



# Micromixer — Batch Version

## Introduction

---

The purpose of this example is to demonstrate how to access the batch functionality in COMSOL from the COMSOL Desktop and use it on your computer. The model also demonstrates how you can measure the speedup of COMSOL on your computer. The speedup is defined as the quotient between the total runtime using only one core of the machine and the runtime using all cores of the machine.

For detailed information about the model, see [Micromixer](#).

---

**Application Library path:** COMSOL\_Multiphysics/Cluster\_and\_Batch\_Tutorials/micromixer\_batch

---

## Modeling Instructions

---

### APPLICATION LIBRARIES

- 1 From the **File** menu, choose **Application Libraries**.
- 2 In the **Application Libraries** window, select **COMSOL Multiphysics>Fluid Dynamics>micromixer** in the tree.
- 3 Click  **Open**.

### STUDY 1

- 1 Click the  **Show More Options** button in the **Model Builder** toolbar.
- 2 In the **Show More Options** dialog box, in the tree, select the check box for the node **Study>Batch and Cluster**. With this setting active, **Batch** is available from the Study node's context menu.
- 3 Click **OK**.

### Batch

- 1 In the **Study** toolbar, click  **Batch** and choose **Batch**.
- 2 In the **Settings** window for **Batch**, locate the **Batch Settings** section.
- 3 Find the **Cluster computing settings** subsection. From the **Settings** list, choose **User controlled**. With this setting, settings specific to the model will be used instead of preference settings.

- 4 Clear the **Synchronize solutions** check box. This setting would synchronize the solution, but we will not use it here.
- 5 In the **Directory** text field specify the directory where to store the model.  
In the **Study Settings** section of the **Settings** window for **Batch**, you specify where to store the model for the batch job:
- 6 Enter a suitable filename in the **Filename** text field, and specify a directory by clicking **Browse** and then making a choice in the **Batch Directory** dialog box or by typing in the directory path directly in the **Directory** text field.

#### *External Process 1*

- 1 In the **Study** toolbar, click  **Compute**.  
In the **General** section, click **Open** to open the file containing the model generated by the batch job associated with this external process in a new COMSOL session.  
In the **Process Status** section, the log shows the total solution time. COMSOL automatically takes advantage of all cores; to measure the speedup, set the number of cores to 1 and run another batch job.
- 2 In the **Model Builder** window, under **Study 1 > Job Configurations** click **Batch 1**.
- 3 In the **Settings** window for **Batch**, locate the **General** section.
- 4 From the **Defined by study step** list, choose **User defined**.
- 5 Select the **Number of cores** check box. Verify that the setting in the associated text field is 1.  
Next, change the filename to create a new External Process node for the run.
- 6 Locate the **Files** section. In the associated text field, enter a name of your choice.
- 7 Click  **Run**.
- 8 In the **Model Builder** window, under **Study 1 > Job Configurations > Batch 1 > Batch Data** right-click **External Process 2** and choose **Attach Job**.

#### **EXTERNAL PROCESS**

- 1 Go to the **External Process** window.  
When the process has finished, compare the total time in the log for the new external process with the previous value. The speedup is equal to the previous value divided by the new value. The speedup is dependent on the mesh size. To improve the numbers, try refining the mesh.

