Getting started with CandyStore

After this tutorial you will know the basic functionality of CandyStore. You will have performed a Eigenfrequency analysis of a hinge. In detail you wil do the following steps:

- Start CandyStore
- Import an STL Geometry file
- Create Meshes
- Create nodesets
- Create boundary conditions
- Create materials
- Create sections
- Save your data
- Export a CCX Input Deck
- (Optional: Run an analysis with CCX)
- Open the results file of the hinge
- Select various result steps and results

What you need:

- A working copy of CandyStore.
- The geometry file hinge.stl (Available in the test-files folder under your installation directory of CandyStore
- A working copy of CCX (only if you want to perform the analysis
- The results file hinge.frd (Either self-created, or available in the test-files folder)

1. Starting CandyStore

Windows:

Open the start menu navigate to the CandyStore folder and open it. Click on the CandyStore Item and CandyStore will be started.

Linux:

Start CandyStore by entering ./CandyStore in the directory, where the binary is located. Alternatively you may want to put an alias in your *rc file.

If you start CandyStore for the first time you will be asked for the location of the help file. You will find the helpfile in the help folder under your installation directory. Select the help file and click O.K

After starting CandyStore you will see the following window.

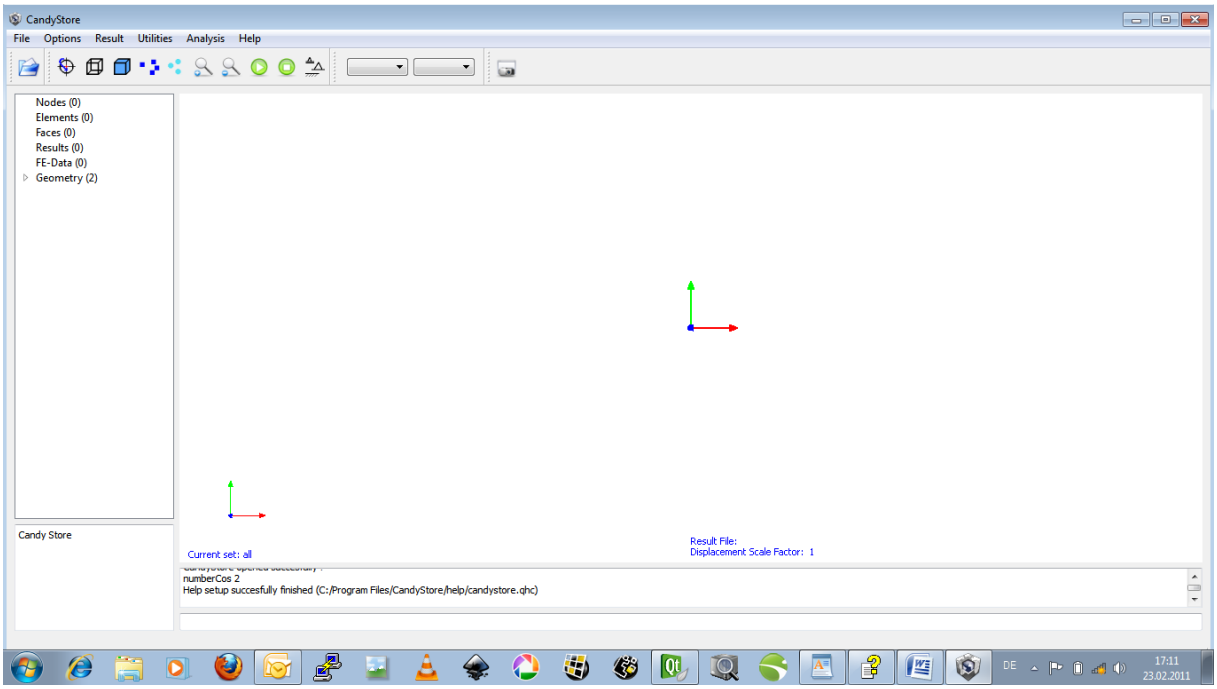

Fig. 1: CandyStore at startup

2. Importing a STL Geometry file

Let's get right into the action! Go to the file menu and click on the import STL Button. In the file dialog select the hinge.stl file.

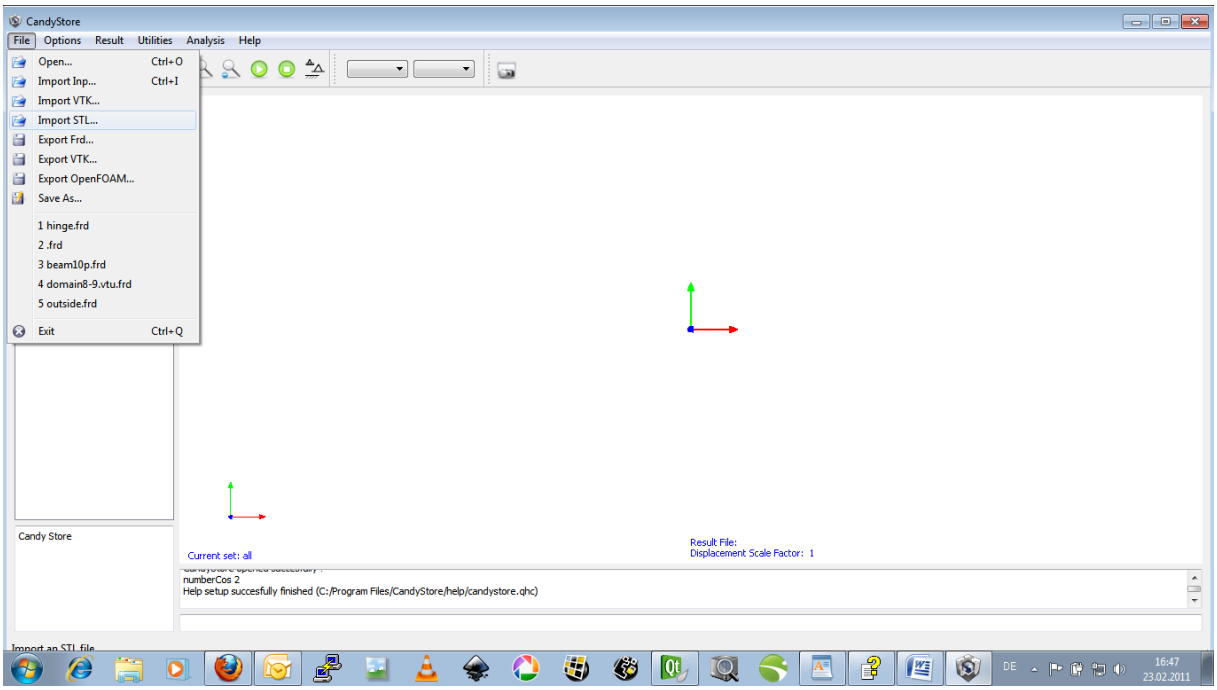

Fig. 2: Importing STL files

After the hinge has been imported you can see, that the model tree on the left hand side has been updated. The Geometry Item now holds 1812 items. Expand the Geometry Item by clicking on the small + and you see which geometry items are currently in the model. 598 Points and 1212 STL Triangles have been imported. The two listed COS (Coordinate Systems) are created by default on startup and always appear.

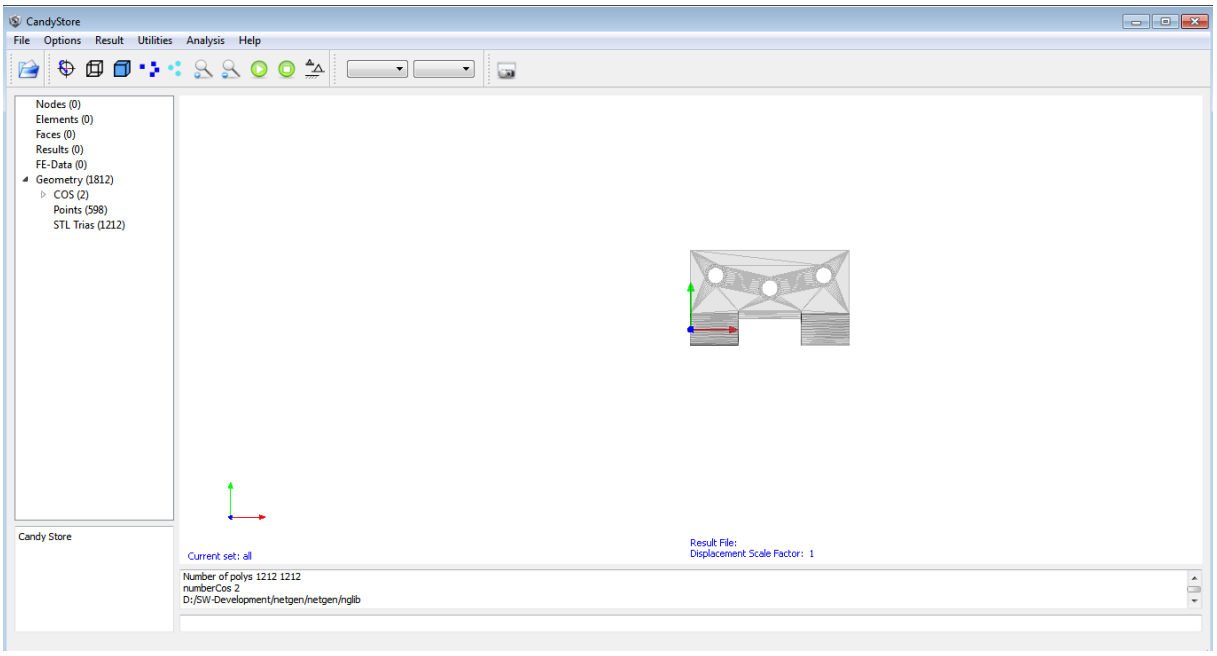

Fig. 3: STL Geometry of the hinge

3. Meshing

CandyStore uses the netgen meshing kernel.

To mesh the stl geometry you have two commands available:

mesh2d <hmax> <secondorderFlag>

creates a 2D surface mesh of the stl geometry and

mesh3d <hmax> <secondorderFlag>

creates a 3D volume mesh.

At the moment the mesh fineness can only be controlled by the hmax parameter. (See netgen manual for detailed description) In simple words: The lower the value the finer the mesh will be. If you are not satisfied with the created mesh, you can delete it with

delmesh

This command deletes all nodes and elements in the model.

For the hinge a hmax value of 1 is a good choice. Mesh the geometry with second order elements:

mesh3d 1 1

or with first order elements

mesh3d 1

This may take a few seconds.

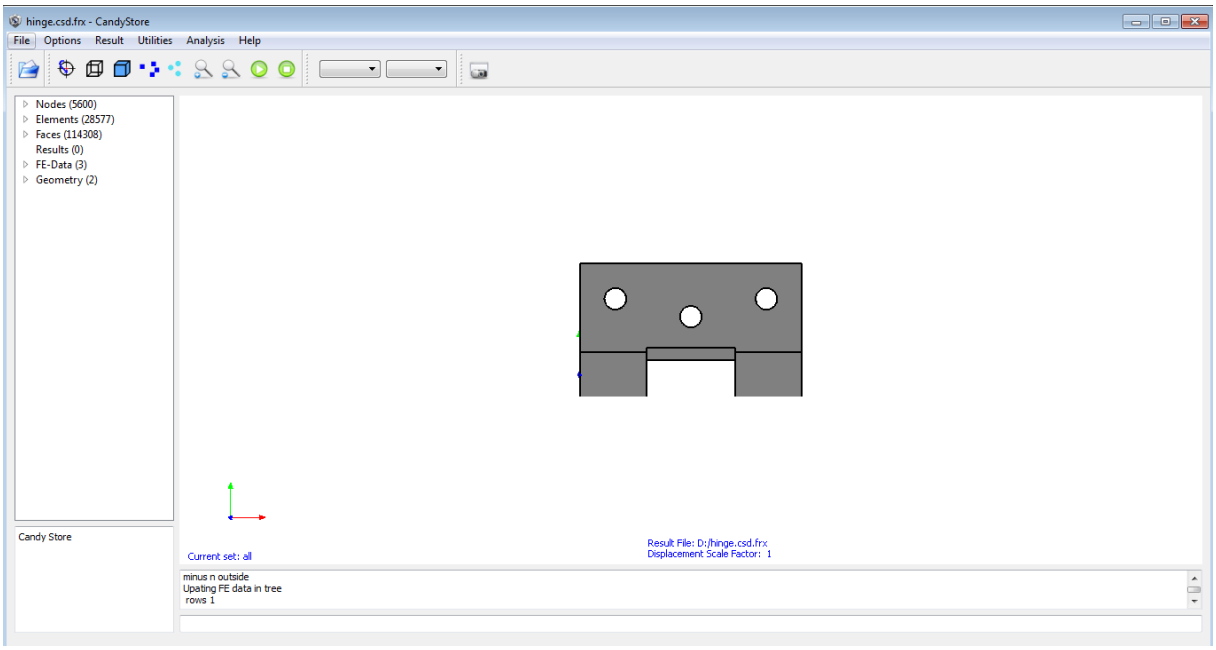

Fig. 4: The hinge after meshing

4. Creating nodesets

As the mesh has been created a nodeset called “outside” has been automatically created. The corresponding item can be seen in the model Tree on the left, when expanding the node item. Before creating a new nodeset, you have to plot some nodes in order to select them. Either click on the ToggleNodes Button in the Toolbar, or use the plot command.

plot <entity> <set>

Plots all entities of a given set. Now, plot all outside nodes of the hinge with the command

plot n outside

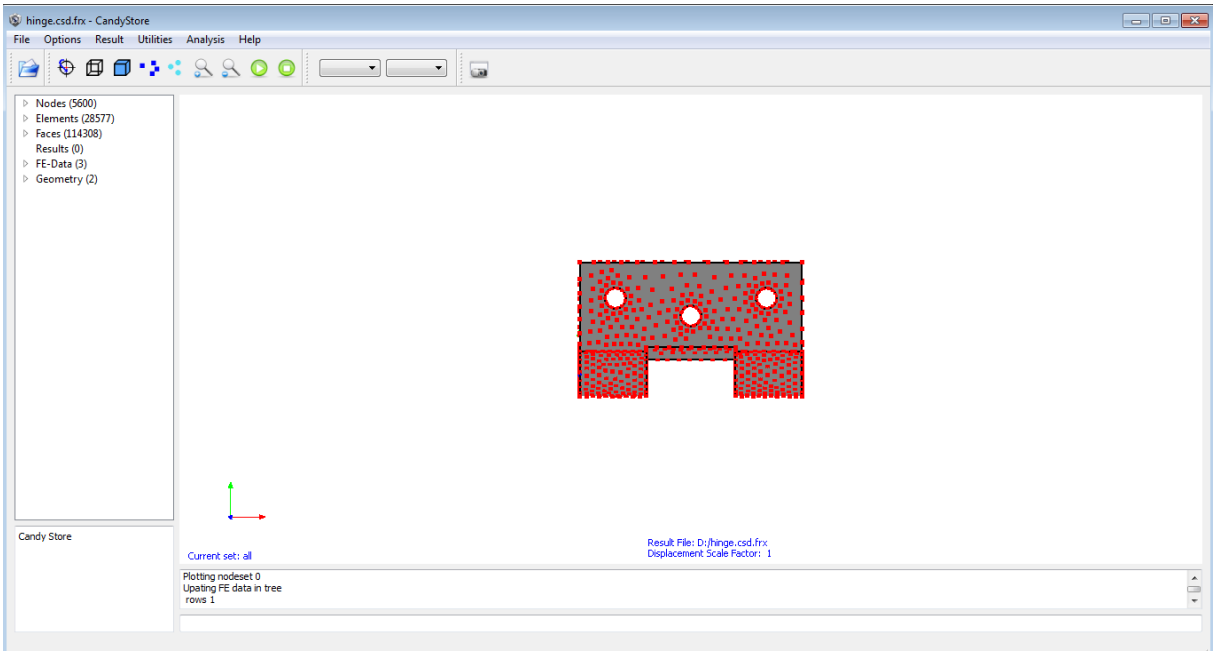

Fig. 5: Plotting set "outside" (command: "plot n outside")

To create some other nodesets enter the command

qadd <name>

This command will create a set called “<name>” if it does not already exist. Furthermore the set will be opened and you can place nodes in it. The current set is displayed in the lower left corner of the graphics window.

As you entered the qadd command a rectangular box appeared under the mouse pointer in the graphics window. This is the selection box. You can modify the size of the selection box by pressing r , then moving the mouse and press r again. The box will be recreated between the two “r”.

The q command always starts in individual selection mode, which means, that only one of the items in the box will be selected (the one with the lowest Z-Buffer value). By pressing “a” when in “q-mode” the selection mode changes to “all”. Now all entities in the box will be selected. By entering “I” you can return to individual selection. The selection mode is displayed in the lower left corner of the graphics screen.

Typing “q” the second time quits “q-mode”.

Let’s create a nodeset with all nodes, that will be fixed afterwards.

Enter

qadd fixation

Now select all nodes on the upper side of the hinge. When all nodes are selected press “q” to exit “q-mode”.

You can see the new set in the model tree.

5. Creating boundary conditions

To create a boundary condition click on the BC Symbol. You will see it after you have clicked on the FE-Data item in the model tree.

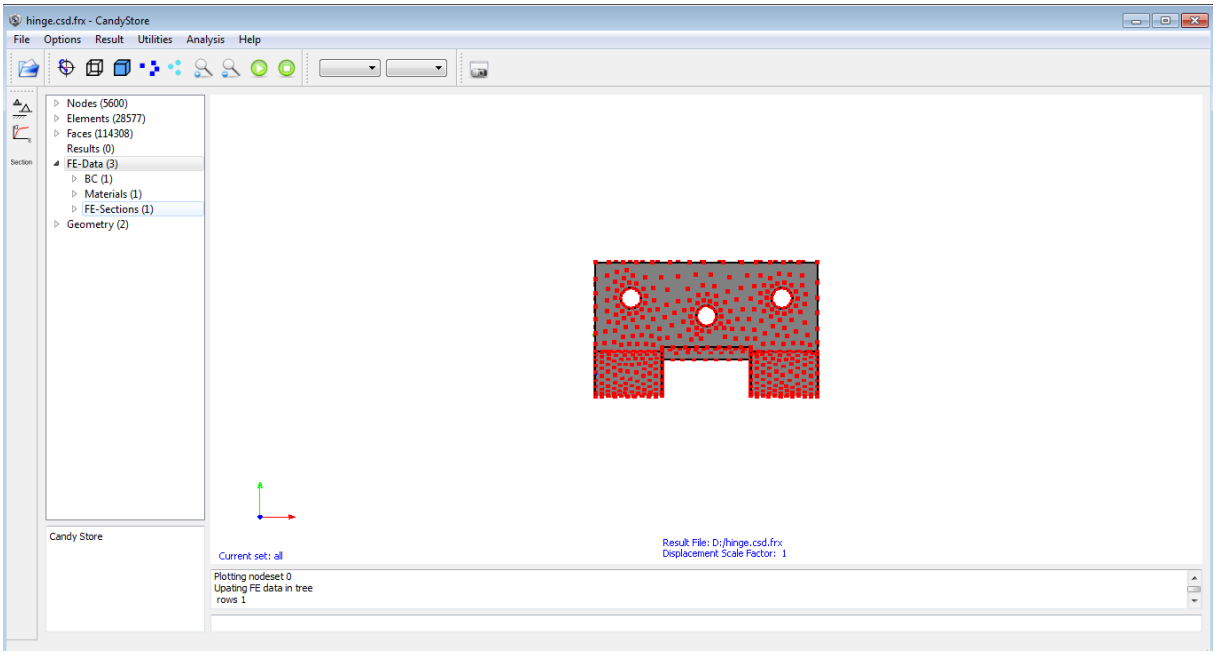

Fig. 6: Expanding the FE-Data item

Create a new BC with name fix, and the fixation set as target set. Enter 1 for the first degree of freedom and 3 for the last DOF. Enter 0 in the value box. To create the BC click “Apply”.

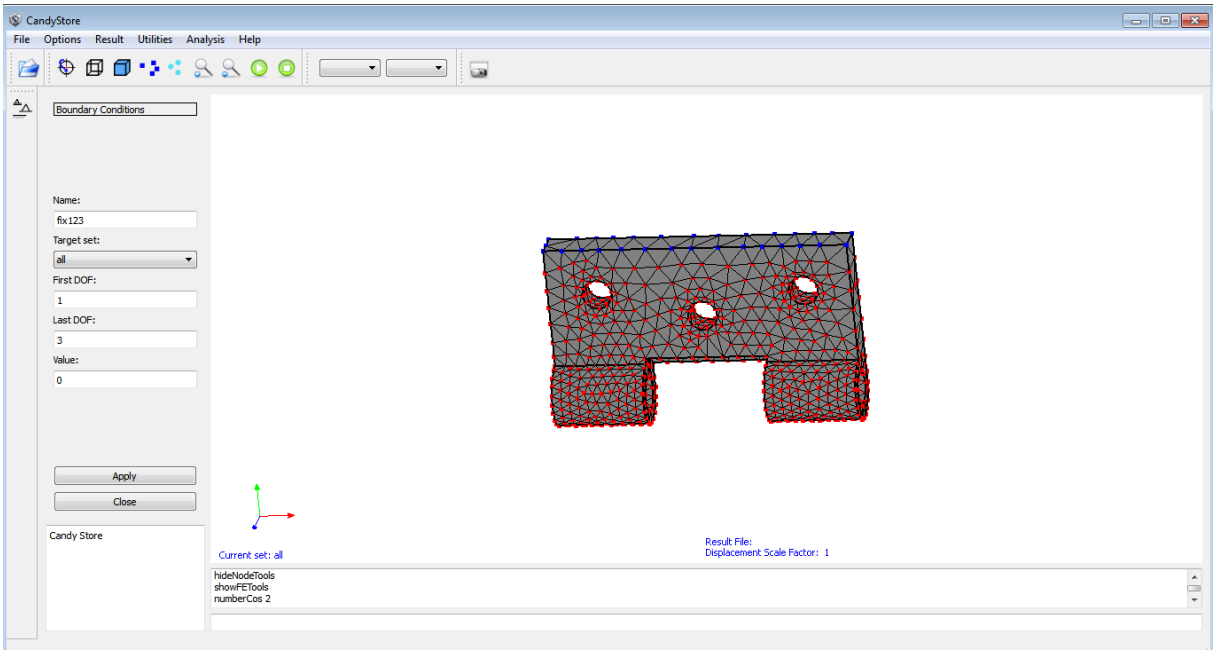

Fig. 7: Creating a boundary condition

6. Creating material

To create a material click on the material Symbol. Like the BC symbol it becomes visible when you click on the FE Data item.

Enter Steel as name, 2100000 for the Young’s Modulus, 0.3 for Poisson’s Ratio and 8E-9 for the density. Click apply to create the material.

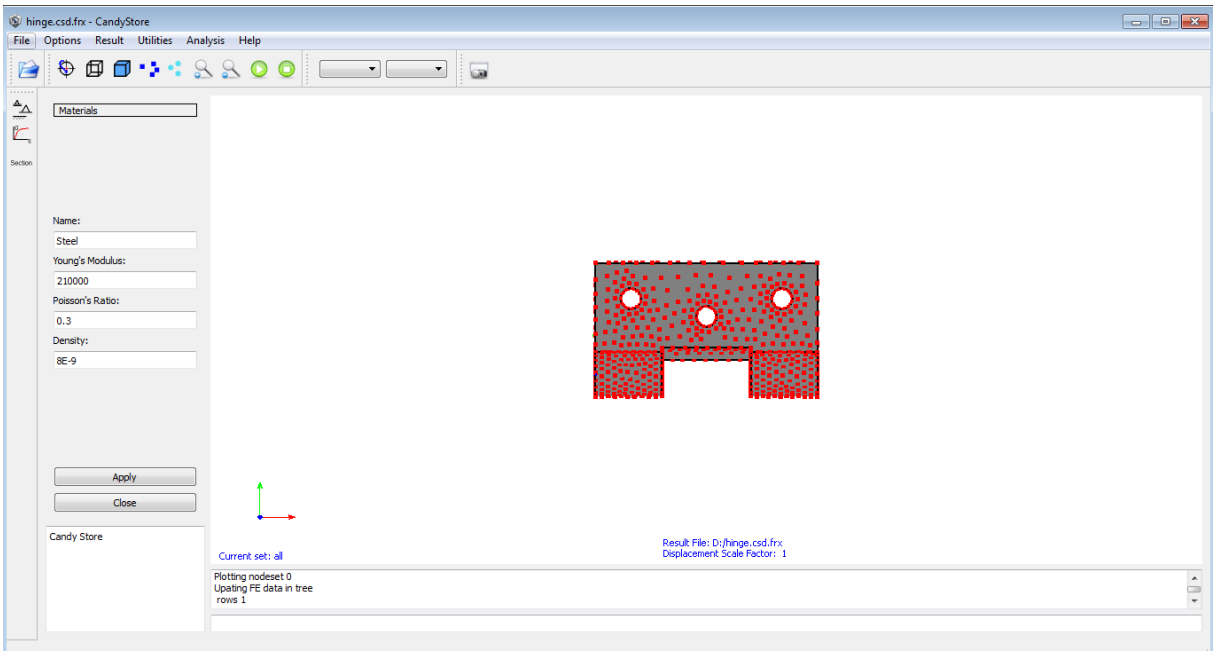

Fig. 8: Creating materials

Alternatively you can create a material by entering the command:

material <name> <Youngs Modulus> <Poisson Ratio> <Density>

The new material will appear in the model tree.

7. Creating a FE section

To create a FE section click on the section button, located in the same toolbar as the material button. In the two Boxes you will find all available element sets and all available maerials. Create a section with name hinge, target set all and the previously created material Steel.

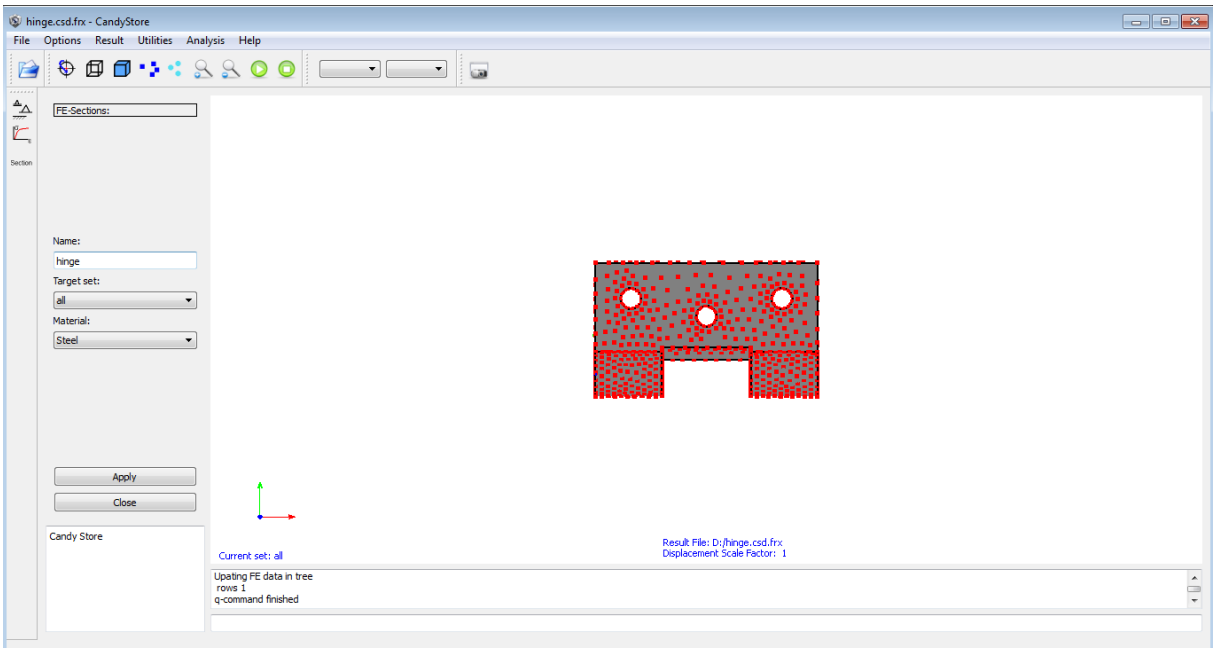

Fig. 9: Creating FE Sections

Alternatively you can create a fe section by entering the command:

fesection <name> <elementset> <material>

The new section appears in the model tree.

8. Save your data

You can save the current model by clicking on the “Save As” Button in the File menu. CandyStore will create two files: The *.csd file (CandyStore Data), and a frx file. The frx file is a regular frd file an can be opened by CGX. It contains all mesh data and, if available, result data. The csd file is an xml file, which contains the link to the frx file plus the additional data. (Sets, Materials,…)

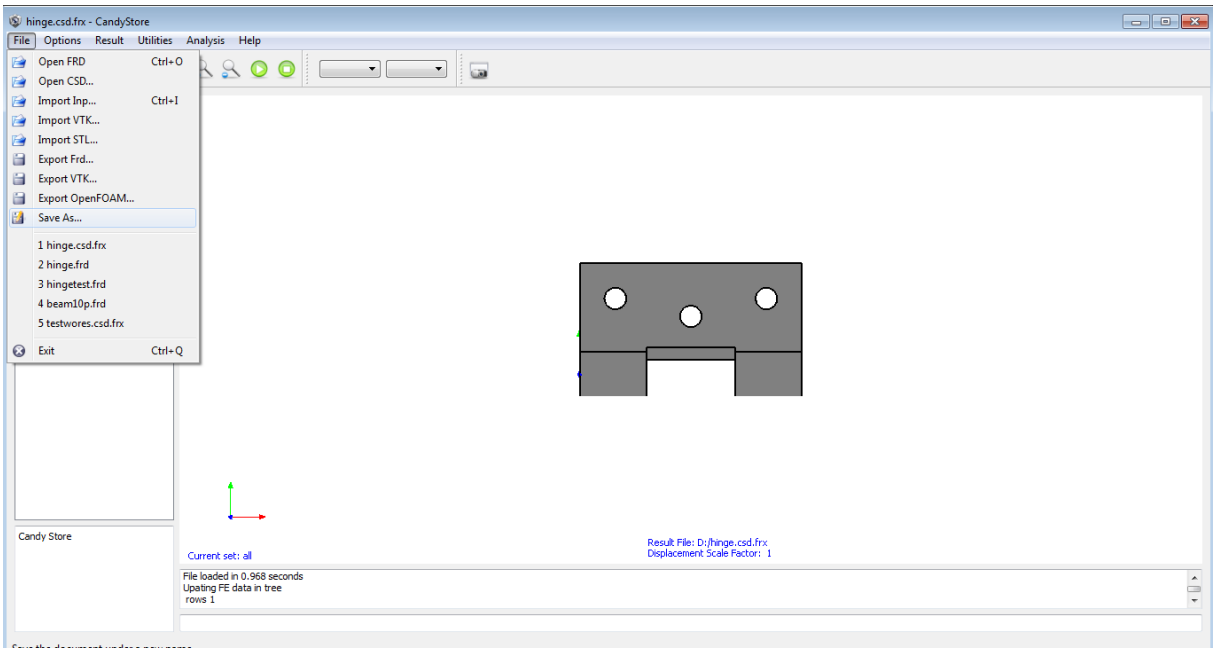

Fig. 10: Saving CandyStore data

9. Export a CCX input deck

With the current model definition, we can analyse the Eigenfrequencies of the hinge. However CandyStore has no solver included and you have to use an external FE Solver. The Solver of choice is CCX. To export an CCX input deck click on Analysis/Export CCX Input Deck and save the file. A frequency step is always added to the input file automatically. (Windows Users: A batch file for execution of CCX will be created in the same folder. If you have specified your CCX binary correctly (Options Menu) you just have to execute the batch file to run the analysis) .

10. Run the analysis

Follow the instructions of the CCX manual. You can get a copy of CCX at www.calculix.de and a windows binary at www. bconverged.com

11. Reading results

After the analysis has been finished a frd file is available. (If you have not performed the analysis, you can find a result file of the hinge in the test-files folder.) Open the frd file in CandyStore.

When the import has finished, you can see the mesh in its base state without results. To select a result of the analysis click on the appropriate item in the model tree. All results are listed under the results item. To load a result click on it. Let's load the first frequency result.

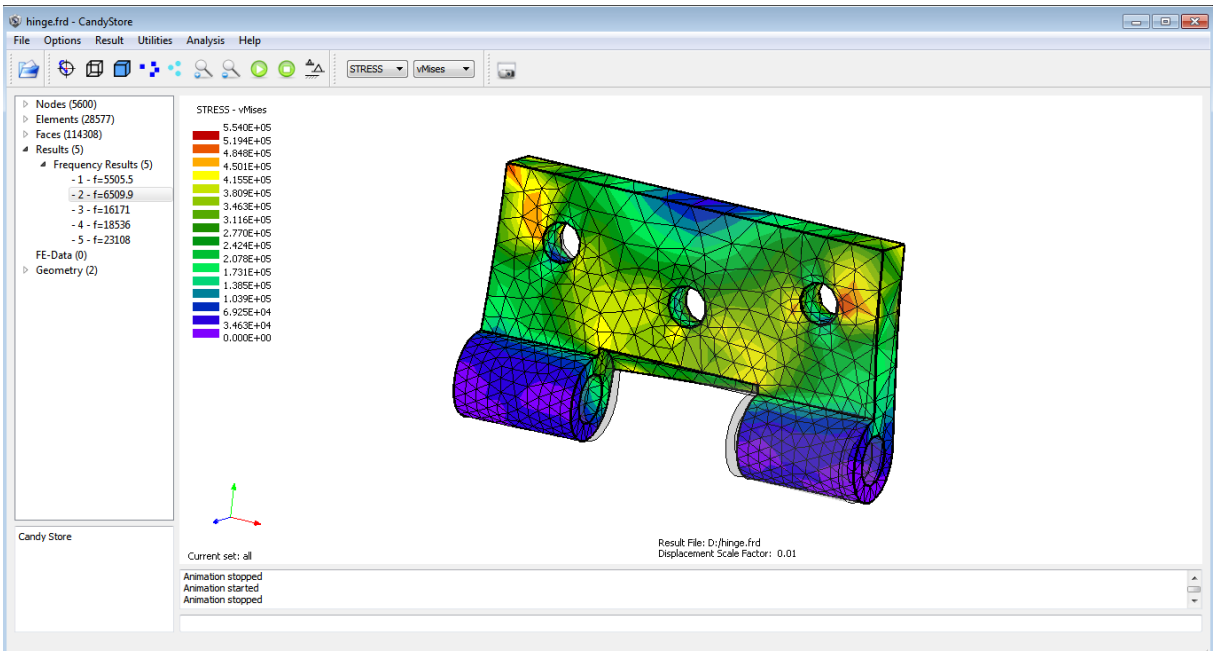

Fig. 11: Displaying results

After a few seconds the results have been loaded and you the displaced state of the model is shown in the graphics view. In the result selector you can select which result should be displayed as fringe plot. To query the values of a specific node you first have to make them visible. If the nodes are visible hold the Shift key down and left click on the node you are interested in. The values will be displayed in the info box in the lower left corner.

To recover the base state of the model click on the root result item.

If you want to take pictures click on the Screenshot item.