
Quantum Computational Physics

Set up / Installation Guide

Summer Term 2026

Due date: No hand-in

Discussion: Tuesday, 14.04.2026 (if needed)

Website: thp.uni-koeln.de/trebst/Lectures/2026-QuantCompPhys.shtml

Setting up IBM Quantum

In the following exercises, we want to make use of the rich quantum computing ecosystem provided by IBM. That includes using the Qiskit framework to simulate quantum circuits and running them on real quantum devices (as students, we get 10 minutes of quantum time per month for free). To do so, we need to (i) create an account on the IBM quantum website and (ii) install the Qiskit framework on our local machine. This exercise will guide you through these steps.

Ideally, everyone should have done this before the first exercise class. If there are any problems, we can work on these together in the first session.

(i) Setting up an IBM account

Let us start by creating an IBM account. This is pretty straightforward and should not take more than a few minutes.

- a)** To make use of the free quantum computing time, we need to use our university email address (`user@smail.uni-koeln.de`) to create an account on the IBM quantum website (<https://quantum.cloud.ibm.com>). Click **Sign in** in the top right hand corner. Then click **Create an IBM Cloud account**. Use your student email address. For **What is your role?** select **Student**, and provide **University of Cologne** as your Company name. Accept the terms and conditions and click **Create account**.
- b)** When your account has been created, you will be redirected to the IBM Quantum Platform dashboard. Agree to the license agreement and click **Create instance**.

Now, we have successfully created an IBM account and can already submit jobs to real quantum computers via IBM's cloud service! However, designing and manipulating quantum circuits can get quite complex in the web-based interface. Therefore, we want to use the Qiskit framework to write our code locally and then submit it to the IBM quantum devices.

(ii) Installing Qiskit

We will give a concise guide on how to install Qiskit on your local machine. For more detailed information, you can always refer to the official Qiskit documentation (<https://docs.quantum.ibm.com/guides/install-qiskit#local>). The commands below (bash) are for a Unix-based system.

If you are using Windows, we recommend that you use the Windows Subsystem for Linux (<https://learn.microsoft.com/en-us/windows/wsl/install>) to run the commands.

- a) First, let's make sure that a supported python version is installed. The latest version of Qiskit (2.3.1) supports python 3.10 or newer. Check which version of python you have installed by running

```
bash
python3 --version
```

in your terminal and update to a supported version if necessary.

To separate the Qiskit installation - as well as other packages we will use in this tutorial - from your other python projects, it is recommended to create a virtual environment. This can be done by running (replace `/path/to/virtual/environment` with the path where you want to create the virtual environment):

```
bash
python3 -m venv /path/to/virtual/environment
```

This will create a new directory at the specified path containing a python interpreter and a pip package manager. To activate the virtual environment, run:

```
bash
source /path/to/virtual/environment/bin/activate
```

You can deactivate the virtual environment at any time by running:

```
bash
deactivate
```

- b) Now that we have our virtual environment set up, we can install Qiskit (with visualization capabilities) by running

```
bash
pip install 'qiskit[visualization]'
```

inside our virtual environment. Additionally, we want to install the `qiskit-ibm-runtime` package, which is needed to access the IBM Quantum devices by running:

```
bash
pip install qiskit-ibm-runtime
```

To efficiently simulate quantum circuits on our local machine, we want to install the Aer package by running:

```
bash
```

```
pip install qiskit-aer
```

Furthermore, we want to use Jupyter Notebooks to write our code. To install jupyter notebooks, run

```
bash
```

```
pip install jupyter
```

We can start a Jupyter Notebook server by running

```
bash
```

```
jupyter notebook
```

in the terminal. This will open a new tab in the browser where we can create a new notebook and start writing code. While this is totally sufficient, we recommend using Jupyter Notebooks inside your favorite IDE. For example, you can use the Jupyter Notebook extension in Visual Studio Code.

- c) Now all that is left is to link your IBM Quantum account to Qiskit. This can be done by running

```
bash
```

```
python3 -c "from qiskit_ibm_runtime import QiskitRuntimeService;
QiskitRuntimeService.save_account(
    channel='ibm_quantum_platform',
    token='<MY_IBM_QUANTUM_API_TOKEN>',
    set_as_default=True
)"
```

in your terminal, where you replace `<MY_IBM_QUANTUM_API_TOKEN>` with your IBM Quantum token. You can find your token in the top left hand corner on the [IBM quantum website](#) after logging in.

Awesome! Now we have successfully set up our IBM account and installed Qiskit on our local machine. We are ready to start writing quantum circuits and running them on real quantum devices. In the next exercises, we will start putting together some simple quantum circuits using Qiskit. If you encounter any problems with the installation, please let us know in the first exercise class.