# Table of Contents

## Silicon nanowire field-effect transistor

- **Introduction** .................................................. 2
- **Band structure of a Si(100) nanowire** ........ 2
  - Building the nanowire ........................................ 3
- **Setting up and running the calculations** .... 5
  - TB09 meta-GGA c-parameter ................................. 6
  - Running the calculations .................................... 6
  - Analyzing the results ....................................... 6
  - Detailed band structure analysis .......................... 7
- **Si(100) nanowire FET device** ......................... 9
  - FET device configuration ................................... 9
  - Setting up the gate .......................................... 10
  - Build the device ............................................ 10
  - Doping the Si(100) wire .................................... 11
- **Zero gate voltage calculation** .................... 12
  - Analyzing the results ..................................... 13
  - Performing a gate scan ................................... 13
  - Analyzing the gate scan ................................... 14
Silicon nanowire field-effect transistor

Version: 2016.0

Introduction

This tutorial shows you how to set up and perform calculations for a device based on a silicon nanowire. You will define the structure of a hydrogen passivated Si(100) nanowire, and set up a field-effect transistor (FET) structure with a cylindrical wrap-around gate. The tutorial ends with calculations of the device conductance as a function of gate bias.

Note

You will primarily use the graphical user interface QuantumATK for setting up and analyzing the results. If you are not familiar with QuantumATK, please go through the tutorial Introduction to QuantumATK.

The underlying calculation engines for this tutorial are ATK-DFT and ATK-SE. A complete description of all the parameters, and in many cases a longer discussion about their physical relevance, can be found in the ATK Reference Manual.

Band structure of a Si(100) nanowire

First step is to set up the Si(100) nanowire and optimize the geometry. You should use the ATK-DFT calculator for this. We then compute the band structure of the nanowire using 3 different computational models; DFT-GGA, DFT-MGGA, and the Extended Hückel method.
**Building the nanowire**

Open QuantumATK and create a new empty project called “silicon_nanowire”. Then launch the Builder.

- Go to Add ▶ From Database and type “silicon fcc” to locate the diamond phase of silicon.
- Add the Silicon (alpha) bulk configuration to the Stash (double-click or use the 🔄 icon).

In the Builder, use the Builders ▶ Surface (Cleave) tool to create the Si(100) facet:

- keep the default (100) cleave direction, and click Next;
- keep the default surface lattice, and click Next;
- keep the default supercell, which will ensure that the wire direction is perpendicular to the surface, and click Next.
- Click Finish to add the cleaved structure to the Stash.
Next, use the Bulk Tools ‣ Repeat tool to repeat the structure twice along the A and B directions:

Finalize the nanowire by following these steps:

- use the Bulk Tools ‣ Lattice Parameters tool to set the A and B lattice vector lengths to 20 Å;
- use the Coordinate Tools ‣ Center tool to center the structure along all directions;
- go to Coordinate Tools ‣ Custom Passivator and apply the 4(SP3) type hydrogen passivation to the nanowire.
Send the H-passivated Si(100) nanowire to the Script Generator by using the in the lower right-hand corner of the Builder window.

**Setting up and running the calculations**

You should now relax the nanowire and compute the band structure. The DFT-GGA method is used for geometry optimization, while the band structure is computed using DFT-GGA, DFT-MGGA, and the Extended Hückel model.

---

**Important**

The TB09 meta-GGA and Extended Hückel models cannot be used for relaxation.

---

In the Script Generator, change the default output file name to `si_100_nanowire.hdf5`, and add the following blocks to the script:

- New Calculator
- OptimizeGeometry
- Bandstructure
- New Calculator
- Bandstructure
- New Calculator
- Bandstructure

Open the first `New Calculator` block, and make the following changes to the calculator settings:

- set the k-point sampling to 1x1x11;
- change the exchange-correlation potential to GGA.

Open the second `New Calculator` block, and make the following changes:
- set the k-point sampling to 1x1x11;
- change the exchange-correlation potential to MGGA.

Open the third New Calculator block, and make the following changes:

- select the ATK-SE: Extended Hückel calculator;
- uncheck “No SCF iteration” to make the calculation selfconsistent;
- set the k-point sampling to 1x1x11;
- increase the density mesh cut-off to 20 Hartree;
- go to the Hückel basis set tab, and select the “Hoffmann.Hydrogen” and “Cerda.Silicon [GW diamond]” basis sets. The latter has been fitted to GW calculations, and gives an excellent description of the silicon bandstructure, including the value of the band gap.

Finally, open each Bandstructure script block and set the number of points per segment to 100.

**Tip**

The default value for the density mesh cut-off is often sufficient, but by increasing it you may sometimes obtain a slightly higher accuracy at a small additional computational cost.

**TB09 meta-GGA c-parameter**

You should also adjust the value of the c-parameter used in the TB09 meta-GGA method. This is essentially a fitting parameter that can be used to adjust the calculated band gap of the material. If the c-parameter is not defined by the user, a value is automatically calculated from the electronic structure of the material. However, this will not work for a system with vacuum regions, like a nanowire.

You will therefore use a fixed value, c=1.0364, which has been fitted to yield an accurate prediction of the fundamental band gap in silicon (1.13 eV).

Transfer the QuantumATK Python script to the Editor using the icon. Then locate the script line defining the variable `exchange_correlation` for the MGGA exchange-correlation, and set the fixed value of c to 1.0364, as illustrated below.

```python
#----------------------------------------
# Exchange-Correlation
#----------------------------------------
exchange_correlation = MGGA.TB09LDA(c=1.0364)
```

**Running the calculations**

Save the QuantumATK Python script as `si_100_nanowire.py` and execute it using the Job Manager. It will take just a few minutes to complete the calculations.

**Analyzing the results**

Once the calculations have finished, the HDF5 data file `si_100_nanowire.hdf5` should appear on the QuantumATK LabFloor. Locate the Bandstructure items inside this file. The order of the item IDs corresponds to the order in which the band structures were computed and saved, i.e. GGA, MGGA, and Hückel.

Select one of the band structures and open the Bandstructure Analyzer in the right-hand Panel Bar to inspect the computed band structure. You can do this for all three band structures, or use the Compare Data plugin to plot a direct comparison of the three band structures.
Detailed band structure analysis

In order to compare the band gaps obtained with the three computational models, it is convenient to use `bandstructure_analyzer.py`, which uses QuantumATK Python for detailed analysis of the gaps in a band structure. The script is shown below:

```python
from QuantumATK import *
import pylab

# Custom analyzer for calculating the band gap of a bandstructure

def fitEnergyMinimum (i_min, energies, k_points, nfit=3):
    """
    Function for fitting the energy minimum located around i_min.
    @param i_min   : approximate position of the energy minimum.
    @param energies: list of energies.
    @param k_points: list of k_points which correspond to the energies.
    @param nfit    : order of the polynomial.
    @return d2_e, e_min, k_min : second derivative of energy,
                                minimum energy, minimum k_point.
    """

    #list of energies
    n = len(energies)
    efit = numpy.array([energies[(n+i_min-nfit/2+i)%n] for i in range(nfit)])
    kfit = numpy.array([i-nfit/2 for i in range(nfit)])

    #special cases
    if i_min == 0:  # assume bandstructure symmetric around zero
        for i in range(nfit/2):
            efit[i] = energies[nfit/2-i]
    if i_min == n-1:  # assume bandstructure symmetric around end point
        for i in range(nfit/2+1, nfit):
            efit[i] = energies[n-1+nfit/2-i]

    #make fit
    p = numpy.polyfit(kfit,efit,2)
    i_fit_min = -p[1]/2./p[0]
    pf = numpy.poly1d(p)
    e_min = pf(i_fit_min)
    i0 = int(i_fit_min+n_min+n)
    w = i_fit_min+i_min+n-i0
    k_min = (1-w)*k_points[i0+n]+w*k_points[(i0+1+n)]

    return p[0], e_min, k_min

def analyseBandstructure(bandstructure, spin):
    """
    Function for analysing a band structure and calculating bandgaps.
    @param bandstructure : The bandstructure to analyze
    @param spin          : Which spin to select from the bandstructure.
    @return e_val, e_con, e_gap : maximum valence band energy,
                                  minimum conduction band energy,
                                  and direct band gap.
    """

    energies = bandstructure.evaluate(spin=spin).inUnitsOf(eV)

    #some dummy variable to help finding the extrema
    e_valence_max = -1.e10
    e_conduction_min = 1.e10
    e_gap_min = 1.e10
    i_valence_max = 0
    i_conduction_min = 0
    i_gap_min = 0
    n_valence_max = 0
```
```
# Locate extrema
for i in range(energies.shape[0]):
    n = 0
    while n < energies.shape[1] and energies[i][n] < 0.8:
        n += 1
    # find maximum of valence band
    if energies[i][n-1] > e_valence_max:
        e_valence_max = energies[i][n-1]
        i_valence_max = i

# find minimum of conduction band
if energies[i][n] < e_conduction_min:
    e_conduction_min = energies[i][n]
    i_conduction_min = i

# find minimum band gap
if energies[i][n]-energies[i][n-1] < e_gap_min:
    e_gap_min = energies[i][n]-energies[i][n-1]
    i_gap_min = i

# Print out results
a_val, e_val, k_val = fitEnergyMinimum(i_valence_max,
                                      energies[:,:,,n_valence_max],
                                      bandstructure.kpoints())
print('Valence band maximum %7.4f eV at [%6.4f, %6.4f, %6.4f]  \
       %e_val, k_val[0], k_val[1], k_val[2])

a_con, e_con, k_con = fitEnergyMinimum(i_conduction_min,
                                       energies[:,:,,n_conduction_min],
                                       bandstructure.kpoints())
print('Conduction band minimum %7.4f eV at [%6.4f, %6.4f, %6.4f]  \
      %e_con, k_con[0], k_con[1], k_con[2])

print('Fundamental band gap %7.4f eV  % (e_con-e_val)')
a_gap, e_gap, k_gap = fitEnergyMinimum(i_gap_min,
                                       energies[:,:,,n_gap_min]-energies[:,:,,n_gap_min],
                                       bandstructure.kpoints())
print('Direct band gap %7.4f eV at [%6.4f, %6.4f,%6.4f]  \
      %e_gap, k_gap[0], k_gap[1], k_gap[2])
return e_val, e_con, e_gap

def analyzer(filename, **args):
    """
    Find band gaps of band structures in netcdf file.
    """
    if filename == None:
        return

    #read in the bandstructure you would like to analyze
    bandstructure_list = nlread(filename, Bandstructure)
    if len(bandstructure_list) == 0:
        print('No Bandstructures in file ', filename)
        return

    for s in [Spin.All]:
        b_list = []
        n = 0
        for b in bandstructure_list:
            b_list.append(b)
            n += 1
            if n % 1000 == 0:
                print(' %d out of %d bandstructures read' % (n, len(bandstructure_list)))
    return b_list
```
Download the script (link: bandstructure_analyzer.py), and execute it using the Job Manager. You should get the following output:

<table>
<thead>
<tr>
<th>Analysis</th>
<th>Valence band maximum</th>
<th>Conduction band minimum</th>
<th>Fundamental band gap</th>
<th>Direct band gap</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>-1.6395 eV</td>
<td>1.6400 eV</td>
<td>3.2795 eV</td>
<td>3.2799 eV</td>
</tr>
<tr>
<td>1</td>
<td>-1.8434 eV</td>
<td>1.8450 eV</td>
<td>3.6884 eV</td>
<td>3.6914 eV</td>
</tr>
<tr>
<td>2</td>
<td>-1.7737 eV</td>
<td>1.7735 eV</td>
<td>3.5472 eV</td>
<td>3.5472 eV</td>
</tr>
</tbody>
</table>

The output shown above gives the band structure analysis for the DFT-GGA, DFT-MGGA, and Extended Hückel models. The script gives the valence band maximum, conduction band minimum, fundamental band gap, and direct band gap from the valence band maximum. The fundamental band gap for the Si(100) nanowire is significantly larger than that of bulk silicon, and all three computational models place it at the Gamma point or close to it.

Furthermore, the TB09-MGGA model gives a gap which is about 0.5 eV larger than the GGA model, which is in accordance with the difference in the band gaps of bulk Si calculated with the GGA and TB09-MGGA methods. The Extended Hückel model is in agreement with the TB09-MGGA method.

**Si(100) nanowire FET device**

You will now consider a Si(100) nanowire field-effect transistor with a cylindrical wrap-around gate. You should first build the device configuration, including the gate and doping of the silicon nanowire, and then use the Extended Hückel model to compute device properties such as zero-bias transmission and conductance vs. gate voltage.

**FET device configuration**

You should use the relaxed nanowire configuration as a starting point for building the device. The contents of the file `si_100_nanowire.hdf5` should be available on the QuantumATK LabFloor. It contains four bulk configurations: The first one (giD000) is the unrelaxed structure, while the remaining ones
correspond to the relaxed structure, calculated with different methods (DFT-GGA, DFT-MGGA and Extended Hückel).

Send the configuration with ID \textit{gID001} to the Builder. It will appear in the Builder Stash. Then use the Bulk Tools \textit{\textbf{\texttimes  Repeat}} tool to repeat the configuration 12 times along C.

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{fig1.png}
\caption{Builder interface showing the configuration and repeat settings.}
\end{figure}

\textbf{Setting up the gate}

Next, use the Miscellaneus \textit{\textbf{\texttimes  Spatial regions}} tool to define the metallic wrap-around gate:

- First, add a new metallic region with a value of 0 Volt.
- Under \textit{Geometry}, select \textit{Tube} to create a cylindrical region.
- Define the geometry of the tube by entering the parameters shown in the image below. The cylinder will extend to the edge of the simulation cell along A and B, and will cover most of the central part of the nanowire along C.

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{fig2.png}
\caption{Spatial regions tool interface showing the parameters and geometry settings.}
\end{figure}

\textbf{Build the device}

Use the Device Tools \textit{\textbf{\texttimes  Device From Bulk}} tool to build the Si(100) nanowire FET device from the bulk nanowire configuration. Use the default suggestion for the electrode lengths.
Then send the device configuration to the Script Generator.

**Doping the Si(100) wire**

You will now introduce doping in the nanowire. Instead of explicitly adding dopant atoms, which would result in a very high doping concentration for this relatively small device, you will set up the system as a p-i-n junction, by adding certain amounts of charge to the electrodes. This will lead to shifts in the Fermi levels of the two electrodes, resulting in a built-in potential in the device.

Use the mouse to select all atoms in the left electrode and open the Miscellaneous • Doping tool. In the window that shows up, choose the following settings:

- **Doping Type**: p-type
- **Value**: $4 \times 10^{19}$ e/cm
- **Unit**: e/cm$^3$ (left electrode)

Next, close the Doping window and select all atoms in the right electrode. Open again the Doping tool, and choose the following settings:

- **Doping Type**: n-type
- **Value**: $4 \times 10^{19}$ e/cm
- **Unit**: e/cm$^3$ (right electrode)
You will now calculate the transmission spectrum of the p-i-n doped Si(100) nanowire FET device at zero gate potential using the Extended Hückel model. In the **Script Generator**, change the default output file name to `si_100_nanowire_fet_pin.hdf5` and add the following blocks to the script:

- **New Calculator**
- **TransmissionSpectrum**
- **ElectronDifferenceDensity**
- **ElectrostaticDifferencePotential**

Open the **New Calculator** block, select the Extended Hückel device calculator, and set up the calculator parameters similarly to what you did above for the bulk nanowire calculations:
Uncheck "No SCF iteration" to make the calculation selfconsistent.
Increase the density mesh cut-off to 20 Hartree.
Go to the Hückel basis set tab and select the "Hoffmann.Hydrogen" and "Cerda.Silicon [GW diamond]" basis sets.
Under Poisson solver, use Neumann boundary conditions in the A and B directions.

In the Transmission Spectrum block, set the energy range to -4 to +4 eV with 301 points.

Save the QuantumATK Python script as `si_100_nanowire_fet_pin.py` and execute it using the Job Manager. The calculation will take about 30 minutes if executed in parallel with 4 MPI processes.

It would be physically incorrect to use periodic boundary conditions along the A and B directions when there are gates present in the system. We therefore choose Neumann boundary conditions along A and B instead.

Analyzing the results

Once the calculation is completed, locate the file `si_100_nanowire_fet_pin.hdf5` HDF5 file on the Labfloor and unfold it. Select the Transmission Spectrum item and use the Transmission Analyzer to plot the transmission spectrum.

The transmission spectrum is shown above. The transmission is zero inside the nanowire band gap, which appears to be roughly 7 eV, significantly larger than computed earlier. This is due to the p- and n-doping, which has moved the valence and conduction band edges with respect to the Fermi level.

Performing a gate scan

Finally, you will calculate the transmission spectrum for different values of the gate bias. You can then plot the device conductance as function of gate bias.
The calculations are most conveniently done using QuantumATK Python scripting. Use the script `nanodevice_gatescan.py` for this. Download it to the active QuantumATK project folder, which also contains the file `si_100_nanowire_fet_pin.hdf5`. The script is reproduced below.

```python
# Read in the old configuration
device_configuration = nlread("si_100_nanowire_fet_pin.nc", DeviceConfiguration)[0]
calculator = device_configurationcalculator()
metallic_regions = device_configuration.metallicRegions()

# Define gate voltages
gate_voltage_list=[-1, 0, 1.0, 2.0, 3.0, 4.0, 5.0, 6, 7, 8]*Volt

# Define output file name
filename = "nanodevice_huckel.nc"

# Perform loop over gate voltages
for gate_voltage in gate_voltage_list:
    # Set the gate voltages to the new values
    new_regions = [m(value = gate_voltage) for m in metallic_regions]
device_configuration.setMetallicRegions(new_regions)

    # Make a copy of the calculator and attach it to the configuration
    # Restart from the previous scf state
    device_configuration.setCalculator(calculator(),
        initial_state=device_configuration)
device_configuration.update()
nlsave(filename, device_configuration)

    # Calculate analysis objects
    electron_density = ElectronDifferenceDensity(device_configuration)
nlsave(filename, electron_density)

    electrostatic_potential = ElectrostaticDifferencePotential(device_configuration)
nlsave(filename, electrostatic_potential)

    transmission_spectrum = TransmissionSpectrum(
        configuration=device_configuration,
        energies=numpy.linspace(-4, 4, 301)*eV,
        kpoints=MonkhorstPackGrid(1,1),
        energy_zero_parameter=AverageFermiLevel,
        infinitesimal=1e-06*eV,
    )
nlsave(filename, transmission_spectrum)

    nlprint(transmission_spectrum)
```

Execute the script using the 🔄 Job Manager or from a command line. The script consists of 8 different calculations and it will take about 5 hours running in parallel with 4 MPI processes. You can, however, decide to reduce this time by reducing the number of gate voltages that the transmission spectrum is computed for – simply edit the variable `gate_voltage_list`.

**Analyzing the gate scan**

Download and execute the script 🛠 conductance_plot.py, which reads the computed transmission spectra, calculates the conductance for each, and plots the conductance against gate bias.
# Read the data
transmission_spectrum_list = nlread("nanodevice_huckel.hdf5", TransmissionSpectrum)
configuration_list = nlread("nanodevice_huckel.hdf5", DeviceConfiguration)

conductance = numpy.zeros(len(configuration_list))
gate_bias = numpy.zeros(len(configuration_list))
for i, configuration in enumerate(configuration_list):
  transmission_spectrum = transmission_spectrum_list[i]
  energies = transmission_spectrum.energies().inUnitsOf(eV)
  spectrum = transmission_spectrum.evaluate()
  gate_bias[i] = configuration.metallicRegions()[0].value().inUnitsOf(Volt)
  conductance[i] = transmission_spectrum.conductance()

# Sort the data according to the gate bias
index_list = numpy.argsort(gate_bias)

# Plot the spectra
import pylab
pylab.figure()
ax = pylab.subplot(111)
ax.semilogy(gate_bias[index_list], conductance[index_list])
ax.set_ylabel("Conductance (S)", size=16)
ax.set_xlabel("Gate Bias (Volt)", size=16)
ax.set_ybound(lower=1e-26)
for g, c in zip(gate_bias[index_list], conductance[index_list]):
    print(g, c)
pylab.show()