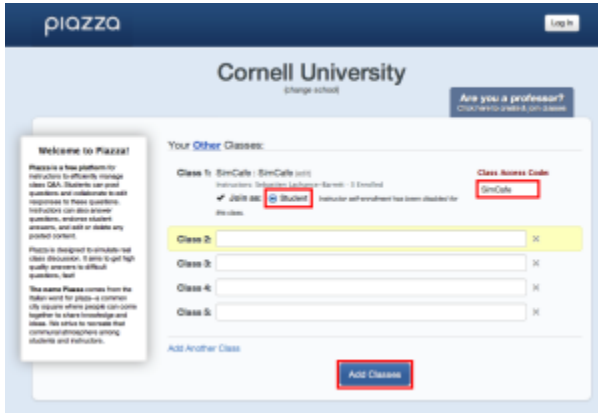


# Piazza Sign-Up Tutorial

## Piazza Sign-Up Tutorial and Directions

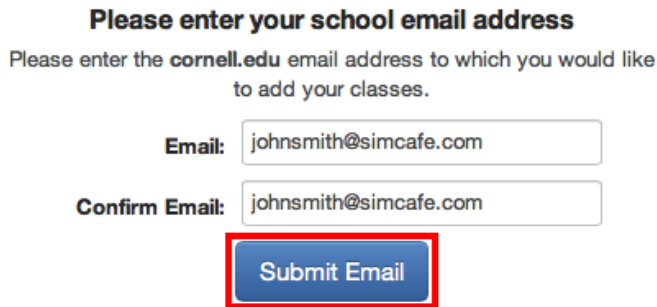
Step 1: Click on this link: [piazza.com/cornell/fall2019/0000](https://piazza.com/cornell/fall2019/0000)

Step 2: Join as a "Student", enter **SimCafe** for the class access code and click on "Add Class"

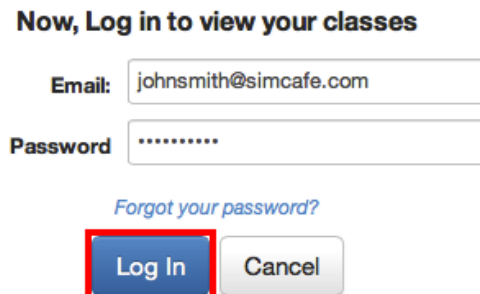


(Click on the picture for higher resolution)

Step 3: Enter your email address. Note that this will work even if you do not have a cornell.edu email address.



Step 4: If you already have a Piazza account: Simply log in.



If this is your first time using Piazza: Retrieve the validation code sent to you by email, set-up a password for your account and enter your full name (note that Piazza can let you post anonymously).

**Check your inbox** for your confirmation email. **Enter the validation code** below so you can access your classes!

\_\_\_\_\_

Submit Code

Please check your bulk mail or spam folder first. [Click here to resend](#) the email. (It may take a few minutes to arrive.)

**Is this your preferred email address?**

○

.....

.....

John Smith

[Create Account](#)

piazza

SIMCAFE ▼

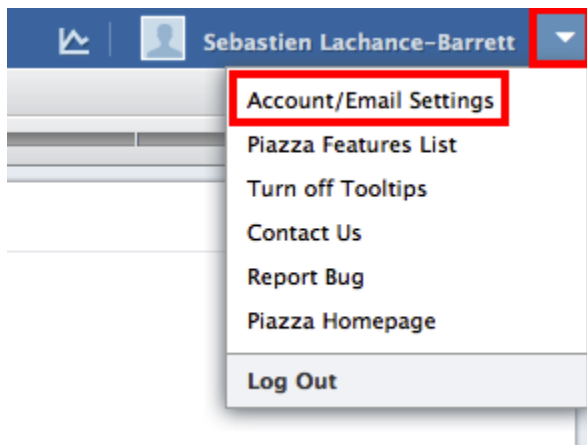
SimCafe SimCafe [Other]

[Show inactive classes](#)

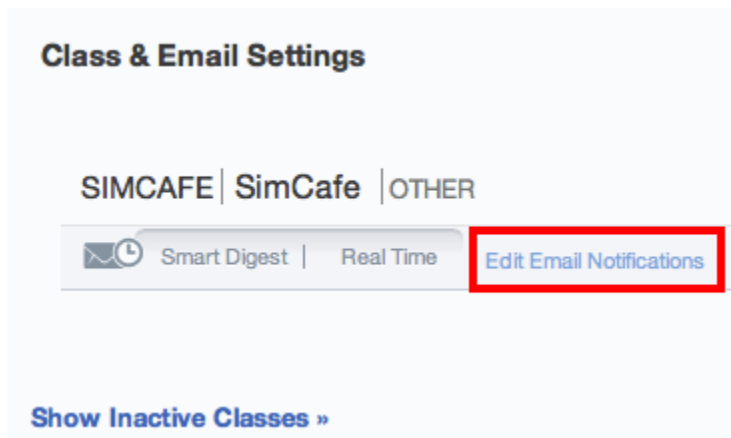
### Create a New Class...

Clone This Class...

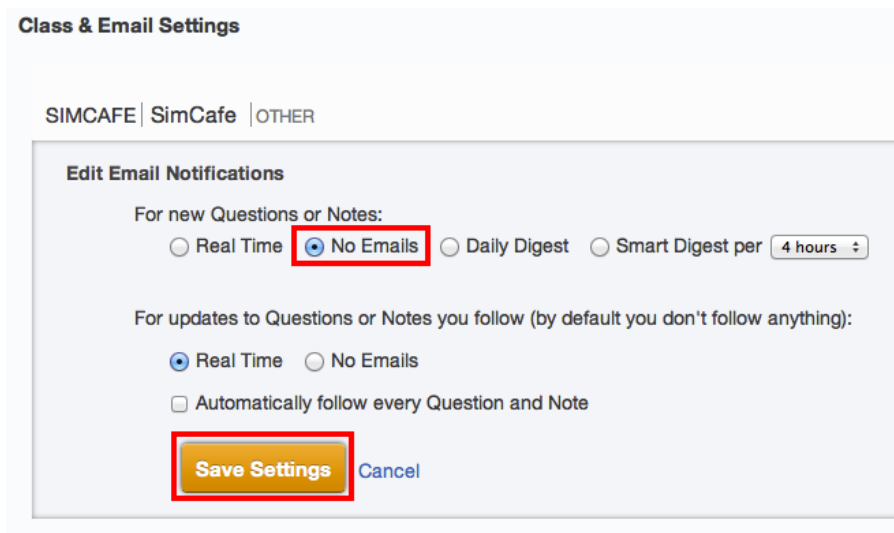
1. Click on the arrow at the top right corner of the page and select "account/email settings".



2. Click on "edit email notifications" under the SimCafe class.



3. Select "No emails" for "new questions or notes" and save your settings.



Note that you will receive real time emails for tutorial discussions that you follow. When you add a follow up discussion for a tutorial, you will automatically follow that discussion and therefore receive emails letting you know of every new comments. If you do not wish to receive any more notifications for a particular tutorial, please unfollow it by clicking on "stop following" as shown on the picture below.



note ☆

stop following

3 views



## FLUENT - Forced Convection

Please compose a new followup discussion for questions or comments about this tutorial. Feel free to help out other people as well!

ansys\_fluent

1 minute ago by Sebastien Lachance-Barrett