

CAE of Sensors and Actuators

Electrostatic Field Example

In this example, we would like to simulate the electric field and the capacitance of a **cylindrical** capacitor, including the surrounding air. The inner electrode consists of a centered rod and the outer electrode is the metallic coat. Between inner and outer electrode a voltage of 1 V shall be applied.

The dimension are as follows

- Radius of inner electrode = 1 mm
- Radius of outer electrode = 6 mm
- Thickness of outer electrode = 1 mm
- Height of cylinder = 100 mm
- Size of additional boundary = 10 mm

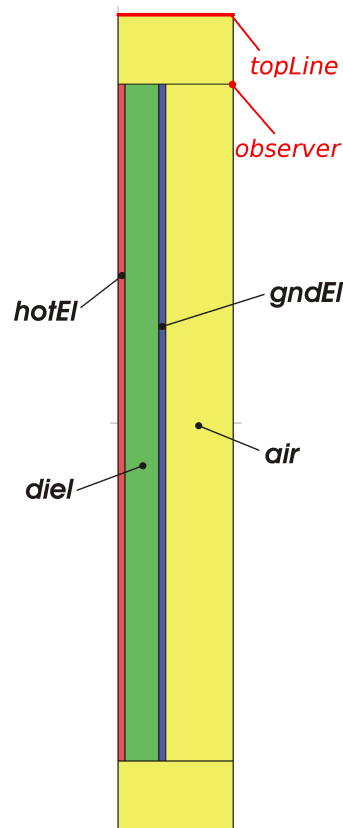


Figure 1: Domain regions of the capacitor model

This example is divided into three parts:

- Part I. Meshing with Ansys
- Part II. Simulation with NACS
- Part III. Postprocessing with Gid

Part II. Simulation with NACS

Tutorial: October 21st, 2014

Further information:

- `nacsBasics.pdf` - overview over NACS interface
- `electrostatics.pdf` - basic steps for modelling electrostatics

NACS simulation with the graphic user interface (gui)

- **Preparations**

1. create your mesh before starting NACS (→ see Part I) and place the `.nmf` mesh file into your simulation directory
2. start NACS gui by typing `nacs-gui` into linux command line
3. create a new simulation via **File** → **New Simulation**

- **Setup and run simulation**

All steps needed to setup and start your simulation can be found under the **Analysis** flag. There you can find on the left hand side of the gui a list of all setup steps starting with **Global** and ending with **Simulation**. In the following section those steps are applied to the electrostatic example.

1. **Global**

- **Global** → **Mesh**

Click **Load Mesh ...** to load the `.nmf` mesh file which you created in Ansys before. As our model shall be axi-symmetric select under **Select 2d Geometry Setup** the option **2D axi**. On the right hand side you can see an overview over the model. It is also possible to display only single entities. To do so unselect **Show Complete Model** under **Mesh Viewer** and select the wanted entities under **Entity List**.

- **Global** → **Coordinate System**

Only needed if special coordinate systems shall be used → not needed here.

- **Global** → **Entity Selection**

Here you have the option to manually select additional entities, e.g. additional observer points. As we did this already in the Ansys script we can omit it here.

2. **Material**

- **Material** → **Assignment**

In the middle column you will find a list of all regions. For a correct simulation you have to assign a material to each region. To do so select on or more regions and choose in the upcoming **Set Material** subwindow the desired material out of the list of **Available Materials** (all materials needed for your simulations will be provided). Now assign the following materials:

`air_region` → `air`

`diel_region` → `SiN`

`gndEL_region` → `Aluminum`

`hotEL_region` → `Aluminum`

3. **Model**

- **Model** → **Analysis**

Under this entry the type of analysis (eigenfrequency, static, transient, harmonic) and the physical field (electrostatics, mechanics, ...) are specified. For the electrostatic example select create a new analysis by clicking **New**. Select in the upcoming

subwindow **Physic Type Selection** -> **EL and Analysis Type Selection** -> **ST** and add it to the analysis by clicking in **Add**.

– **Model -> Region Assignment**

Here you select which regions shall be simulated. For the given example select under **Physic Regions** -> **Elec** the **All Regions** box and click on **Apply**.

– **Model -> Damping**

As there exist no damping model for electrostatics, this section can be skipped.

4. Boundary Conditions

– **Boundary Conditions -> Set Condition**

Click on **New -> Elec** to get a list of all possible boundary conditions for electrostatics (→ see [electrostatics.pdf](#) for detailed description). The task descriptions says that the capacitor shall be loaded with 1 V. The simplest way to model this outer voltage is to set one of the electrodes to a potential of 1 V and the other one to ground. To do so apply the following steps:

Go to **New -> Elec -> Potential**, select **hotEL_region** under **Entities**, set **Components -> value** to 1.0 and click on **Apply**.

Go to **New -> Elec -> Ground**, select **gndEL_region** under **Entities** and click on **Apply**.

5. Results

– **Results -> Set Result**

New -> Elec displays a list of all possible output quantities. Some of them can only be calculated on Regions (e.g. Energy) and other ones only on Groups (e.g. Charge). To avoid the selection of wrong entities, only the usable entities are listed under the **Entities** subwindow. For this example we want to calculate the electric field, the electric potential and the capacitance of the whole setup and additional the electric potential at the observer point.

* Electric field:

To calculate the electric field of the setup select **New -> Elec -> FieldIntensity** and enable the **All Regions** box.

* Electric potential:

The electric potential in contrast to the energy and the electric field can be calculated also on groups. The reason for this is, that some results like the electric field and the energy are **element** results, i.e. they can only be calculated on area or volume elements. As groups consist only of nodes or lines, one has no such elements there. The electric potential however is a **nodal** results and can thus be calculated everywhere a node exists, i.e. nodes, lines and regions. To calculate the electric potential of the whole setup and extra on the observer point go to **New -> Elec -> Potential** and select the **All Regions** box and confirm with **Apply**. Then go again to **New -> Elec -> Potential** and select now **observer_group** and under **Monitor Output** the box at **text** (see below for explanation) Do not forget to click on **Apply** as otherwise your selection is not stored.

Important note: Inside the NACS gui some sections (e.g. the section **Results**) are very large. Therefore it might be possible, that some buttons are not visible as they are out of the screen. If you encounter this problem you can try to win some space by grabbing the icon bar between menu bar and **Analysis** tab and move it asside. Another option is to change the resolution of your screen.

* Capacitance:

The capacitance cannot be selected as result, so one has to calculate it by hand out of the available results. One possibility is to calculate the electric charge Q on the electrodes and use the known driving voltage $U = 1$ V to calculate the capacitance via $C = Q/U$. But as said above we cannot calculate the charge on regions, but only on groups and as the electrodes are regions, this option will not work here. The alternative is to calculate the total electric energy W of the setup. Via $W = 1/2 CU^2$ one can then calculate the capacitance. For this approach one

has to consider that one needs the total energy which is stored in the capacitor, which includes not only the energy in the dielectricum but also the energy in the surrounding air (→ stray field). In that sense the size of the surrounding air region will influence the value of the capacitance, so make sure that this region is already such large that most of the energy is captured. To calculate the electric energy select **New -> Elec -> Energy** and enable the **All Regions** box. Also select the text output to be able to read out the electric energy later for the computation of the capacitance.

– **Results -> Result Format**

In general there exist two types of results in NACS.

* **text results**

Text results are written to simple text files in the folder **history** in your simulation directory and store column-wise data which can be read in into matlab, gnuplot, etc. Those text files are created **for each** node / element (depending on result type) in the selected region or group. The only exceptions are regions results like the energy. For those results one file is created per region. To write out text files you have to select the **text** under **Set Result -> Monitor Output**. Attention: This option holds for all selected regions and groups, so if you want to write out the electric potential only at the observer point but want to calculate it on all regions, you have to create two results of type electric potential like described above.

* **mesh results**

Mesh results in contrast to text results are not written to separate files for each node / element. Instead the specified result at each node / element is written together with the information about the geometry (i.e. with the mesh) to a single file. Those files are either of the Ansys format or the Gid result format and have to be opened with the corresponding application (→ see **gidTutorial.pdf**). For this example as well as the assignment tasks the Gid format is used, so select it via **Mesh Results -> GiD Format**.

6. Solution

– **Solution -> Solver**

This category allows to change the algorithm which solves the internally created equation system. However, at the moment there exists no other option than **automatic**, so this category can be skipped.

7. Simulation

– **Simulation -> Start**

Click on **Save and Start** to start your simulation. Simulation with multiple steps (like transient or harmonic) can also be stopped and resumed.

- **Bonus: Check python script**

All who are familiar with the Python scripting language can take a look at the file **nacsTutorial.py**. This Python script contains all commands which were selected in the gui. It is in fact possible to setup the simulation just by creating a fitting python script. However, unless you really know what you are doing, use the gui for setting up your simulations.