

# USTREAMSIM

## Users Guide

---

**Version 2.0**

---

SEPTEMBER 2020

---

## Table of Contents

<b>Overview .....</b>	<b>3</b>
<b>STEP 1 – Registration and Login.....</b>	<b>4</b>
<b>STEP 2 - Dashboard .....</b>	<b>4</b>
<b>STEP 3 - New simulation.....</b>	<b>6</b>
<b>STEP 4 - Selecting a model.....</b>	<b>7</b>
<b>STEP 5 – Run a Basement simulation.....</b>	<b>8</b>
<b>Option 1: Run a stand-alone Basement simulation.....</b>	<b>8</b>
<b>Option 2: Run a Basement simulation for calibration .....</b>	<b>10</b>
<b>Option 3: Run a Basement simulation with scenarios.....</b>	<b>16</b>
<b>STEP 6 – Run a SWMM simulation.....</b>	<b>22</b>
<b>Option 1: Run a stand-alone SWMM simulation.....</b>	<b>22</b>
<b>Acknowledgements .....</b>	<b>26</b>

---

## Overview

USTREAMSIM: integrated modelling in urban systems and streamflow simulation in rivers (1D/2D)

This manual presents the USTREAMSIM service, available for public use at the Infraestrutura Nacional de Computação Distribuída (INCD). This service allows for an integrated 1D / 2D modeling using the combined use of the [Basement](#) and [SWMM](#) (Storm Water Management Model) models for modeling runoff and drainage systems. Basement is a freeware simulation tool for hydro- and morphodynamic modelling. It's a 2D tool for numerical modeling of environmental flow and natural hazard events. SWMM is a 1D tool that helps predict runoff quantity and quality from drainage systems.

In this first version, it's only possible to work with the Basement model. SWMM and the Basement and SWMM coupled will be available in later versions.

The service is accessible at <https://ustreamsim.ncg.ingrid.pt> supported by the computational resources from INCD - Infraestrutura Nacional de Computação Distribuída (<https://www.incd.pt>), a node of the EGI - European Grid Infrastructure (<https://www.egi.eu>).

Access to the service is granted through a registration procedure, by sending an email to [aoliveira@lnec.pt](mailto:aoliveira@lnec.pt), with user name and affiliation. After acceptance of the registration, an email with credentials will be sent to the user. USTREAMSIM is developed and maintained by LNEC - Laboratório Nacional de Engenharia Civil (LNEC - <http://www.lnec.pt>) and LIP - Laboratório de Instrumentação e Física Experimental de Partículas (LIP - <https://www.lip.pt>). This manual presents a detailed guide to help users use the USTREAMSIM service, illustrating its use step by step.

## STEP 1 – Registration and Login

To obtain access credentials, please send an email to [aoliveira@lnec.pt](mailto:aoliveira@lnec.pt) and wait for the approval to get your credentials.

Go to <https://ustreamsim.ncg.ingrid.pt> and login with your username and password.



Figure 1 - Preview of USTREAMSIM login page

## STEP 2 - Dashboard

After the first login the dashboard will be empty.

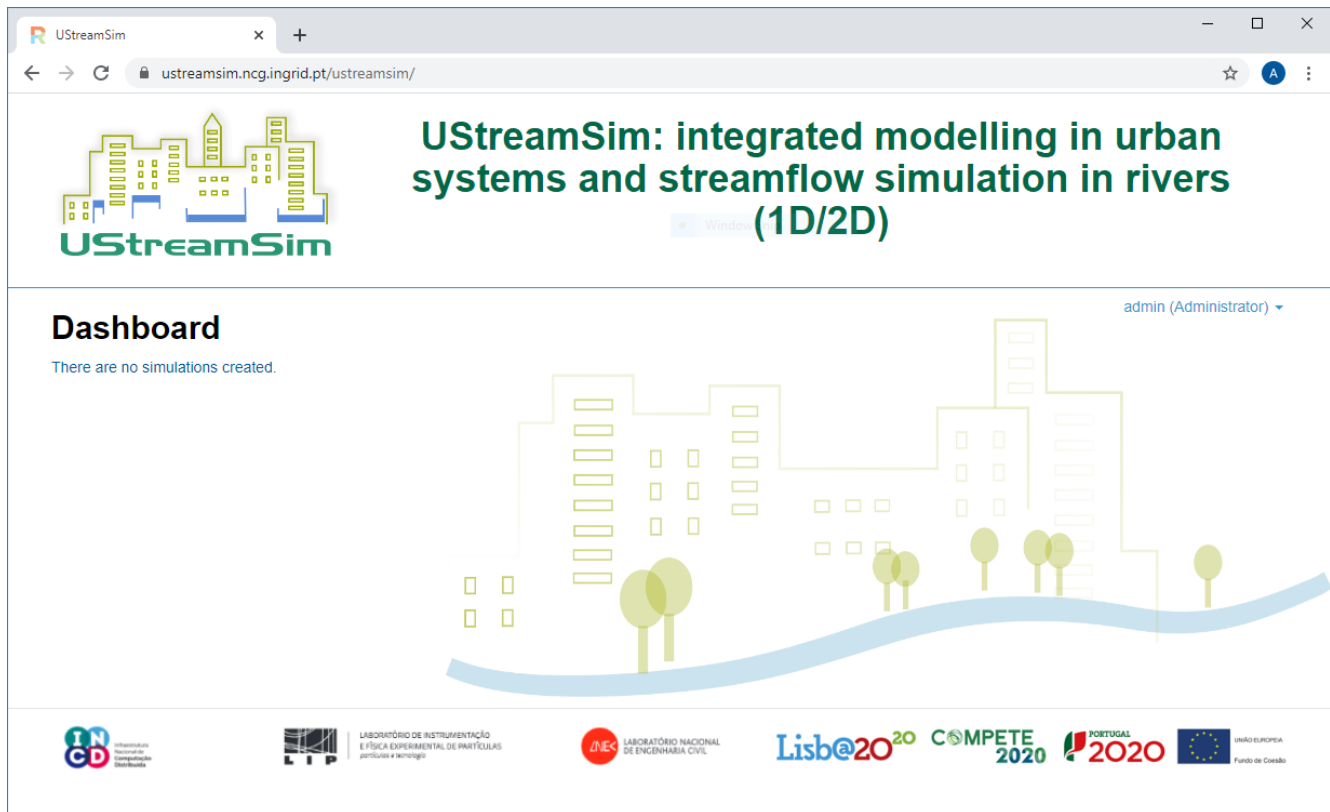


Figure 2 - Preview of the empty dashboard

When new simulations are created the dashboard will list all available simulations. The user can only see his/her own simulations. The list shows simulation name, creation date, status and a download link to get simulation outputs. The status can be Submitting, Running, Success or Error. The Submitting status exists for a short period of time when the simulation is prepared to be send to the remote server. After submission, the simulation status changes to Running meaning that the model is running on the server. When the simulation ends, the status changes to Success if everything went well or Error if something didn't go well. When the user clicks the Download link, he/she gets a Zip file with all the files used in the simulation including model's input, log, error and result files. The user can also delete completed simulations.

To facilitate access to each simulation results, the dashboard presents a list of 10 simulations at a time, with a pagination mechanism to navigate between lists whenever necessary.

**UStreamSim: integrated modelling in urban systems and streamflow simulation in rivers (1D/2D)**

admin (Administrator) ▾

### Dashboard

Name	Date submitted	State	Results data	Action
My_eighth_simulation	May 14, 2020, 2:20 p.m.	Success	Download	
My_seventh_simulation	May 14, 2020, 12:27 p.m.	Success	Download	
My_sixth_simulation	May 14, 2020, 12:11 p.m.	Success	Download	
My_fifth_simulation	May 14, 2020, 11:18 a.m.	Success	Download	
My_fourth_simulation	May 13, 2020, 4:08 p.m.	Success	Download	
My_third_simulation_70	May 12, 2020, 6:22 p.m.	Success	Download	
My_third_simulation_65	May 12, 2020, 6:22 p.m.	Success	Download	
My_third_simulation_60	May 12, 2020, 6:22 p.m.	Success	Download	
My_third_simulation_55	May 12, 2020, 6:22 p.m.	Success	Download	
My_third_simulation_50	May 12, 2020, 6:22 p.m.	Success	Download	

« 1 2 »

INSTITUTO NACIONAL DE INGENHARIA CIVIL  
 LABORATÓRIO DE INSTRUMENTAÇÃO E FÍSICA EXPERIMENTAL DE PARTÍCULAS, POLÍMEROS E TECNOLOGIA  
 LABORATÓRIO NACIONAL DE ENGENHARIA CIVIL  
 LISBOA 2020  
 COMPETE 2020  
 PORTUGAL 2020  
 UNIÃO EUROPEIA - Fundo de Coesão

Figure 3 - Preview the dashboard with pagination

## STEP 3 - New simulation

To create a new simulation, the user must open the menu below the user name and choose *New Simulation*

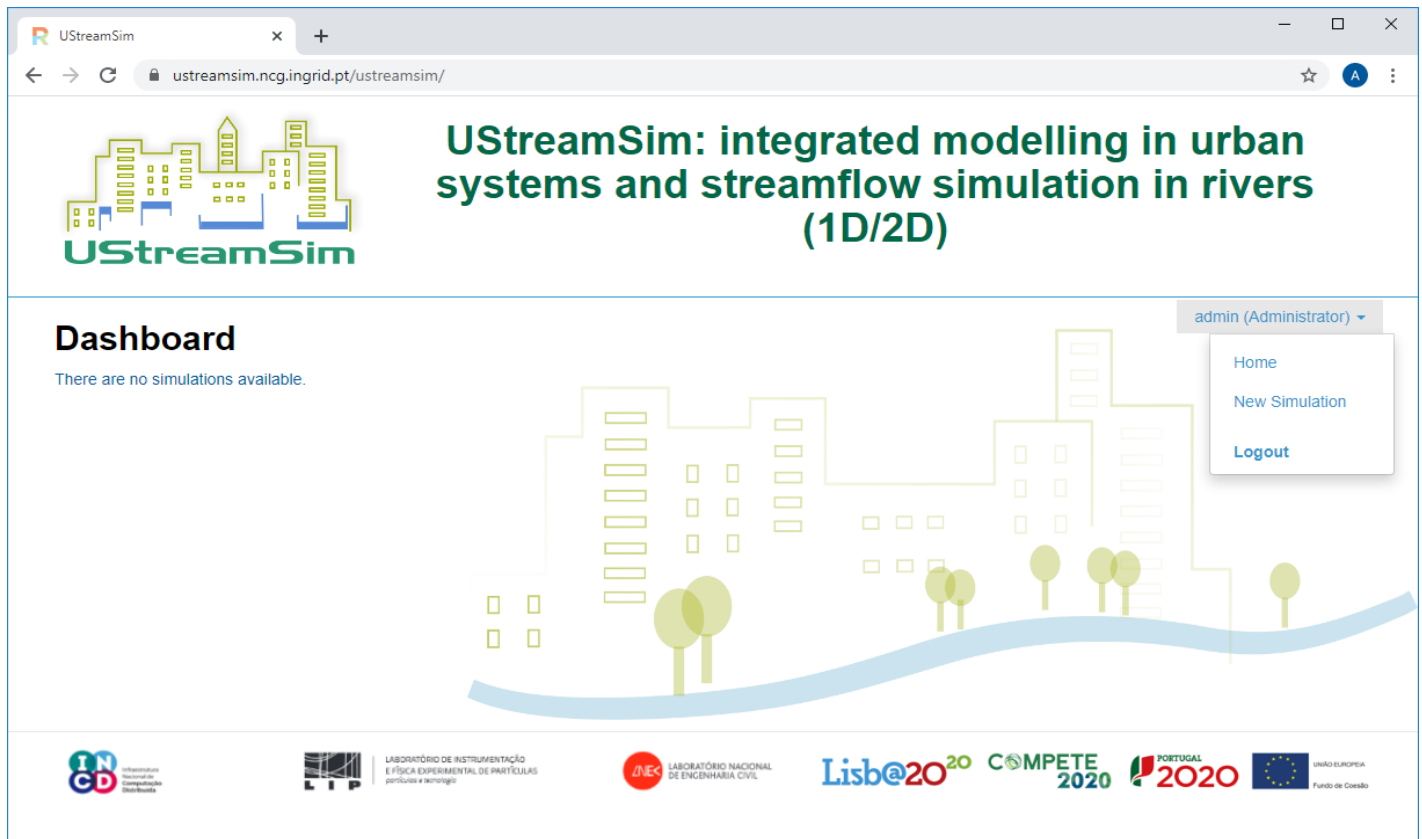


Figure 4 - Preview of New Simulation menu

After that, a sequence of steps is necessary to setup the simulation conditions.

## STEP 4 - Selecting a model

In this step the user will choose the model to use in the simulation. The user can choose between model Basement and model SWMM. For now, calibration and scenarios simulations are not available with the model SWMM.

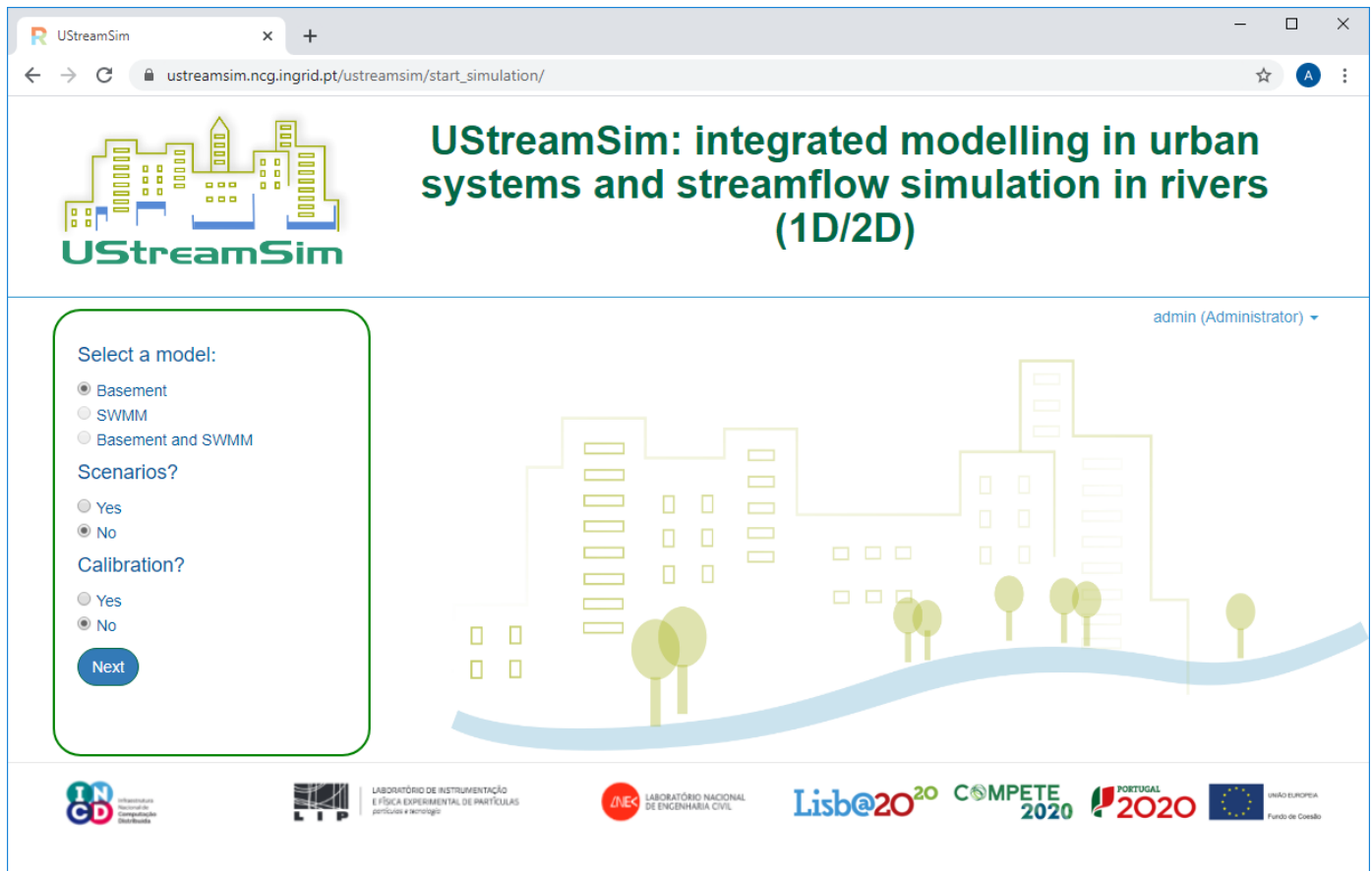


Figure 5 - Preview of choosing a model

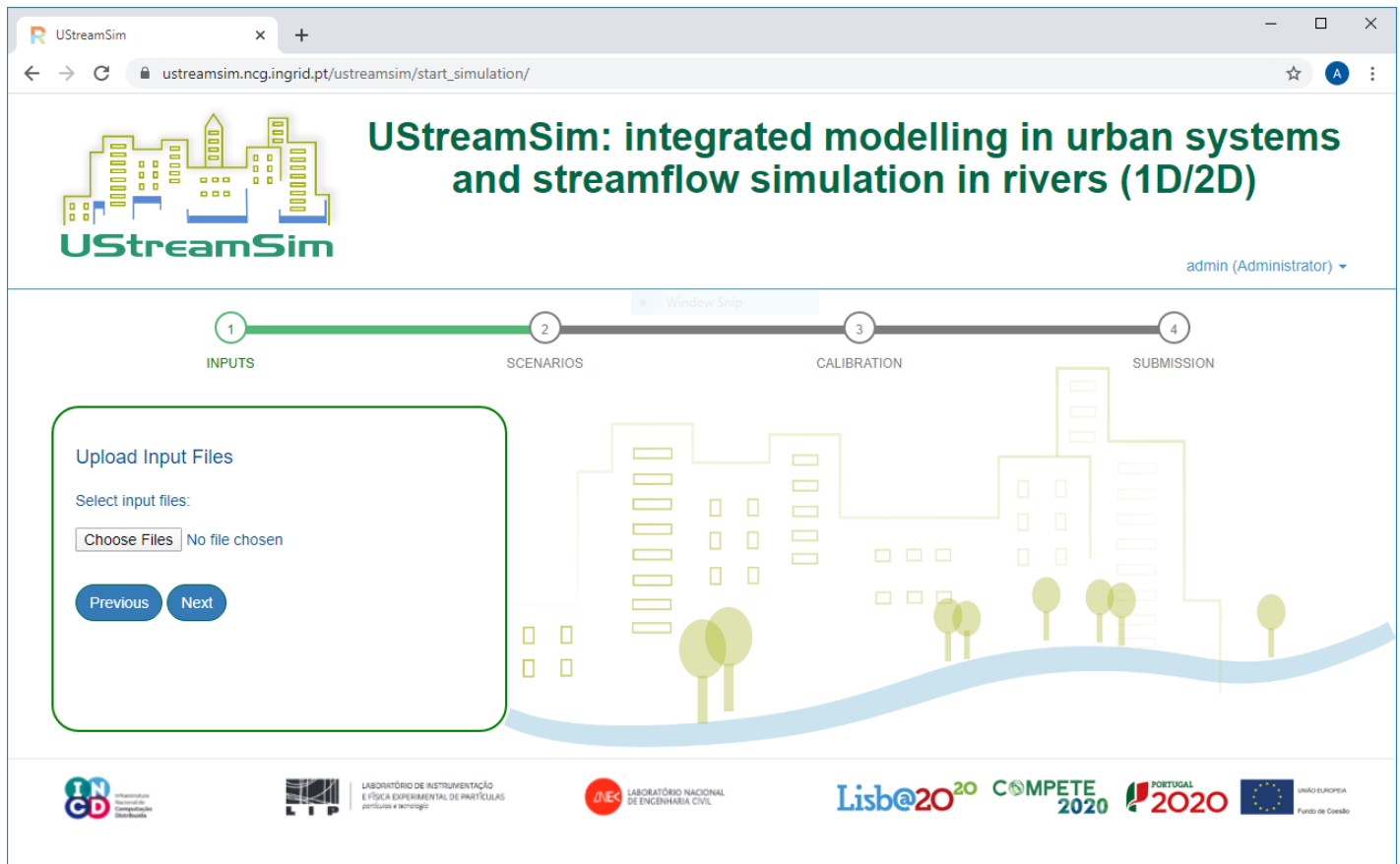
## STEP 5 – Run a Basement simulation

After choosing the Basement model, the user can decide either to run a single simulation (option 1), run multiple simulations for calibration parameter values purposes (option 2) or run multiple simulations using different scenarios (option 3).

### Option 1: Run a stand-alone Basement simulation

The user chooses the Basement model and “No” option for Scenarios and Calibration. After Clicking Next, a new menu is shown (Figure 6).





**Figure 6 - Preview of Upload Input files menu**

In this new window, the user can choose the files he/she wants to use in the simulation. For model Basement, these input files include command base file (.bmc), a mesh file (.2dm), boundary condition files (.txt) or a restart file. Click Next.

The user must then provide a name for the new simulation. This identification will show up in the simulations table (Figure 7).

The user can go back if the files uploaded are not correct. After checking if the input files are correct, if the user wants to make changes, click Previous and move backwards in the steps. Otherwise, just click Start Simulation to initiate the simulation.

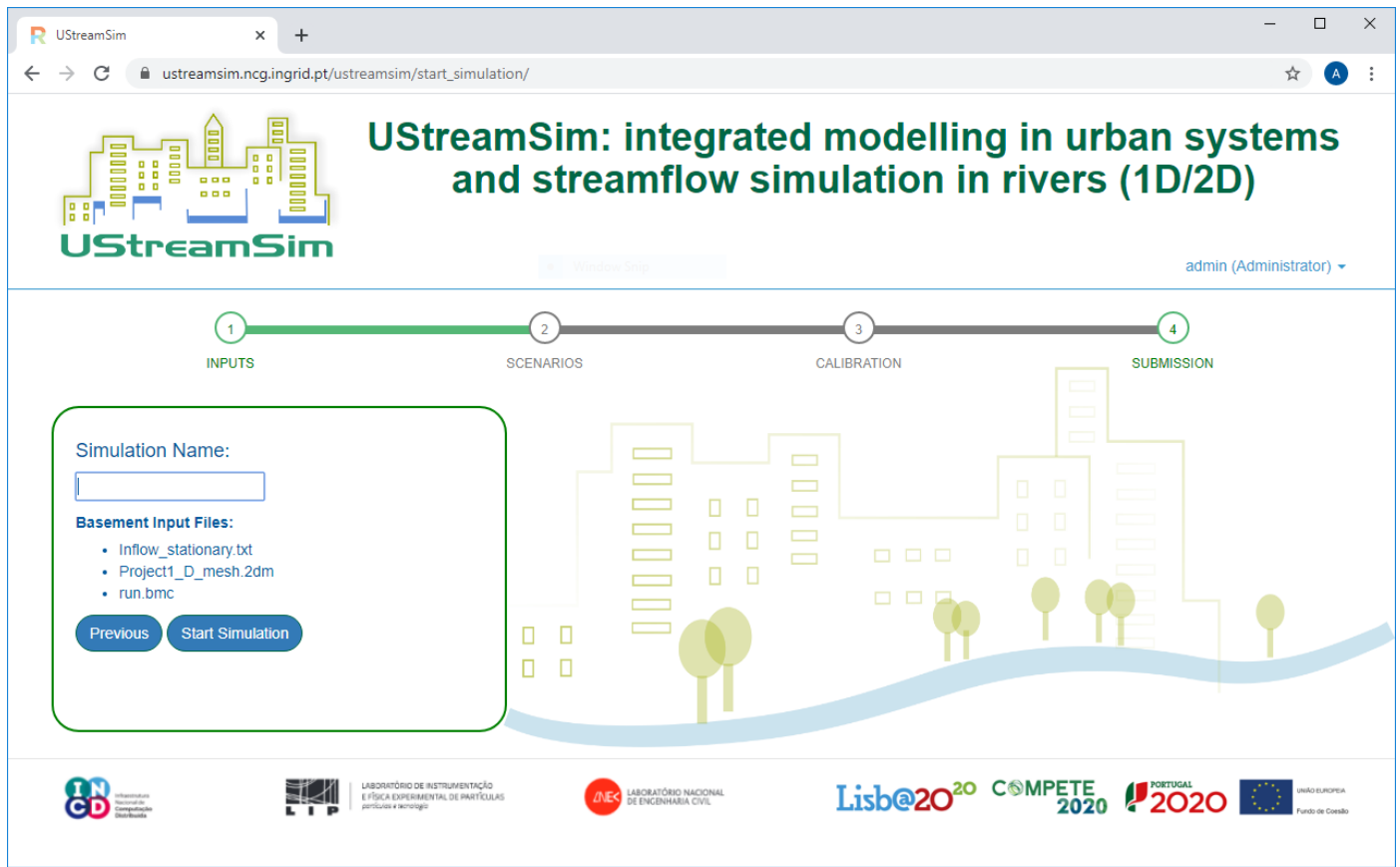
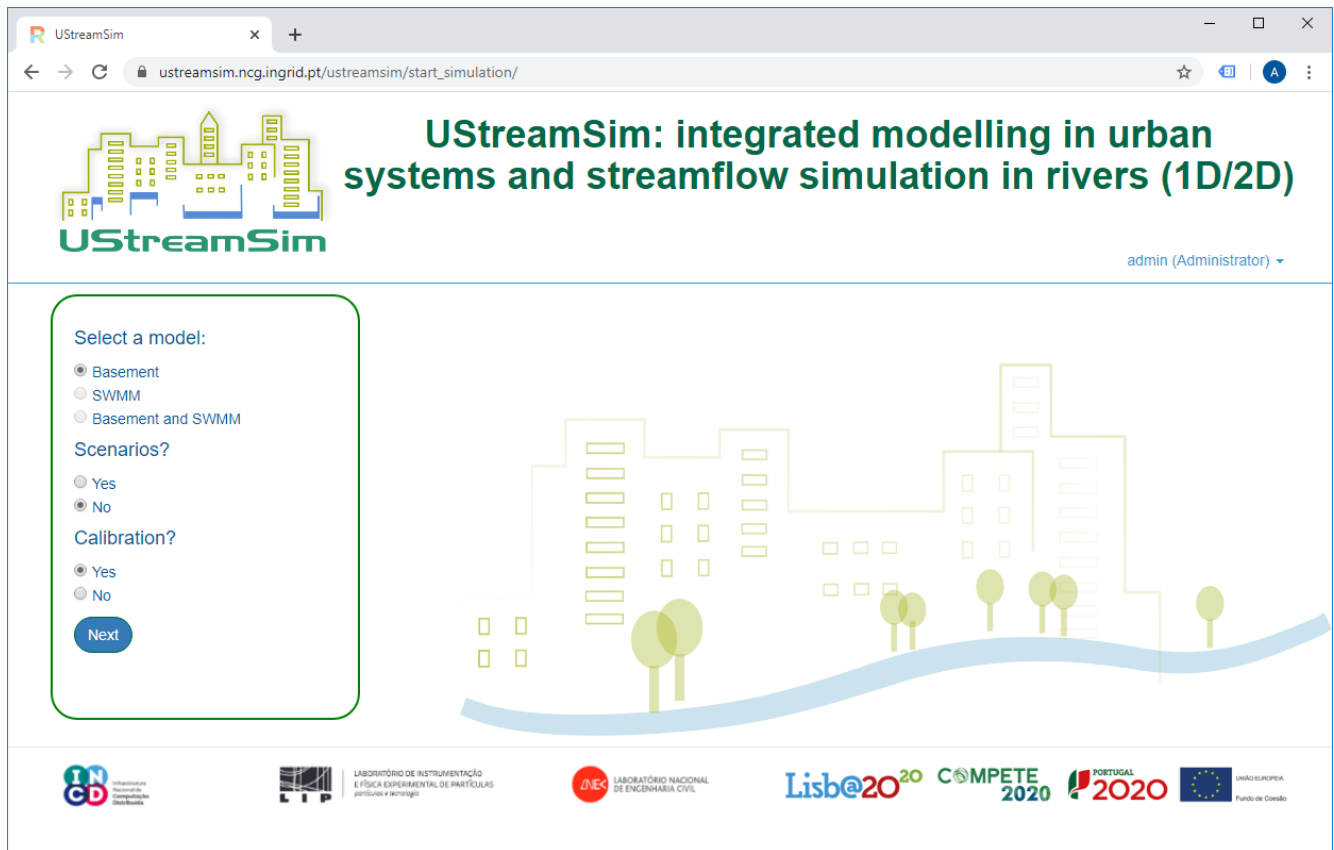


Figure 7 - Preview of Start Simulation menu

After submission, the user is redirected to the dashboard where he can manage his simulations.

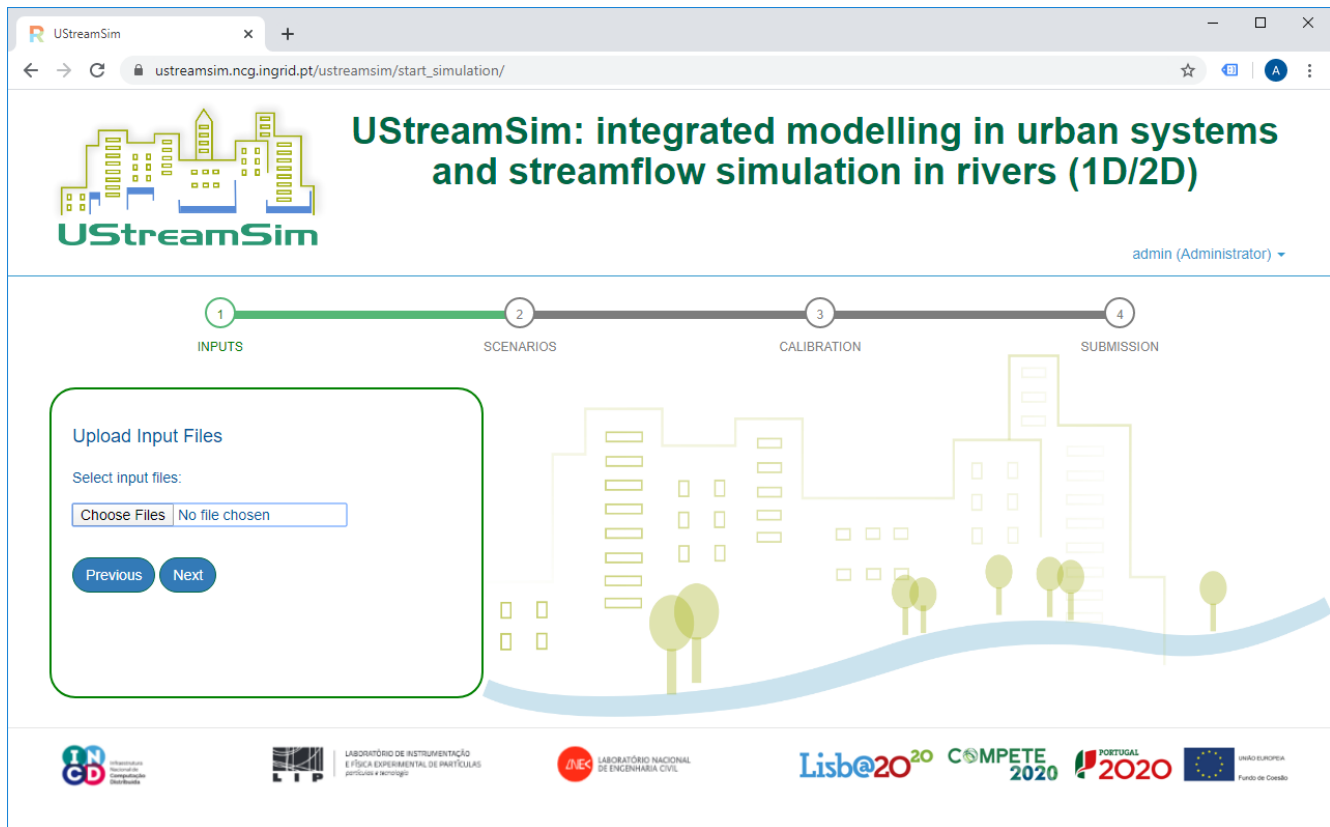
## Option 2: Run a Basement simulation for calibration

The user chooses the Basement model and “Yes” for Calibration. Automatically Scenarios is set to “No”. Click Next.



*Figure 8 - Preview of a simulation for calibration*

In the next window, the user can choose the input files he wants to use in the simulation.



*Figure 9 - Preview of Upload Input Files menu*

For this model is expected a command base file (.bmc), a mesh file (.2dm) and boundary condition files (.txt).

The Basement model calibration is done by varying the Default\_friction (Strickler coefficient) parameter. The user defines a minimum value, a maximum value and a step value to define the set of values. For each of this values a new simulation will be created. The minimum and maximum value can vary between 10 and 110 and the step value between 1 and 10. By default, the minimum and maximum value are set to 30 and the step value to 5 (Figure 10).

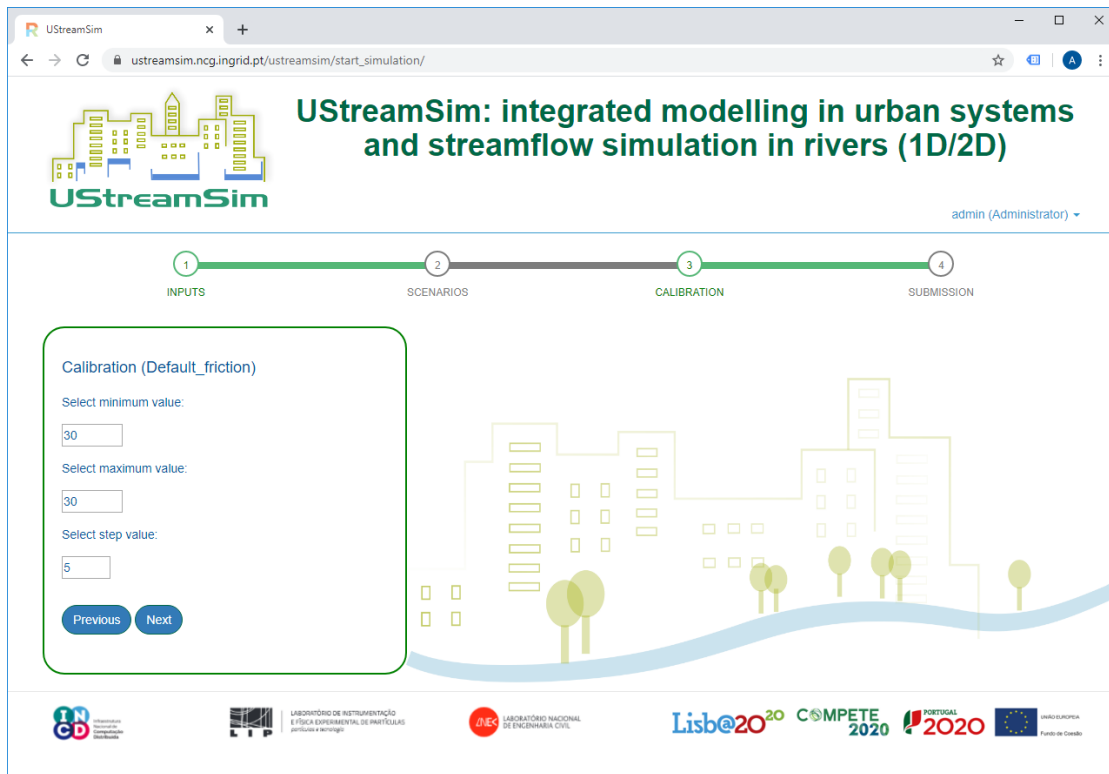
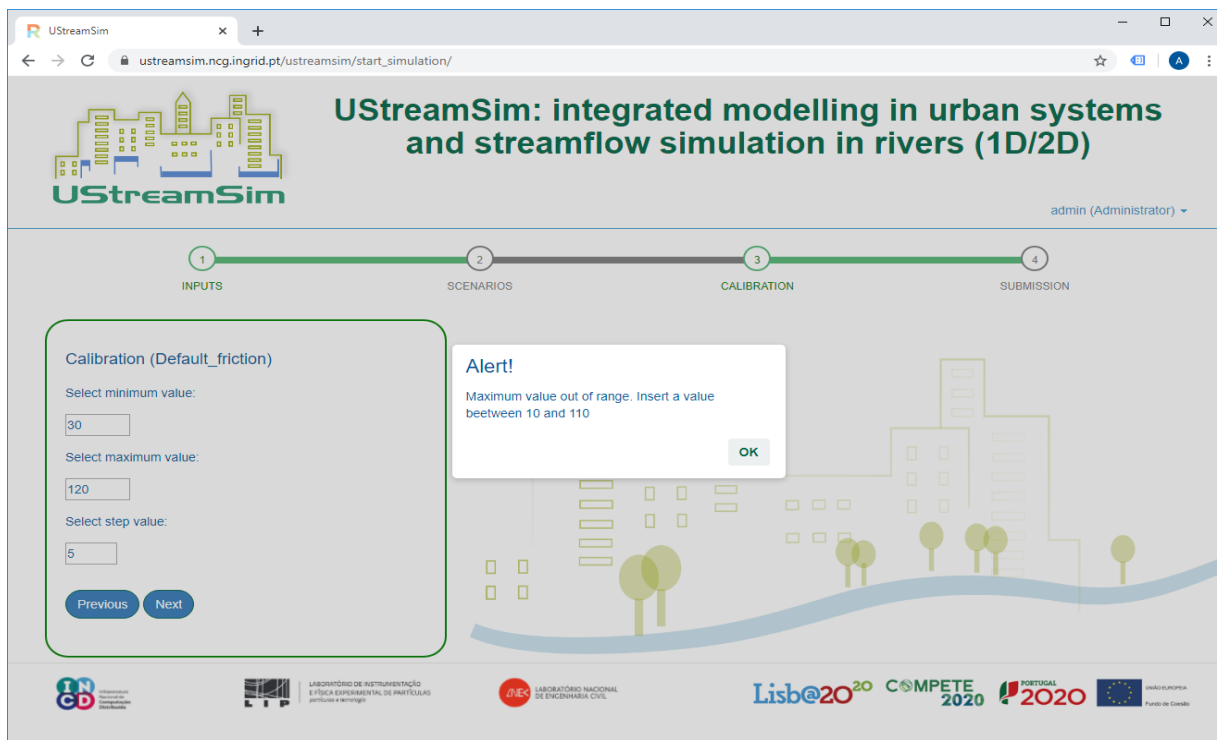


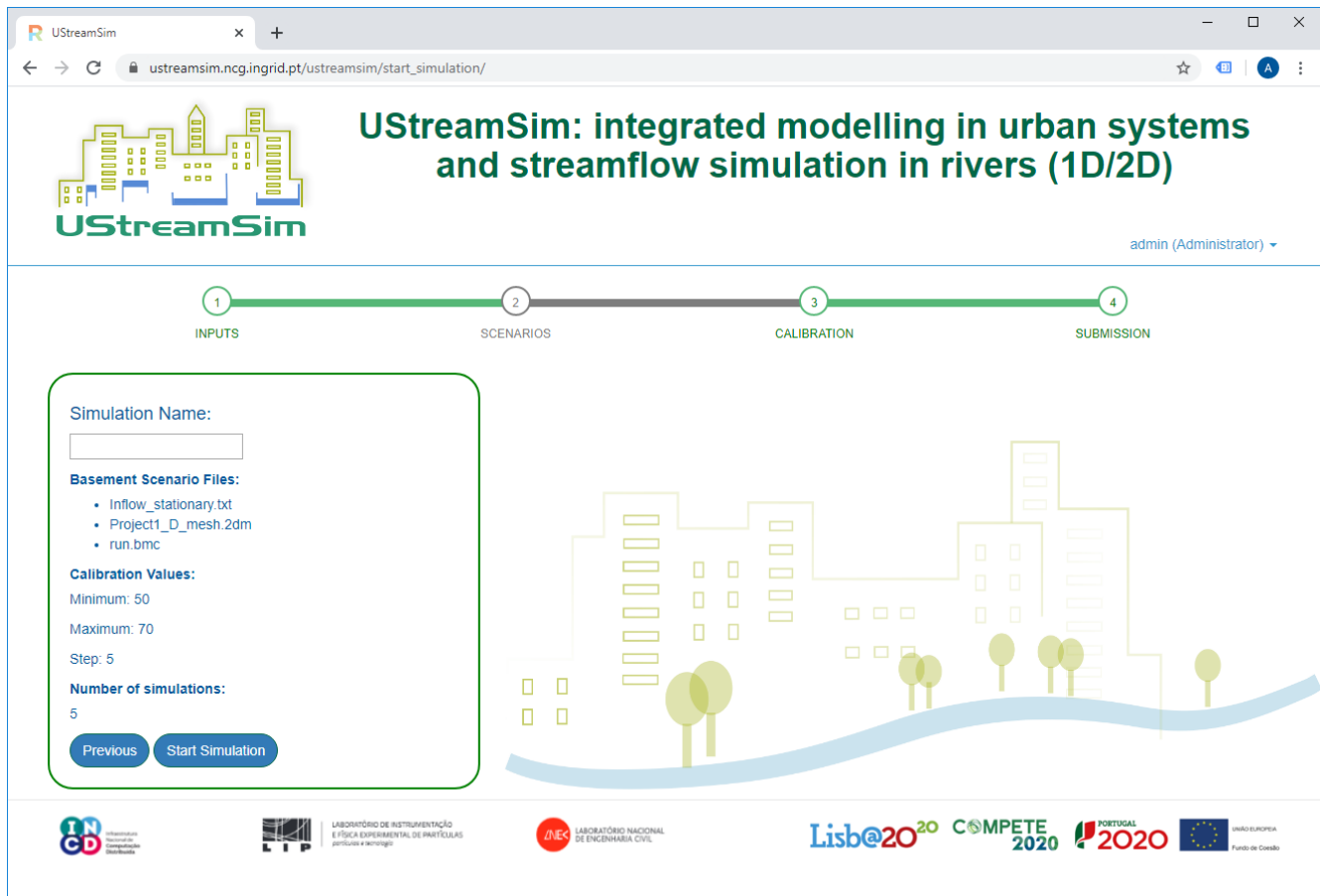
Figure 10 - Preview of Calibration values menu

When entering the values, if the user makes an error, an alert window is displayed indicating the error made (Figure 11).



**Figure 11 - Preview of Alert window when maximum value is out of range**

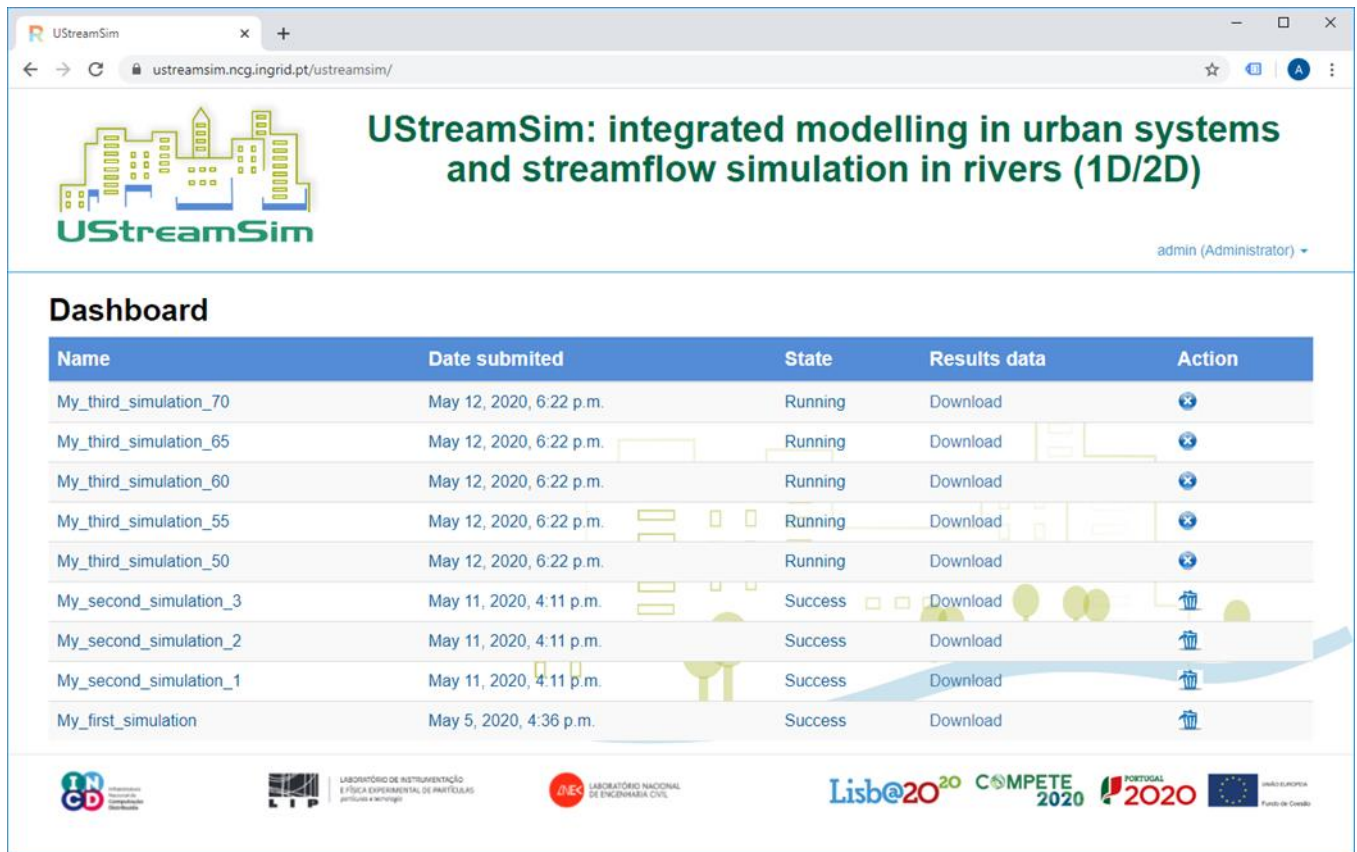
In this final step, the user must provide a name for the new simulation. Check if the input files and the calibration values are correct. If the user wants to make changes, click Previous and move backwards in the steps. As we run this model for calibration, we will have several simulations, one for each calibration value. Since there will be more than one run, the number of simulations is displayed for user information.



*Figure 12 - - Preview of Starting a Simulation for calibration*

If the number of simulations is greater than 5, the user will see an alert indicating the number of simulations. This is useful because launching too many simulations can take lots of time or can run out of computational resources. Press OK to close the alert and make changes if necessary. Click Start Simulation.

After submission, the user is redirected to the dashboard. The dashboard has now five new lines each one representing a different simulation for a different calibration value. This value is added to the simulation name to distinguish between them.

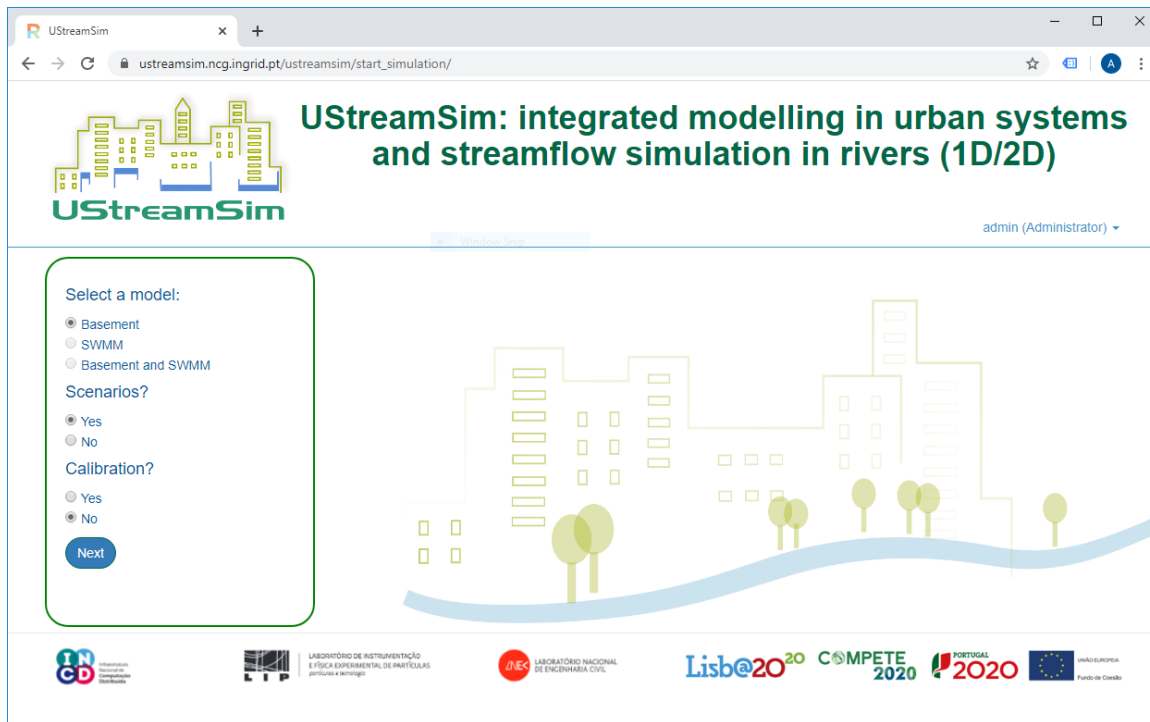


*Figure 13 - Preview of Dashboard with Running and Successful simulations*

### Option 3: Run a Basement simulation with scenarios

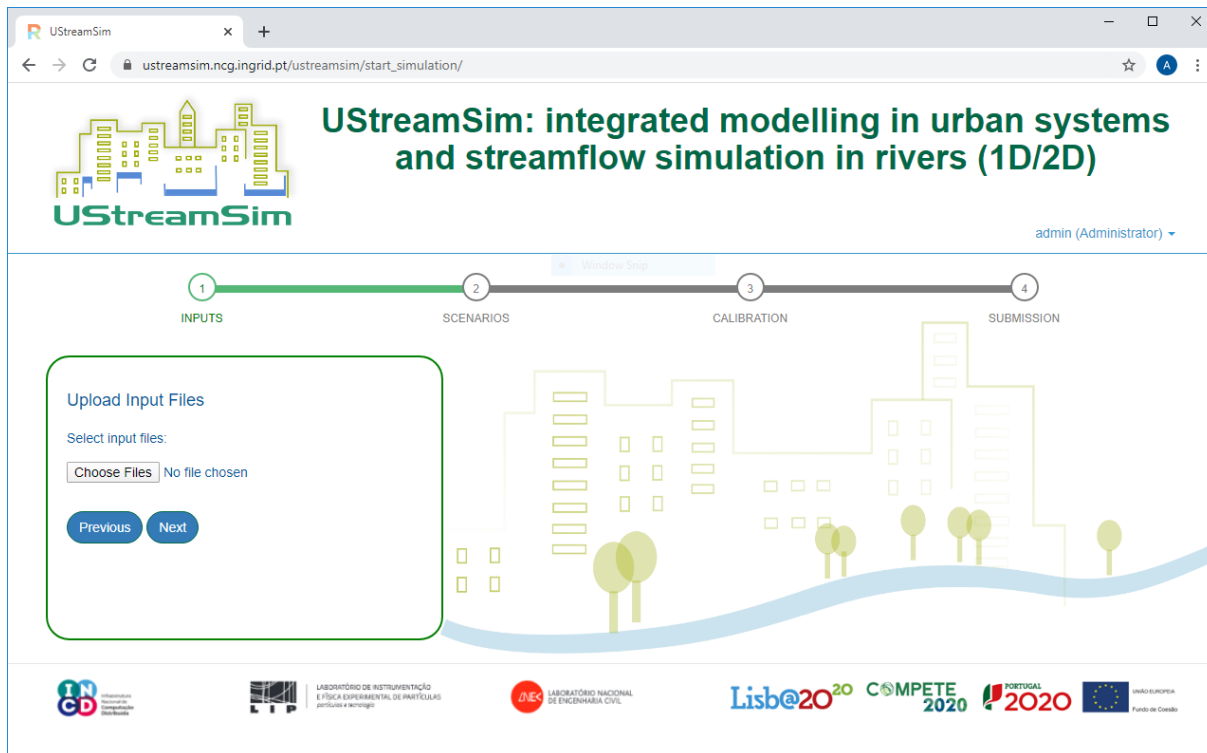
The user chooses the Basement model and “Yes” for Scenarios. Automatically Calibration is set to “No”. Click Next.





*Figure 14 - Preview of a simulation with scenarios*

In the next window, the user can choose the input files he wants to use in the simulation.



**Figure 15 - Preview of Upload Input Files menu**

As input files, this model expects a command base file (.bmc) and a mesh file (.2dm). Before uploading this files, the user must define in the command base file the blocks BOUNDARY where he/she wants to apply the various boundary condition files.

Next, the user has to choose the files he wants do use for simulating different scenarios. For each scenario the user can upload a set of files (.txt), one for each boundary condition defined in the command base file. It's important to upload the boundary condition files by the same order as BOUNDARY blocks appear in the configuration file (Figure 16).

Click Next.

The screenshot displays the UStreamSim web application in a browser. The page title is "UStreamSim: integrated modelling in urban systems and streamflow simulation in rivers (1D/2D)". The user is logged in as "admin (Administrator)". A progress bar at the top indicates four steps: 1. INPUTS (active), 2. SCENARIOS, 3. CALIBRATION, and 4. SUBMISSION.

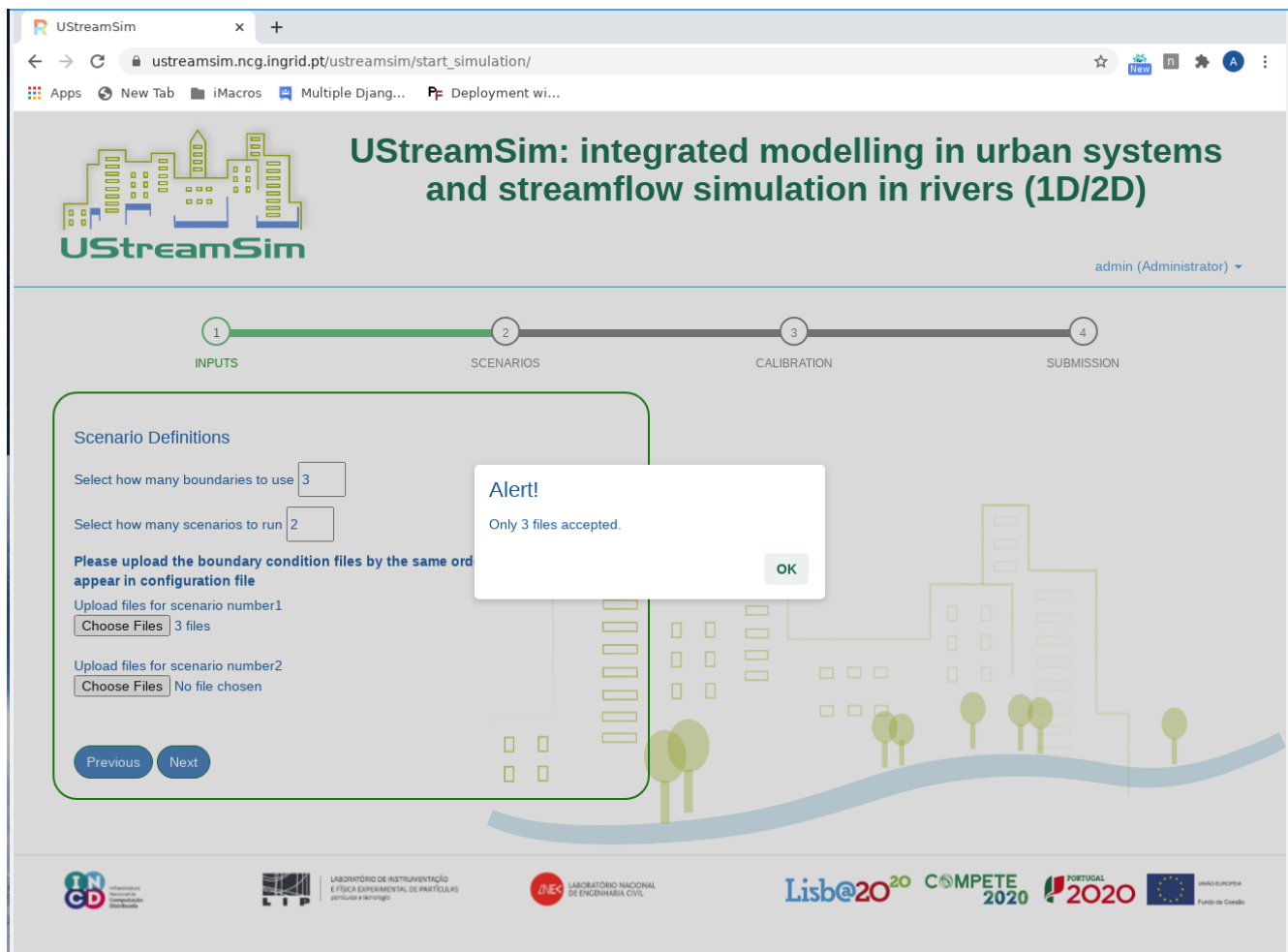
The "Scenario Definitions" section is highlighted with a green border. It contains the following fields and instructions:

- "Select how many boundaries to use" with a text input containing the value "3".
- "Select how many scenarios to run" with a text input containing the value "2".
- A blue instruction: "Please upload the boundary condition files by the same order as the boundaries appear in configuration file".
- "Upload files for scenario number1" with a "Choose Files" button and a counter showing "3 files".
- "Upload files for scenario number2" with a "Choose Files" button and a counter showing "3 files".

At the bottom of the form are "Previous" and "Next" buttons. The background features a stylized illustration of a city skyline and a river. The footer contains logos for INCD, LIP, INES, Lisb@2020, COMPETE 2020, PORTUGAL 2020, and the European Union.

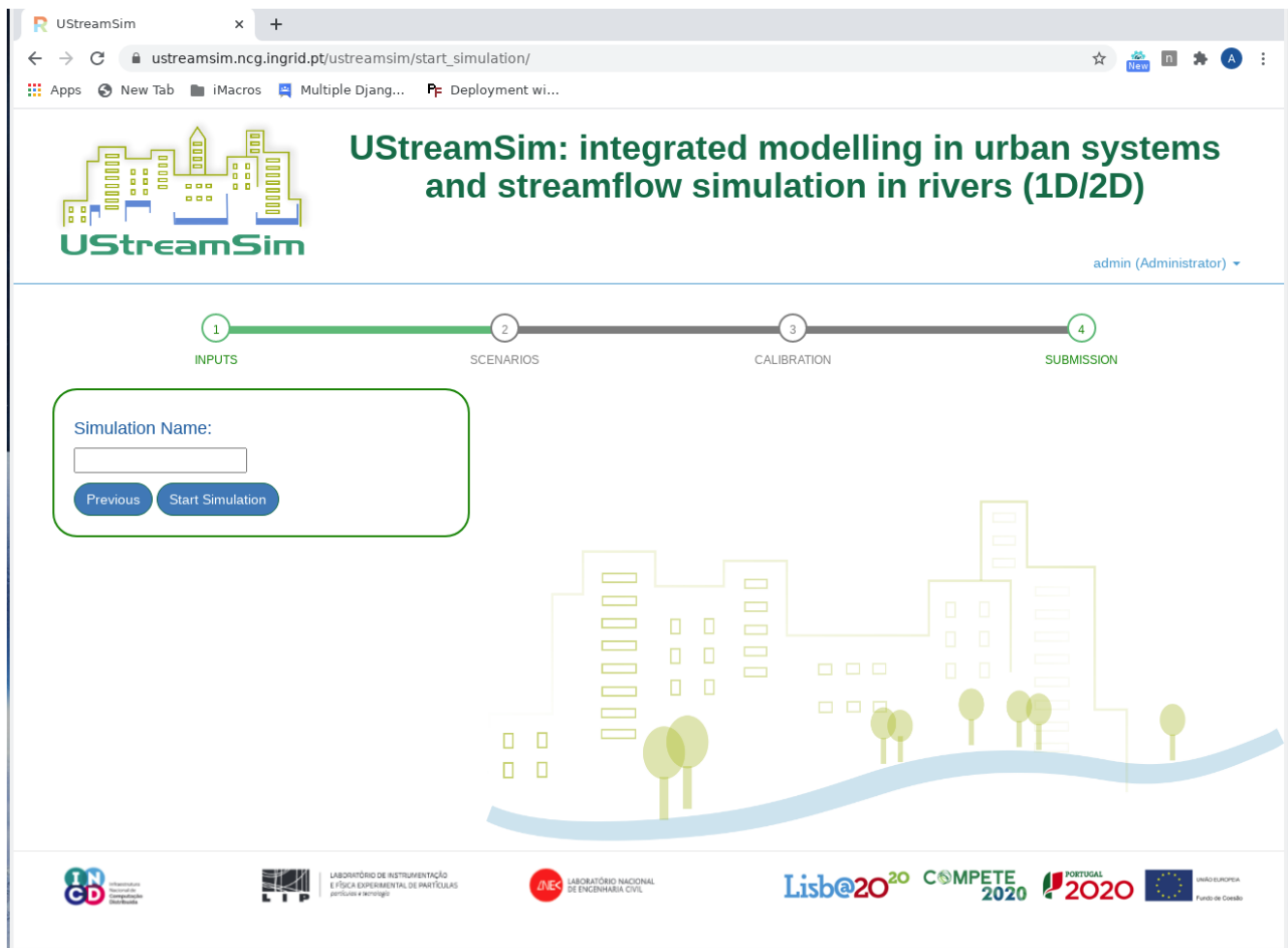
*Figure 16 - Preview of Scenario Definitions menu*

If the number of files uploaded in a scenario is different from the number of boundaries to use, an alert window is displayed indicating the correct number of files to upload (Figure 17).



*Figure 17 - Preview of scenarios value alert*

In this final step, the user must provide a name for the new simulation. If the user wants to make changes, click Previous and move backwards in the steps. As we run this model with scenarios we will have several simulations, one for each scenario to run (Figure 18).



*Figure 18 - Preview of Starting a Simulation with scenarios*

Click Start Simulation.

After submission, the user is redirected to the dashboard. The dashboard has now new lines each one representing a different simulation for a different scenario run. A sequential number is added to the simulation name to distinguish between them (Figure 19).

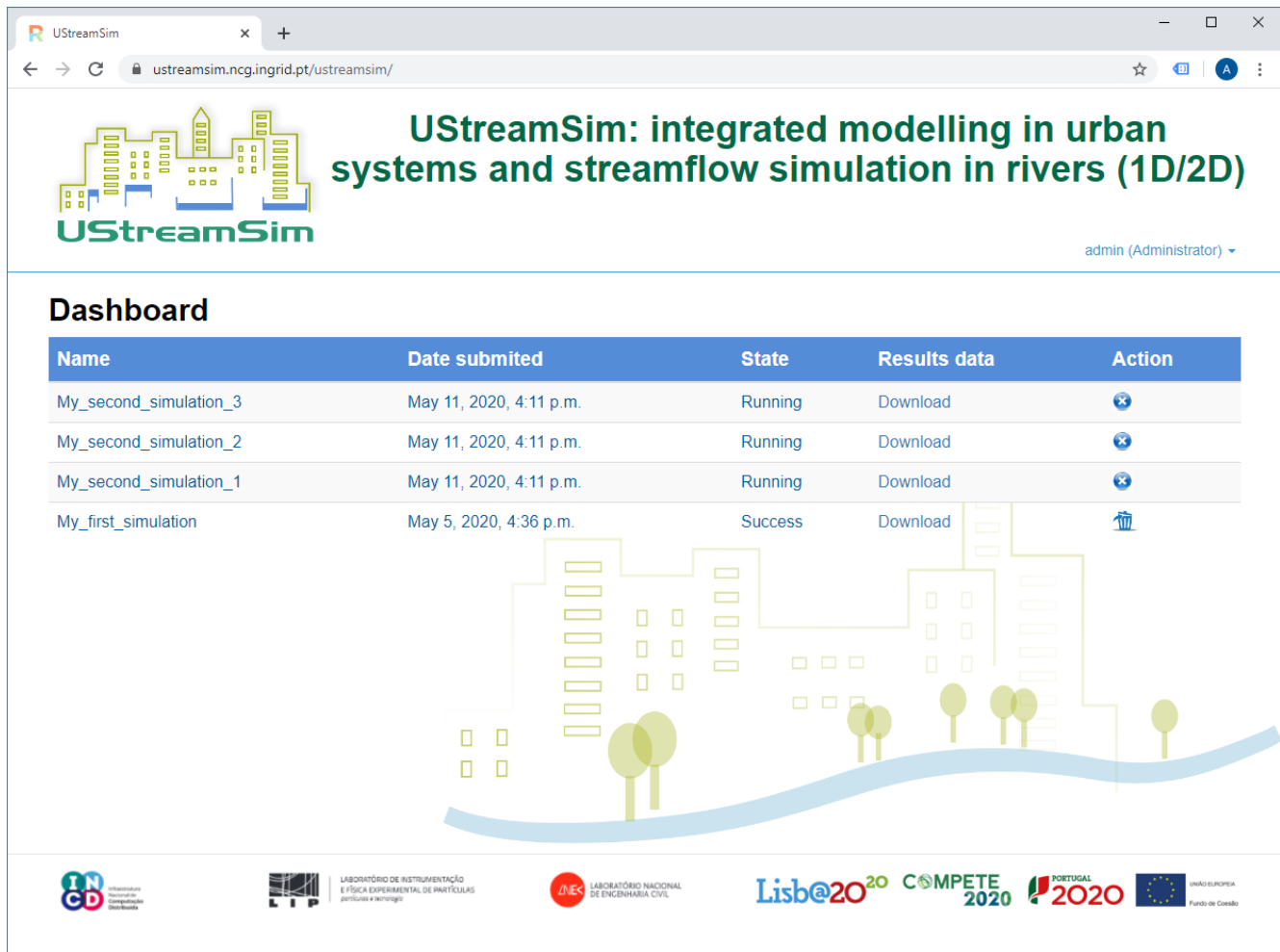


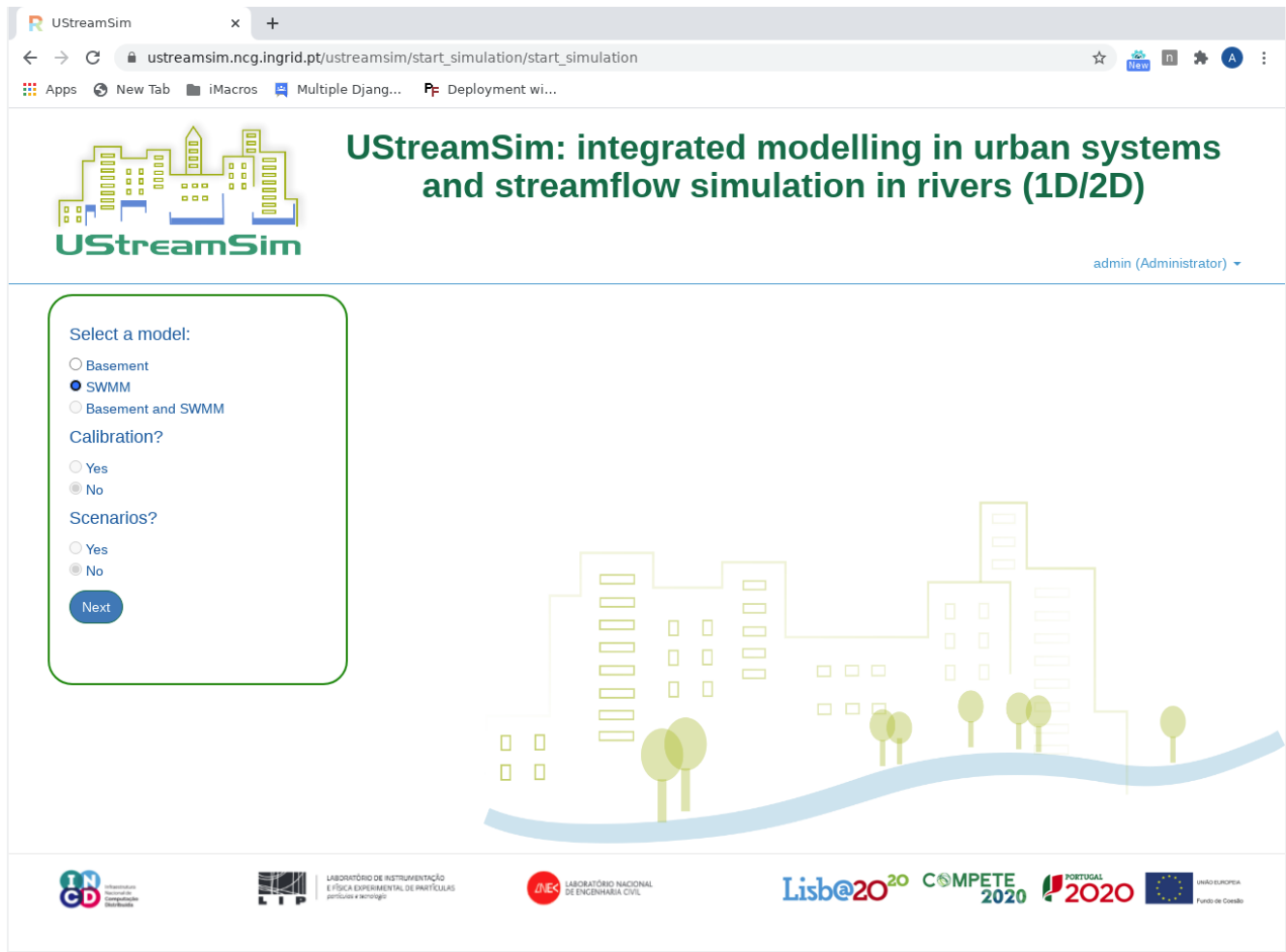
Figure 19 - Preview of Dashboard with running simulations

## STEP 6 – Run a SWMM simulation

For now, when choosing the SWMM model, the user can only run a single simulation (option 1). Run multiple simulations for calibration parameter values purposes or run multiple simulations using different scenarios will be available later.

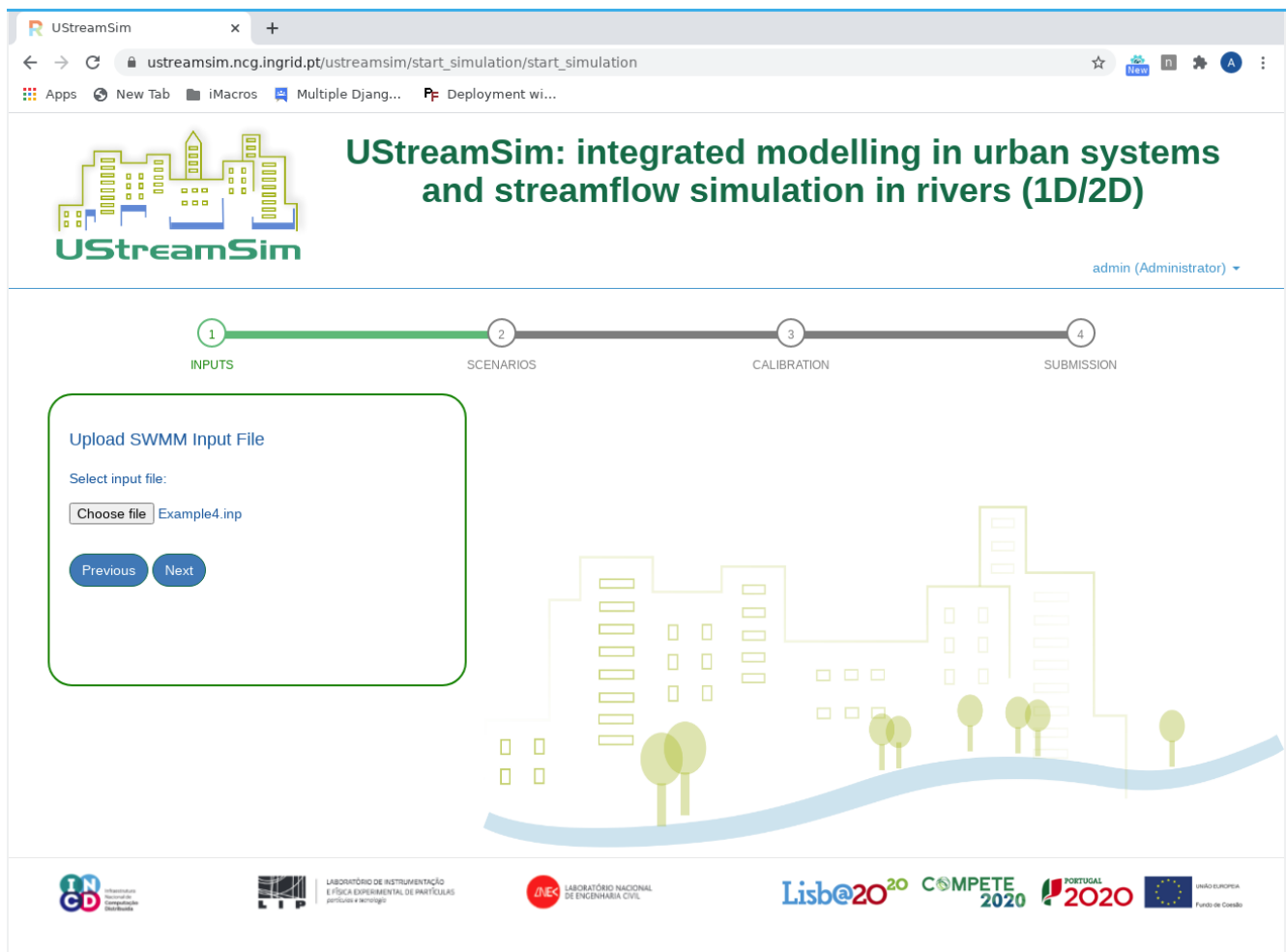
### Option 1: Run a stand-alone SWMM simulation

The user chooses the SWMM model and automatically the option for Scenarios and Calibration are disabled (Figure 20).



*Figure 20 - Preview of choosing the SWMM model*

After Clicking Next, a new menu is shown (Figure 21).

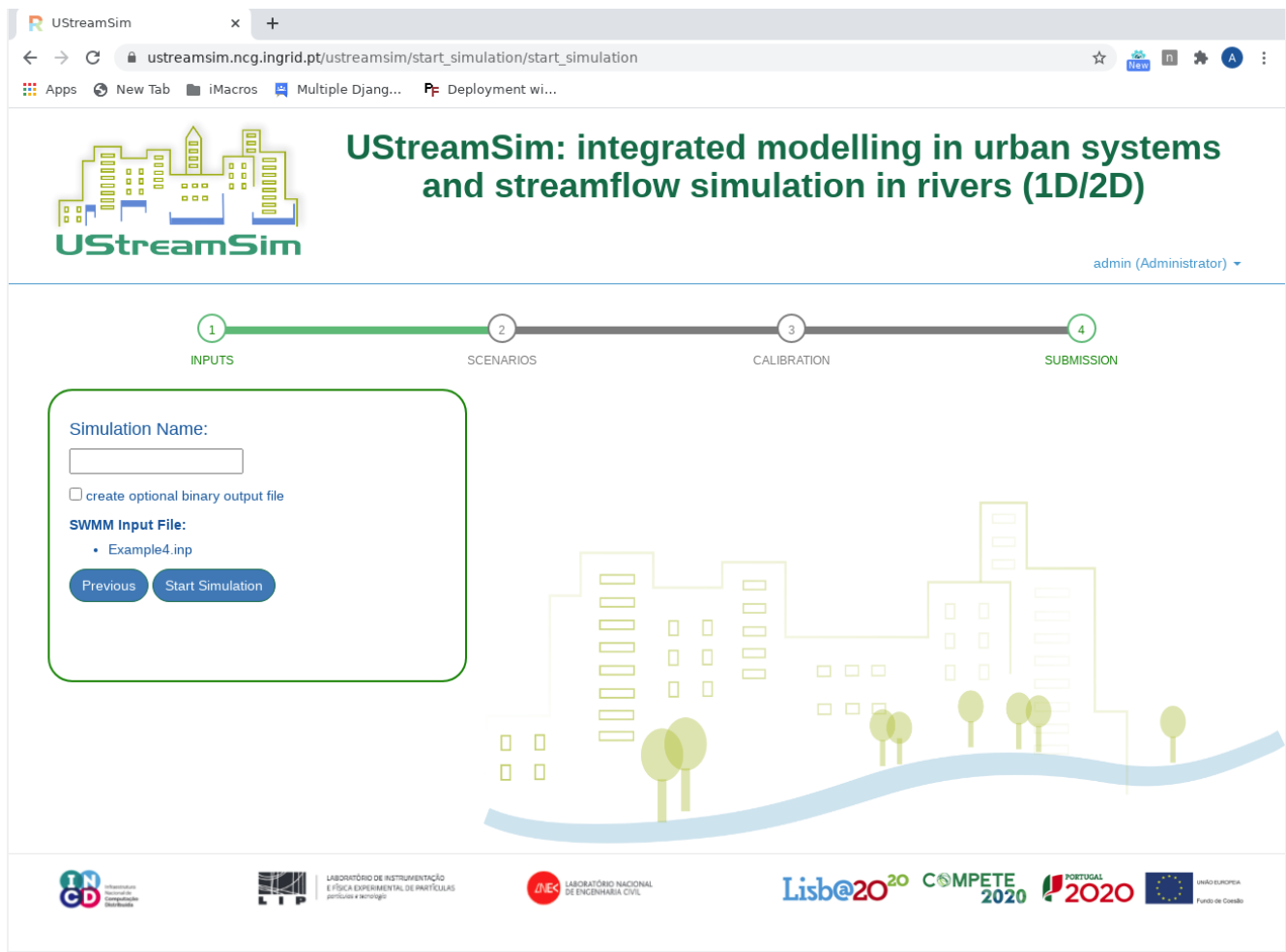


*Figure 21 - Preview of Upload Input file menu*

In this new window, the user can choose the input file (.inp) he/she wants to use in the simulation. Click Next.

The user must then provide a name for the new simulation. This identification will show up in the simulations table. The user can also choose if he/she wants to create an optional binary output file. This optional file stores all time series results in a special binary format that will require a separate post-processor program for viewing. If no binary output file is created, then all time series results will appear in the default report output file (Figure 22).





*Figure 22 - Preview of Upload Input file menu*

The user can go back if the file uploaded is not correct. After checking if the input file is correct, if the user wants to make changes, click Previous and move backwards in the steps. Otherwise, just click Start Simulation to initiate the simulation.

As many simulations can take several days to complete, the user needs to be notified of what is going on with their runs. Whenever a simulation ends, the user receives an email with information about the simulation. This email contains the name of the simulation, the date it was created and the state with which it ended (Figure 23).

From [ustreamsim@lnec.pt](mailto:ustreamsim@lnec.pt) ☆

Subject **Ustreamsim simulation**

To Me <[amendes@lnec.pt](mailto:amendes@lnec.pt)> ☆

The simulation `My_eighth_simulation` created at 14/05/2020, 13:20:17 ended with success.

*Figure 23 - Preview of Email send to the user*

## Acknowledgements

The USTREAMSIM service is one of the thematic services of the INCD project, funded by the Lisboa2020 Operational Program through the INCD project (LISBOA-01-0145-FEDER-022153) funded by European Union under grant agreement No. 777536.

This service takes advantage of models freely available online. We would like to acknowledge:

- [ETH Zurich - Laboratory of Hydraulics, Hydrology and Glaciology \(VAW\)](#) for the availability of model [BASEMENT](#)

The development team also acknowledges all colleagues that have helped to setup and test the several versions of the service.

