2.4834E-001	9.6761E+000
2.0571E+005	[Interacted,Detected]	8.8464E-001	3.7272E-002	9.5222E+000
4.9639E+005	[Interacted,Detected]	1.4167E-001	5.1386E-001	1.0142E+001
2.3020E+005	[Interacted,Detected]	2.1222E-001	-1.8650E-001	1.0295E+001
6.0745E+005	[Interacted]	3.5140E-002	-2.6117E+000	5.7534E+000
5.2415E+005	[Interacted]	6.2516E-001	1.6171E+000	7.4020E+000

Picture 22. A part of Geometry Analysis report.

## Solving Test Problems

There are several test problems, integrated in the REGIONS Gamma-Gamma program. They can be used as demonstrations of total solution procedure for simplest problems. Use main menu command Help -> Execute Tests to see existing test problems. Select a test in the shown Select Objects window and press Apply button to run test problem solution. Test Execution process can take several minutes. Test execution progress can be viewed in the Log window. After test execution successfully completed, the calculated result will be displayed.