SEMBA: BRIEF USER GUIDE (26/05/2016)

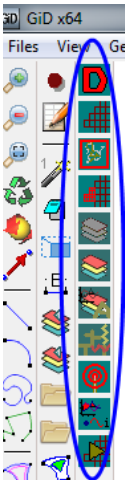
1. GiD A-UGRFDTD PROBLEM-TYPE

The structure of the tools has been fully embedded into a single tool (semba.exe) with all options managed under the GiD problem-type. Separate toolchains have been fully deprecated.

1.1. INSTALLATION

The Problem Type for A-UGRFDTD, named semba.gid must be copied under the problemtypes directory of the GiD installation tree. Ensure that the sub-directory bin is within semba.gid directory, containing the semba.exe and ugrfdtd.exe compiled tools.

The first time GiD is launched, go to Data and select the semba Problem Type. A lateral menu bar with new icons should appear. The new buttons have a characteristic dark-green color to identify them as part of the problem type



Updated problem types can safely overwrite previous ones. When opening a .gid created with an old problem type, just re-click in the semba one and choose to transform the data in order not to lose the previous information.

All quantities are defined in SI unless specified otherwise. Note that GiD always uses neutral units for the geometry. Scaling factors only serves to create the .nfde files with the right units by scaling all the neutral GiD units.

1.2. TOOLBAR DESCRIPTION

Simulation data is accessed by clicking the  button. This opens a window dialog as shown in **Figure 1**(left). The Solver tab let us modify the following options:

- *Solver:* Let us select the solver that we want to use. It defaults to ugrfdtd.
- *Ending:* Either specify the number of time steps or the physical time to end the simulation.

- *Final Time:* Defines the time at which simulation ends.
- *Time step:* Specify a time step for the simulation. If set to 0 the solver will select the maximum possible time step that warranties stability.
- *CFL:* Overrides the time-step chosen to force a desired Courant number.
- *Default sampling period:* Sets the value at which probes will sample data unless they are particularly set to a different value.

Another tab Figure 1**(left)** just applies to options for the ugrfdtd solver that are translated to with command-line options (for an updated list of them see [T5D1u1])

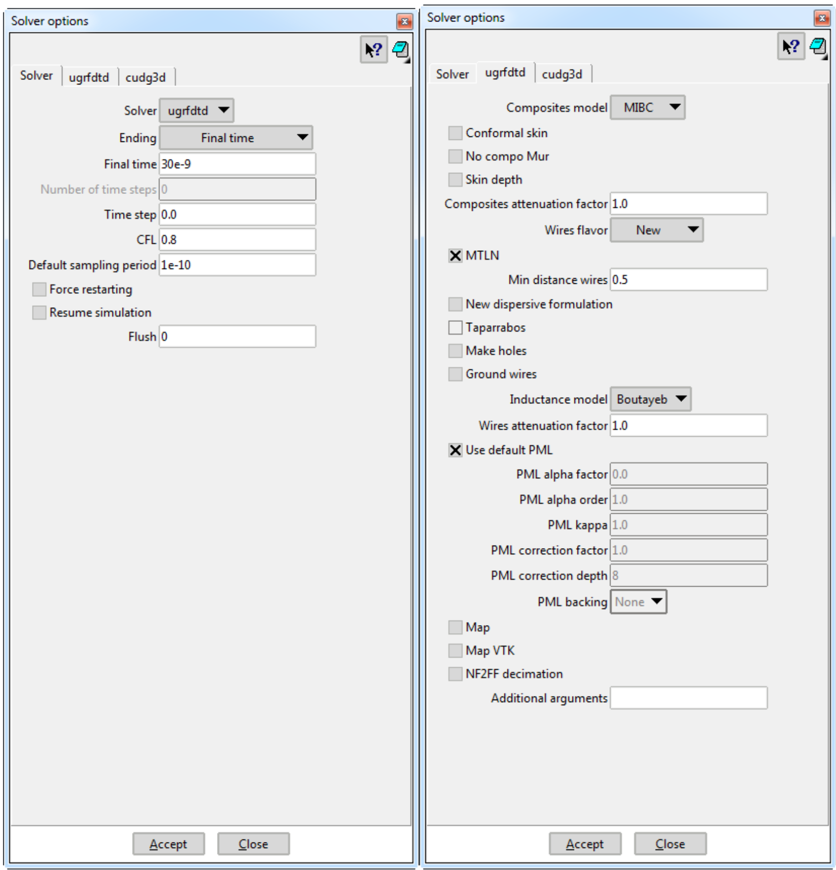


Figure 1: Solver options dialog

Mesher options: . The Mesher tab of this menu permits to choose between the conformal ugrMesher and the structured zMesher with options to produce postmsh GiD postprocessing files (to visualize the mesh) and/or .vtk files for their visualization withing Paraview. The conformal mesher can be tuned also choosing the forbidden length parameter (see [T5D1u1]).

Author: Salvador G. Garcia

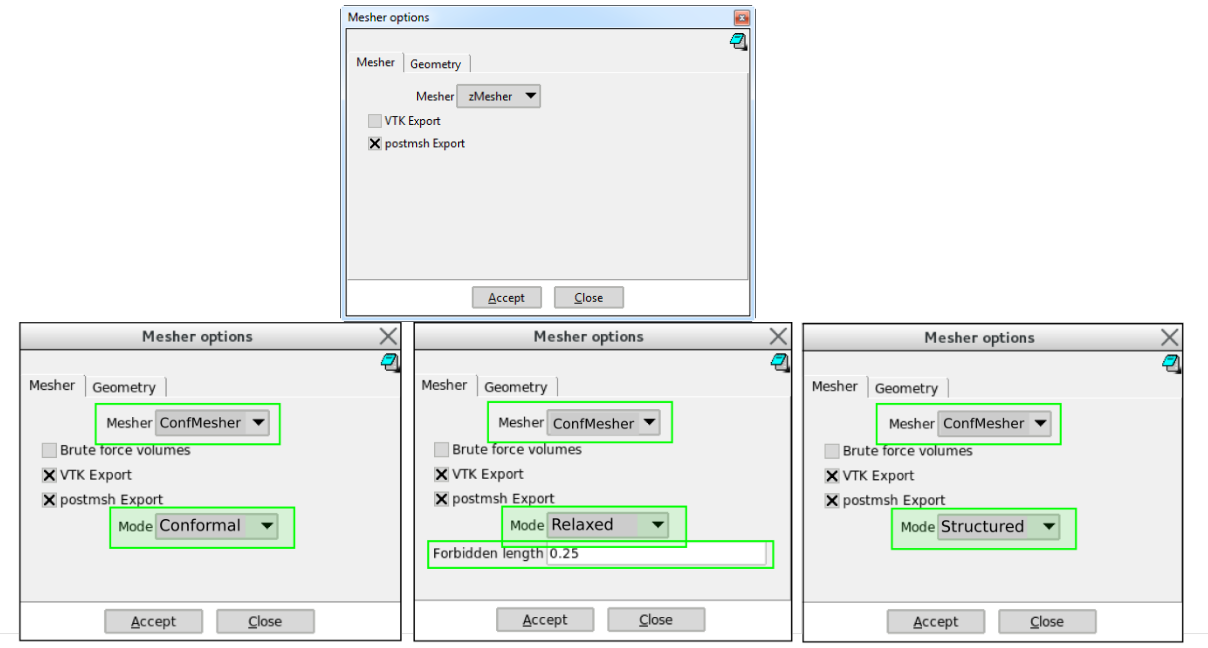
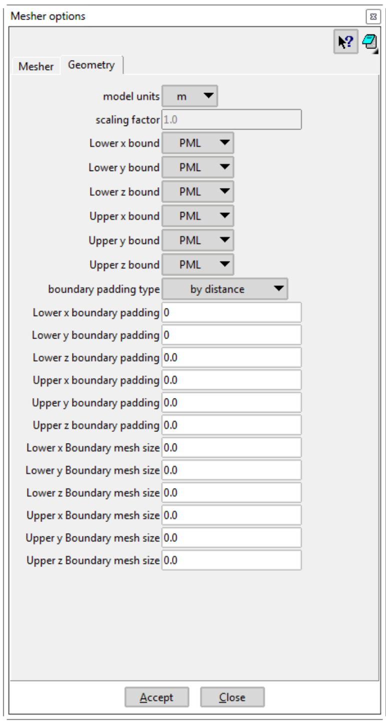


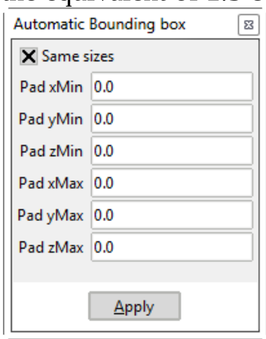
Figure 2: Mesher options dialog

The Geometry tab permits to choose

- *Model units:* Equivalence of natural GiD units. Note that the scaling factor DOES NOT AFFECT any other quantity while working in GiD (always uses neutral units). It only serves to create the .nfde files with the right units by scaling all the neutral GiD units.
- *Upper/Lower x/y/z bound*: Specifies the computational boundary of the domain. E.g. If you define a PML, several layers of absorbing material will be added to the domain in order to simulate an open space behind the boundary.
- *Boundary paddings*: This option has been added with respect to the June'14 version. It specifies the padding in natural GiD units added to the bounding box found with the Create Bounding box button,
- *Boundary mesh sizes*: The space step at the final cell of the Boundary padding (in GiD neutral units), immediately adjacent to the boundary conditions. E.g. If you define a PML, several layers of absorbing material will be added to the padded domain in order to simulate an open space behind the boundary. A non-uniform mesh grading algorithm is used to go from the discretization in the uniform mesh inside the Bounding Box, to the beginning of the boundary conditions.

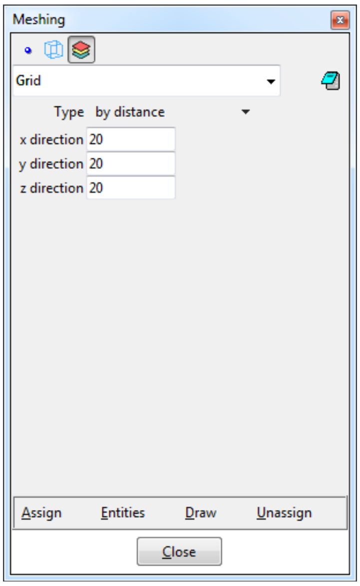


Calculate an automatic bounding box: When the button is clicked the program draws a bounding box surrounding the whole structure, including an optional padding region (in Gid neutral units). It is strongly recommended at this stage to leave at least the equivalent of 1.5 cells in equivalent GiD units as padding.



Meshing: This meshing data menu is opened by clicking the button, allows users to define meshing options over specific parts of the geometry. It includes one options

Grid is a condition (not a material!) that is assigned usually to the grid layer typically the one with the bounding box). It allows the user to define the number of cells in each direction or the geometrical size of the cells by distance (in the neutral units of the GiD model). It is mandatory to define this condition for proper work of the mesher. Double check that **only one grid is defined.**

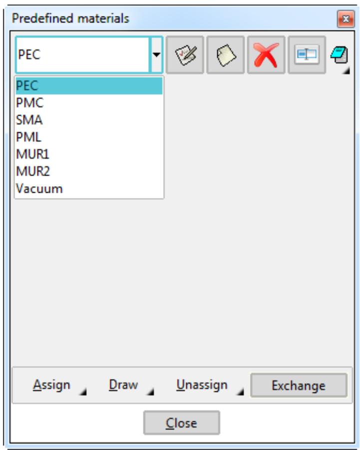


Physical models: The Physical model icons are used to specify respectively predefined basic materials, advanced material, and wire models on different parts of the geometry. Materials are assigned by clicking the *Assign* button and then selecting the entities where we want to assign the material. Each geometrical entity can have defined only one material on it. If a material is assigned to an entity that already has a material on it, the previous material will be removed. The *Draw* → *All materials* button will show what materials are assigned to each entity. New materials can be defined by clicking the *New material* button that opens a dialog to put a name to the new material. Buttons common to all their dialogs are

- Assign : select the geometrical entities with that material
- Draw : plots the conditions/materials (just to make sure that the assignment is correct)
- Unassign : erases previous assignments (use the first time to ensure that no duplicities exist).
- Exchange : loads a material database
- : updates the definition of the material with the current data
- : creates a new material with a new name
- : erases this material
- : renames the material
- : invokes other options of the problem type from inside this menu

Predefined Materials: A set of predefined materials are provided. The ones used in UGRFDTD are

- *PEC:* Perfect Electric Conductor materials can be defined only over surfaces.
- *PMC*: Perfect Magnetic Conductor materials can be defined only over surfaces.
- *Vacuum*: free-space
- *PML*: PML bodies to absorb waveguide modes.



Basic Materials:  Plain classic linear materials and dispersive materials

- *Classic material*: A classic material is a bulk material with constant electric permittivity, magnetic permeability, and electric and magnetic conductivities. These materials **can be defined only over volumes** (surfaces and lines are ignored since they lack of physical sense) When selected, they will show cases to put data corresponding to the material. *Permittivity* and *permeability* are relative to vacuum values. *ElecCond* and *MagnCond* are in SI units.
- *Dispersive material*: A bulk material with electric permittivity and magnetic permeability specified by a set of pole/residues data found with the UGRMAT.p Matlab tool (see [T5D1u1]). These materials **can be defined only over volumes** (surfaces and lines are ignored since they lack of physical sense)

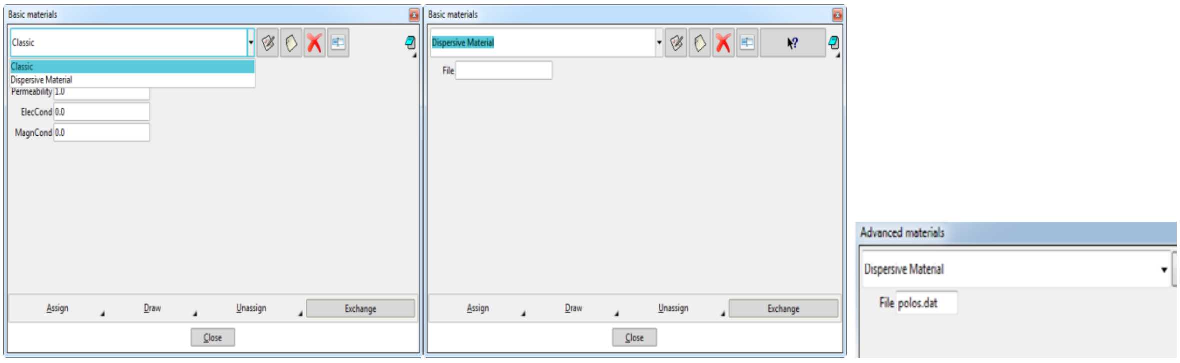
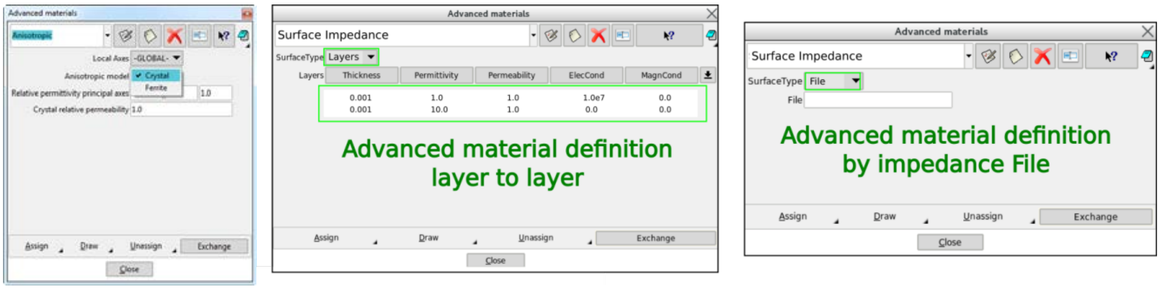


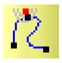
Figure 3: Basic materials dialog.

Advanced materials: Advanced materials menu is opened by clicking the  button. There are two kind of advanced materials:

- *Anisotropic material:* Characterized by their dielectric and magnetic tensors
- *Surface impedance:* These models can exclusively be defined over surfaces and they currently specified in two manners: By their pole-residue data trough input files, or by a set of layers specifying their constitutive parameters and their thickness.



Wires and terminations: Wires and terminations menu is accessed by clicking the  button. There are two kinds of materials within this dialog: the cable and the connector.

- Radius, resistance and inductance for a cable (Figure 5 left). The resistance and inductance are given in SI units per meter. Cables can only be assigned over lines. Two cables are considered to be different if they have different names.
- Connectors: (Figure 5 right)) are used to specify how a cable is terminated or connected to other cable or a surface. By default the solver will assume that the cables are terminated in an infinite impedance, unless they are grounded to PEC or lossy media (in this case a proper connection resistance is added). Connectors can only be assigned over segments; the user must create a small line close to the ending and define the type of connector over that segment. For this, GiD has a button  to divide a line into two by creating a point in it. Tangent vectors along the ending of the line must be oriented outwards its ending point in order for the mesher to correctly place the connectors. For this you can use the Utilities->Swap normals->lines together with the View->Normals->Lines option (Figure 4). Refer also to [eT1D1] for more information.

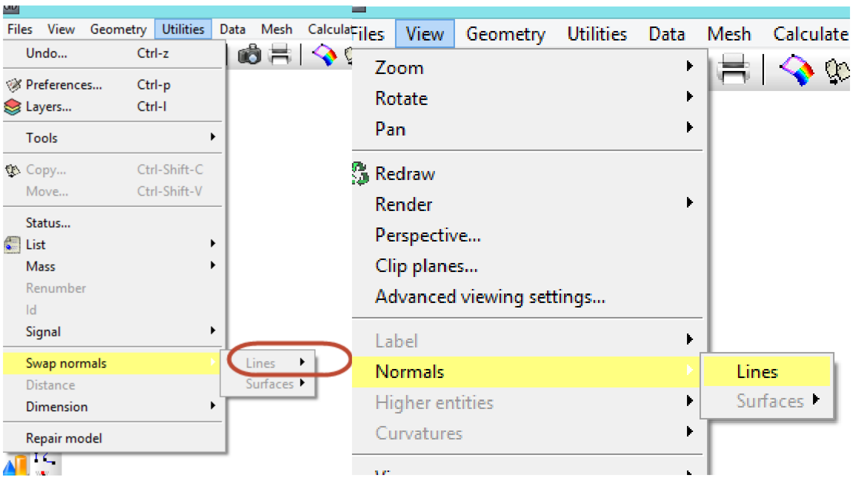


Figure 4: Normals (tangents on lines) orienting under GiD.

Each type of connector has a resistance, inductance, and capacitance values in SI units. Next to the cases appears a diagram of the circuit model used for that connector (see [eT1D1] for the actual implementation of these circuits in the wire solver).

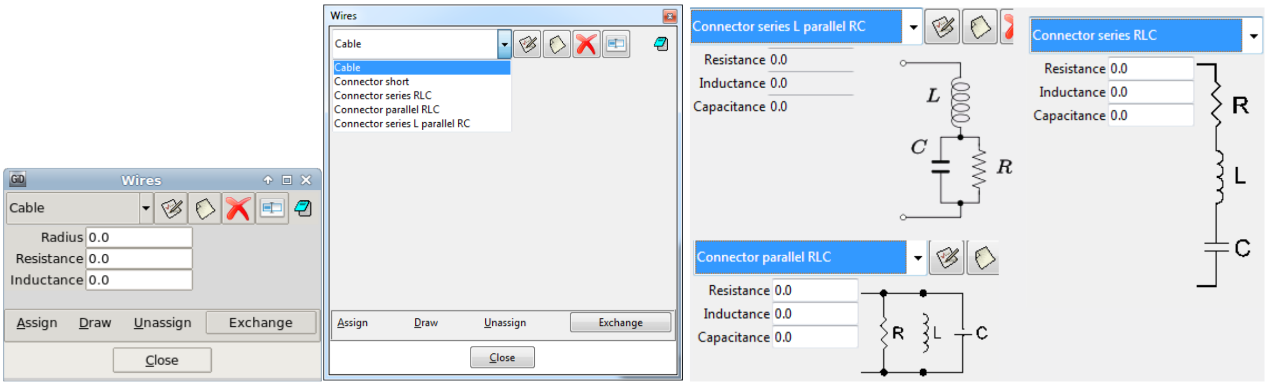
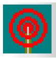


Figure 5: Cable and connectors dialog.

Electromagnetic sources: The electromagnetic sources menu let users to define excitations of fields. Figure 6 shows the dialog that opens when the button  is clicked. Only the following ones are currently supported:

- *Planewave on layer: (do not use the Planewave on volume –no longer supported-).* This excitation defines an incident plane wave with linear polarization. The direction cases indicate the direction of propagation of the wave. The polarization indicates the direction of the electric field vector. The field excitation allows the user to select between a Gaussian type $\left[\exp\left(\left(t-\mu\right)^2/\left(2\sigma^2\right)\right)\right]$ with μ being the time delay and σ the spread, or specified in an ASCII file. The planewave source must be assigned to a layer (e.g. the grid layer), on which a volume or a surface (not meshed) must exist to define the Huygens box limits. Remember to leave at least 1 cell (in equivalent GiD neutral units between the box and the geometry –except for symmetric problems with periodic boundary conditions-.

- *Nodal sources along lines:* Voltage and current sources (hard or soft) defined along a line.

- *Waveguide ports:* Currently support for TE/TM square waveguide excitation, and TEM ports exist.

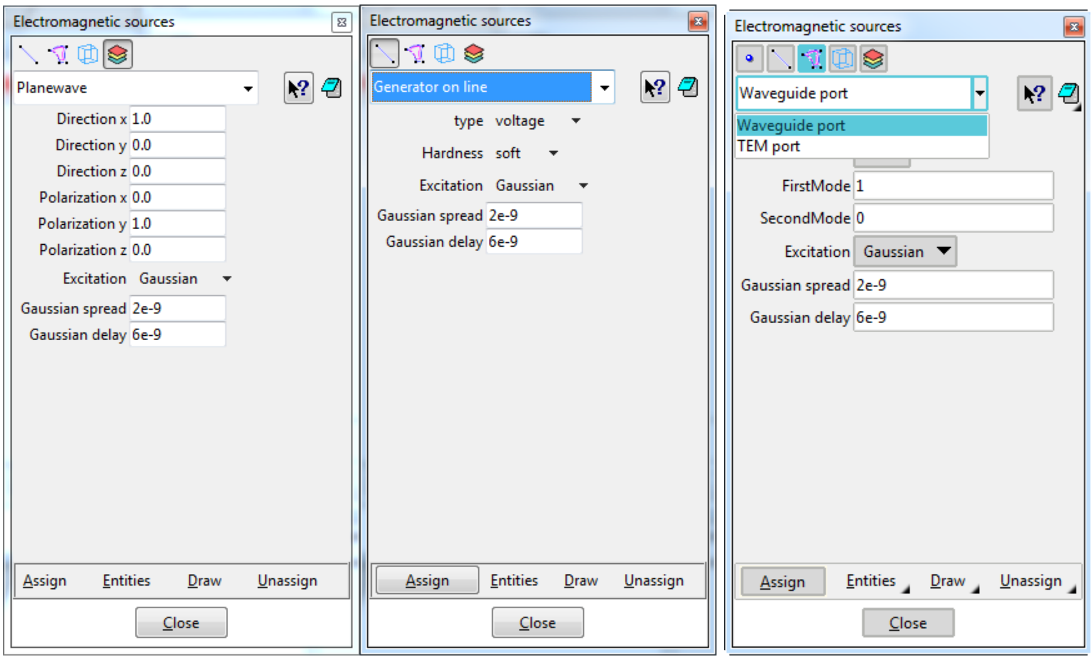
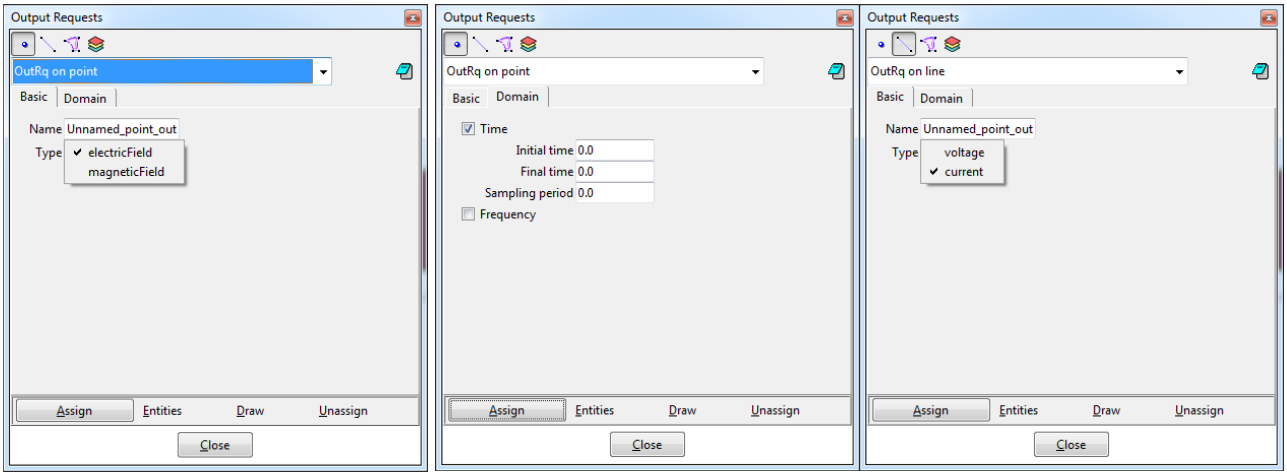


Figure 6: Electromagnetic source definitions

Output requests: Output requests can be defined by clicking on icon which opens a menu as the one shown in Figure 7. Surface and volumic probes produce .vtk and .xdmf animation files.



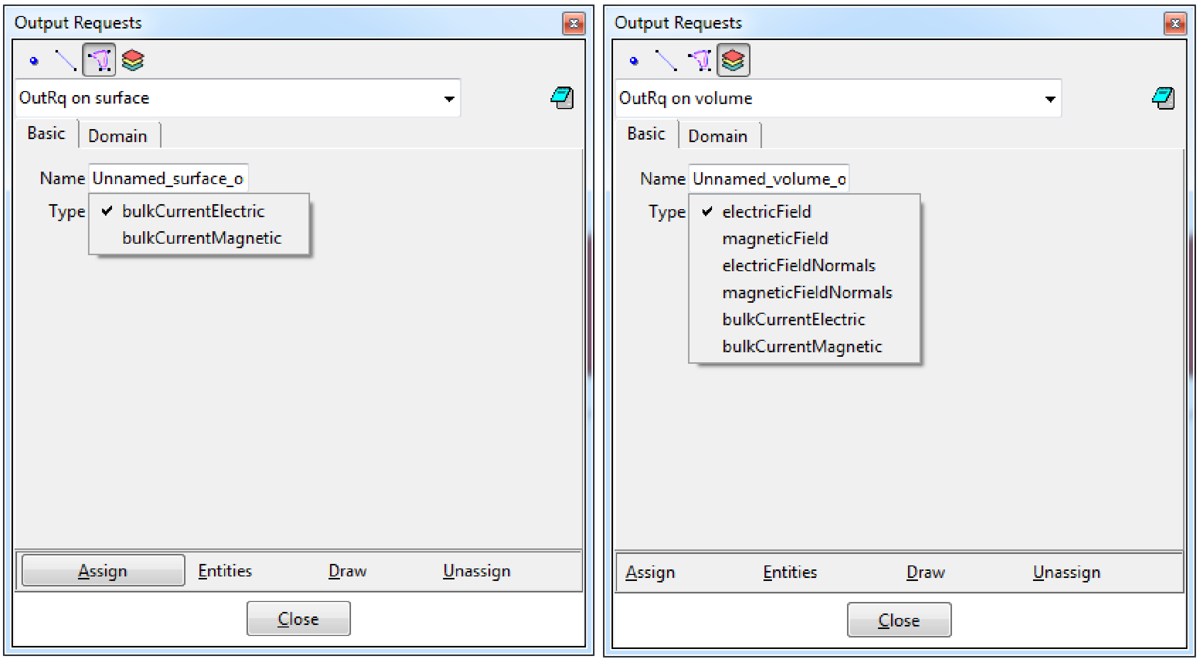
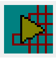
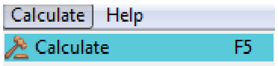


Figure 7: Output requests menu.

Generation of the .smb file: just click  after meshing the model.

Generation of the .nfde file: just click 

to launch the .nfde file creator. Upon successful completion the geometry can be viewed by accessing the Post-process tool of GiD.

In order to launch the ugrfdtd solver:

1. Copy the full xxx**.gid** directory to the simulating machine.
2. Launch, if necessary the .p/.m Matlab scripts for materials
3. Launch **ugrfdtd.exe** inside the ugrfdtd sub-folder.

2. GiD DEFINITION OF MULTIWIRES

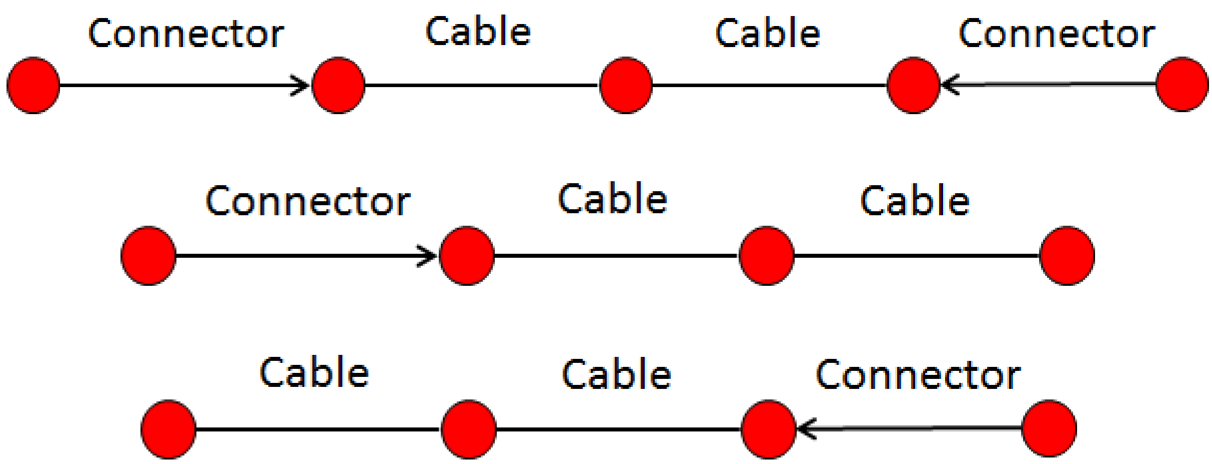
Wires are formed by any line whose associated material is Cable or Connector type. Cable type materials are described by their radius, resistance, and inductance per unit length. Connector type materials represents the connections between cables and another cables and/or groundings, which require their lumped impedance (currently RLC circuits, short-circuits and open-circuit). Connectors must be defined on lines, due to the impossibility of defining materials on nodes under GiD. In order to define connectivity rules, we make use of the GiD concept of Line Normals (in fact tangents to the lines). To display the direction of the lines, click *View-> Normals-> Lines* and select lines studied. To change the direction you should select *Util ->Invert Normals->Lines* and select the lines whose direction you want to change.

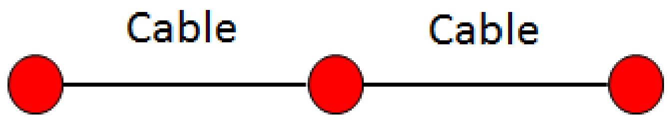
A Wire is composed of:

1. A series of lines **in the same Layer** with same Cable type connected as a linear graph. Doing so, they will be ohmically connected in an automatic manner with nodes along the cable having at most 2 lines attached. Loops are allowed, as far as the nodes at the crossing point are different (be careful with the “collapse” instruction in order not to created unwanted connections in loops).
2. If material type connectors are required at one end, the Cable-type line must be split into two lines, and a Connector must be assigned to its extreme. For that Connector-type line to be **CORRECTLY** attached to an end, it must have its Normal pointing to the intermediate cable-type lines (inwards to the point used for the splitting=outwards the ending point). If a Wire doesn’t have a Connector in an extreme, that extreme will be an open-circuit and will not be connected to another Wires, though it will eventually be grounded to lossy and PEC surface and bulk materials. Note that a connector-type material is a special case of a cable-type material with properties INHERITED from the cable properties of the cable-type pointed by its normal. **If a wrong direction is specified the connector may wrongly inherit other cable properties (in case of junctions) or even neglected (in case of grounded extremes).**

In summary, each cable may be finally composed by one or more lines: two optional ending ones, with the Connector conditions, and intermediate ones of Cable type, from which the connectors inherit their cable properties (radius and p.u.l. parameters).

For example:





In other to make connections cable-type and connector-type lines must be **IN THE SAME LAYER (different-layer cable and connector-type materials are considered isolated and never connected)**. They next convention is followed to determine their junctions

1. Node sharing cable-type lines (without Connectors):
    a. Wires with **different** cable-type are disconnected
    b. Wires with the **same** cable-type are connected only if junctions of **no more than two** of them exist. Otherwise they are disconnected.
2. Node sharing lines with Connector-type materials:
    a. If they have **different** connector-type materials wires are disconnected (the connector is thus only used in case of grounding to surface or bulk lossy/PEC materials is topologically found)
    b. If they have the **same** connector-type materials they are connected according to the connector properties (remember: for this the Connectors must be **correctly** attached to the extremes, that is the line-normal must be pointing **inwards to the splitting the node, and consequently outwards the connection ending point**, otherwise the mesher may abort and/or produce wrong connections).
3. Node sharing 2 lines, one with a Connector, and the other one with a Cable material : this situation must be avoided since it may lead to wrong interpretations depending on the normal orientation.

A safe practice, hence, to connect different wires is:

1. Place them at the same layer (lines in different Layers will not be connected)
2. Attach the desired cable-type materials to all lines belonging to the wires.
3. Make them share the same node (use the collapse tool if needed)
4. Split each of the wires near their connection point, and attach the **SAME** connector-type material **correctly** to the sharing node (line normal pointing outwards the connection node), otherwise interpretation errors and/or program aborts may occur.

2.1. WIRE SOURCES AND PROBES

To create a current probe or a source generator on a cable, it must be associated with a **POINT**. The line must be split at that point to correctly place the source or the probe, which will be attached to the (meshed) segment nearest to that point in the direction of the normal to the line. Hence **NORMALS** of lines attached to the source/probe point must have the **SAME** direction. Be careful for not compromising the connector normal criteria described above when defining sources/probes (don't use the same splitting points to define connectors and source/probes).