



## User Manual

accu:rate GmbH

accu:rate GmbH

Goethestrasse 28 | D-80336 Munich

+49 89 21553869

[crowd-it@accu-rate.de](mailto:crowd-it@accu-rate.de)

January 26, 2026

# Contents

1	Preface	7
1.1	Disclaimer	7
1.2	Introduction	7
1.3	Document Structure	8
1.3.1	Microscopic Simulations	8
1.3.2	Macroscopic Calculations	8
2	crowd:it Introduction	9
2.1	User Interface	9
2.1.1	General Menu Bar	9
2.1.2	Mode-Dependent Menu Bar	10
2.1.3	Toolbar	11
2.1.4	Floor Plan View	13
2.1.5	Floor infos	13
2.1.6	Playback bar	13
2.1.7	Information Panel	13
2.2	Updates and Help	14
2.2.1	Documentation	14
2.2.2	Keyboard Shortcuts	15
2.2.3	Help	15
2.2.4	Quick Intro for Microscopic Simulations	15
2.2.5	Quick Intro for Macroscopic Calculation	15
2.2.6	License	15
2.2.7	Versions	16
2.2.8	Quick Intro for Microscopic Simulations	16
2.2.9	Quick Intro for Macroscopic Calculation	16
2.2.10	How-tos for a quick start	16

## Contents

---

2.2.11	Help from accurate	16
2.2.12	License	17
2.2.13	Updates	17
2.3	General	17
2.3.1	Escape ( <b>ESC</b> key)	17
2.3.2	Shortcuts	17
2.3.3	Selection Behavior	17
2.3.4	Probability Distributions	19
2.3.5	Inserting a Background Image	19
2.3.6	Logo	20
3	Floors	21
3.0.1	Insert	21
3.0.2	Replace	21
3.0.3	Viewing and Editing Properties	21
3.0.4	Move	22
3.0.5	Delete	22
4	Geometry	23
4.1	Create Geometry	23
4.2	Edit Geometry	23
4.3	Simplify Geometry	24
4.4	Tips for Creating Geometry	24
5	Microscopic simulation	25
5.1	The microscopic modeling process step by step	25
5.1.1	Step 1: Prepare the geometry	25
5.1.2	Step 2: Modeling	26
5.1.3	Step 3: Simulate	26
5.1.4	Step 4: Analyze	26
5.1.5	Step 5: Export results	26
5.2	Quick intro	27
5.3	Prepare Geometry	27
5.3.1	Drawing Geometry yourself	27
5.3.2	Importing Geometry from a CAD File	28
5.3.3	Preparing Geometry with Vectorworks	29
5.3.4	Simplify Floorplan	33

5.4	Creating Simulation Objects	33
5.4.1	Simulation objects settings	34
5.4.2	Origin	34
5.4.3	Destination	36
5.4.4	Scaled Area	37
5.4.5	Directed Scaled Area	37
5.4.6	Waiting zone	37
5.4.7	Queue	41
5.4.8	Stairs	41
5.4.9	Escalator	42
5.4.10	Elevator	43
5.4.11	Portal	44
5.4.12	Macro Area	44
5.5	Paths, Path Snippets, and Sets	45
5.5.1	Paths	46
5.5.2	Path snippets	49
5.5.3	Sets	50
5.5.4	Selection Processes	51
5.5.5	Open Hierarchy Window	55
5.6	Population Settings	56
5.6.1	Personas	56
5.6.2	Groups	57
5.7	Simulation Calculation	57
5.7.1	Simulation Menu	57
5.7.2	Start Simulation	58
5.7.3	Run simple assessment	60
5.7.4	Further Editing	60
5.8	Visualize	61
5.8.1	Agents	61
5.8.2	Display Evaluations	62
5.8.3	Colors	62
5.8.4	Object Labels	63
5.8.5	Set Start Time	63
5.8.6	Playback Speed	63
5.9	Analyze	63
5.9.1	Agents	63

5.9.2	Evaluation objects	64
5.9.3	Evaluation Settings	67
5.10	Result Presentation	68
5.10.1	Charts	68
5.10.2	Summary Tables	73
5.10.3	Critical area detection	75
5.10.4	Heatmaps	76
5.11	Export	78
5.11.1	Screenshot	78
5.11.2	Video	79
5.11.3	Reporting	80
5.12	Quick Edit	80
5.13	Advanced Tools	81
5.13.1	Automatic Report Generation via Console	81
5.14	Troubleshooting	82
5.14.1	Slow simulation calculation: Performance tips and tricks	82
5.14.2	Out of memory error	84
6	Macroscopic Calculations	85
6.1	Workflow	85
6.2	Quick Introduction	86
6.3	Define Way Elements	86
6.3.1	Create start element	86
6.3.2	Create Corridor	87
6.3.3	Create Stair up or down	88
6.3.4	Create Bottleneck	88
6.4	Connect way elements	89
6.4.1	Merging	89
6.4.2	Splitting	89
6.5	Overview of all modeled way elements	90
6.6	Persona table	90
6.7	Modeling Tips	91
6.8	Visualization Options in Macroscopic Mode	91
6.9	Calculation	92
6.10	Results	92
6.10.1	Overview Table	92
6.10.2	Detailed Analysis	93

## Contents

---

6.10.3 Exporting Results . . . . .	93
6.10.4 Calculation Parameters . . . . .	94
6.11 Analysis . . . . .	94
6.12 Export . . . . .	95
7 Glossary	96

# Chapter 1

## Preface

### 1.1 Disclaimer

This user manual introduces the user to the operation of the crowd:it software. The models and algorithms implemented in crowd:it are derived from accepted and recent research literature. Nevertheless, *accu:rate* cannot assure that reality will not deviate from any and/or all simulation results. The proper use of crowd:it is the sole responsibility of the user.

### 1.2 Introduction

Congratulations! You have selected one of the most advanced pedestrian simulators on the market. Founded in 2014, following studies at the Technical University of Munich (TUM), *accu:rate GmbH* provides the latest research in crowd simulation to practitioners including engineering firms, event organizers, crowd managers and firefighters. Regardless of the scenario type; be it low numbers of people in an office, or enormous industrial complexes, museums and outdoor events, the state-of-the-art mathematical models at the core of our software deliver valid results you can trust. With *crowd:it's* clean and self-explanatory user interface, you can focus on the project and not the software.

The following documentation serves as a reference. Online video tutorials and in-person training sessions are also available.

## 1.3 Document Structure

The document is divided into two main sections:

1. Description of the creation of microscopic simulations
2. Description of the creation of macroscopic calculations

### 1.3.1 Microscopic Simulations

The documentation follows the typical workflow of a project. Usually, this begins with the simplification of CAD plans before certain objects and areas are marked as simulation objects (e.g., entrances and exits).

For this purpose, commercial 2D CAD software packages, such as Autodesk™ AutoCAD, are suitable. Plans created in CAD software can be directly imported into *crowd:it*.

Next, the movement sequences to be simulated and the paths of individuals (hereinafter referred to as *agents*) are modeled. Once the simulation calculation is complete, the behavior can be analyzed.

Finally, results are generated and exported for further use.

### 1.3.2 Macroscopic Calculations

Macroscopic calculations are a simplified method for analyzing pedestrian flows. They enable an initial estimation of evacuation times and can be used in *crowd:it* exclusively for the calculation of evacuation scenarios.

The description also follows the typical workflow:

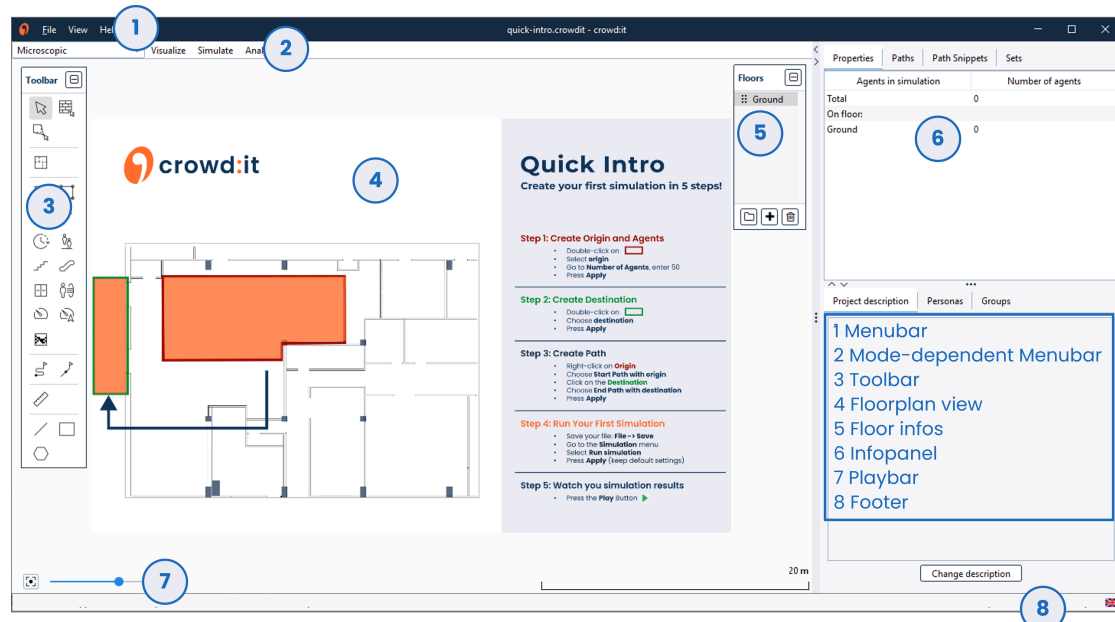
- No precise geometry is required for a macroscopic calculation. However, it is recommended to import a floor plan of the scenario to be analyzed as a background image.
- Various path elements and routes can then be defined.
- The calculation is performed instantly, and the results are displayed in the right section of the user interface, where they can be analyzed and exported.

We wish you great success with your *crowd:it* project!

# Chapter 2

## crowd:it Introduction

### 2.1 User Interface



The user interface is divided into several sections, which are explained below.

#### 2.1.1 General Menu Bar

At the top of the screen is the **menu bar**, which contains general functions: On the far left is **File**. Here, you can open and save simulations or add a general description to the scenario (which is displayed in the **Information Panel**).

Next to **File** is **View**. Here, you can adjust general display settings, such as showing a legend.

The following options are available:

- **Legend:** Shows/hides the legend at the bottom left of the display.
- **Scale:** Shows/hides the scale of the geometry at the bottom right of the display.
- **Background Image:** Shows/hides a background image if one has been set. A background image can be inserted as described [here](#).
- **Logo:** Shows/hides a logo if one has been set. A logo can be set as described [here](#).
- **Reset Toolbar:** Moves the toolbar back to its original position at the top left of the drawing area.
- **Reset Floor Selector:** Resets the floor selector to its original position at the top right of the drawing area.

Each of these commands is independent of the others.

Under **Help**, you can access the menu items **Open documentation**, **Shortcuts**, **Get help from accu:rate**, **Quick intro for microscopic simulations**, **Quick intro for macroscopic calculations**, **License details**, and **About**. More information can be found [here](#).

## 2.1.2 Mode-Dependent Menu Bar

Below the general menu bar is a mode-dependent menu bar that allows switching between different calculation modes.

The dropdown menu on the far left lets you switch between **Microscopic** and **Predtetschenski & Milinski** (macroscopic mode).

In **Microscopic Mode**, the following menu options are available:

- **Visualize:** Adjust visualization settings ([see here](#)).
- **Simulate:** Start simulations and manage results ([see here](#)).
- **Analyze:** Access tools for analyzing simulation results and exporting screenshots and videos ([see also here](#)).

In **Macroscopic Mode**, the following menu option is available:

- **Visualize:** Adjust visualization settings ([see here](#)).

### 2.1.3 Toolbar



The toolbar provides quick-access tools for editing and is displayed on the left. It changes based on the selected simulation mode (microscopic or macroscopic).

#### Toolbar in Microscopic Mode


Depending on whether simulation results are available, the toolbar offers different tools. The selection tool is active by default, allowing you to select simulation objects.

The toolbar is divided as follows:












##### **Select Objects (Both Edit and Result Mode)**

-  Select simulation objects
-  Select walls

##### **Edit Geometry (Edit Mode Only)**

-  Edit simulation objects and evaluations
-  Simplify floor geometry

##### **Draw New Objects (Edit Mode Only)**





-  Draw and Assign new wall
-  Draw and Assign new simulation object
-  Draw and Assign Origin
-  Draw and Assign Destination
-  Draw and Assign Waiting Zone
-  Draw and Assign Queue
-  Draw and Assign Stair
-  Draw and Assign Escalator
-  Draw and Assign Elevator
-  Draw and Assign Portal
-  Draw and Assign Macroscopic area

##### **Connect Objects (only in Edit Mode)**

-  Create Path

-  Create Path Snippets

### **Measure and Evaluate (both in Edit and Result Mode)**



-  Measure distances on the plan
-  Draw Tripwire
-  Draw Evaluation Rectangle
-  Draw Evaluation Polygon

### **Select Agents (only in Result Mode)**

-  Identify single or multiple Agents

### **Toolbar in Macroscopic Mode**








#### **Select Objects**

-  Select simulation objects
-  Select walls

#### **Edit Geometry**

-  Edit object geometry

#### **Draw New Objects**

-  Draw wall
-  Draw simulation object
-  Draw start element
-  Create corridor
-  Draw stair up
-  Draw stair down
-  Create bottleneck

#### **Connect Objects**

-  Connect way elements

#### **Measure**

-  Measure distances on the plan

## 2.1.4 Floor Plan View

The **Floor Plan View** displays the geometry of the simulation. Here, you can view and edit your simulation and the associated geometry.

**Navigate** within the view by holding the scroll wheel or using the arrow keys on your keyboard.

You can **zoom in and out** by scrolling with the mouse or using the slider located at the bottom left of the floor plan view.

Next to the slider, you can center the **floor** by clicking on . The zoom will adjust so that the floor is fully visible.

## 2.1.5 Floor infos

The floor information is located at the top-right corner of the floor plan view. Here, you can switch between floors, create new ones, or delete existing ones. For more details, see [here](#).

## 2.1.6 Playback bar

The playback bar only appears in microscopic mode when a simulation has already run. It allows you to play, pause, and rewind the simulation. Additionally, the simulation can be played step by step. The timeline shows the current time step, and the pointer can be moved forward or backward. The playback speed can be adjusted under **Visualize > Set playback speed ...**

## 2.1.7 Information Panel

The information panel is located to the right of the floor plan view and displays context-related information.

### Information Panel in Microscopic Mode

#### ***Properties Tab***

This tab is divided into two sections:

- Upper section: Displays general information about currently selected objects.
- Lower section: Allows simulation-related settings.

### *Upper Section*

During the modeling phase, the upper section displays information about the currently selected objects in the floor plan.

- If **no object** is selected, the total number of agents in the scenario and per floor is displayed.
- If **one or more objects** are selected, a table with all currently selected simulation objects appears.

A right-click on a table entry opens a context menu with further options. If **simulation results are available** and one or more agents are selected in the **floor plan view**, the tab shows specific information about these agents.

### *Lower Section*

The lower section contains the following tabs:

- **Project Description** – Store metadata about the project.
- **Personas** – Define different agent groups (*personas*) ([see here](#)).
- **Groups** – Define various groups ([see here](#)).
- **Evaluation** (only visible if simulation results are available): Configure detailed congestion evaluation settings.

#### ***Paths, Path Snippets, Sets Tab***

Here, you can view and edit the existing [Paths](#), [Path Snippets](#), and [Sets](#) in a tree structure.

#### ***Result Tab (Only visible if simulation results are available)***

This tab summarizes simulation results, including statistics on agents, sources, and destinations.

## **2.2 Updates and Help**

### **2.2.1 Documentation**

Under **Help > Open documentation**, you can access the latest *crowd:it* documentation.

## 2.2.2 Keyboard Shortcuts

Under **Help** > **Shortcuts**, you can view an overview of keyboard commands.

## 2.2.3 Help

## 2.2.4 Quick Intro for Microscopic Simulations

Under **Help** > **Quick intro for microscopic simulations**, a *crowd:it* file opens, providing a fast introduction to using *crowd:it*.

## 2.2.5 Quick Intro for Macroscopic Calculation

Under **Help** > **Quick intro for macroscopic calculations**, you find a introduction for macroscopic calculations.

### Help via Hint Buttons

In many dialogs, you will find hint buttons marked with . Clicking this button provides context-sensitive help and explanations.

### Help from accu:rate

Under **Help** > **Get Help from accu:rate**, you can submit a support request. For data protection reasons, project details are not included by default but should be provided for faster assistance.

### How-tos for a quick start

On our website, we offer several short how-tos to help you get started with *crowd:it*. You can find them [here](#).

## 2.2.6 License

Under **Help** > **License Details**, you can view and edit license information.

## 2.2.7 Versions

### Version Information

Under **Help** > **About *crowd:it*...**, you can view the current version of *crowd:it* and the simulation core, as well as access the latest changelog.

### Updates

Each time *crowd:it* starts, it automatically checks for updates. If available, they are downloaded in the background. Once the download is complete, a notification appears. Clicking it restarts *crowd:it* with the new version.

### Version Conversion

Some updates require a new file format. When opening the application next time, a dialog will appear for automatic conversion. The conversion is necessary to continue working with the file. A new file is created, while the original file remains unchanged.

## 2.2.8 Quick Intro for Microscopic Simulations

Under **Help** > **Quick intro for microscopic simulationse**, a *crowd:it* file opens, providing a fast introduction to using *crowd:it*.

## 2.2.9 Quick Intro for Macroscopic Calculation

Under **Help** > **Quick intro for macroscopic calculations**, you find a introduction for macroscopic calculations.

## 2.2.10 How-tos for a quick start

We offer several short how-tos on our website to help you get started with *crowd:it*. These can be found [here](#).

## 2.2.11 Help from *accu:rate*

Under **Help** > **Get Help from *accu:rate***, you can submit a support request.

### 2.2.12 License

Under **Help** > **License details**, you can view and edit license information.

### 2.2.13 Updates

At each startup, *crowd:it* automatically checks for updates and downloads them in the background.

## 2.3 General

The software includes some recurring behaviors, which are described in this chapter.

### 2.3.1 Escape (ESC key)

Press Esc to cancel a running action.

The following actions can be terminated with Esc:

- Creating paths
- Creating simulation objects and way elements
- Drawing or moving measurement lines (tripwires) or polygons
- Tracking an agent
- Editing settings

### 2.3.2 Shortcuts

In *crowd:it*, various functions can be accessed via shortcuts. A complete overview can be found in the **menu bar** under **Help** > **Shortcuts...**

### 2.3.3 Selection Behavior

Some elements, such as simulation objects, walls, and agents, can be selected directly in the **floor plan view**.

The following basic conventions apply in *crowd:it*:

#### **Left Mouse Click: Selecting Elements**

There are multiple methods available for selecting elements. Objects can be selected by simply clicking on them or by drawing a selection rectangle.

Regardless of the method chosen, selection behavior can be influenced by specific key combinations:

- **Without an additional key:** The current selection is cleared, and a new selection begins.
- **SHIFT key:** New elements are added to the existing selection.
- **CTRL key:** Elements are added to the selection if they are not already selected. If they are already selected, they are removed.
- **SHIFT + CTRL simultaneously:** Removes elements specifically from the selection.

### Selecting Multiple Elements

To select multiple objects at once, you can draw a rectangle around the desired objects.

- Drag the mouse **from left to right**, and only objects that are completely inside the selection rectangle will be selected.
- Drag the mouse **from right to left**, and objects will be selected even if only part of them is inside the selection area.

### Middle Mouse Click

- A **middle mouse click** on a simulation object allows you to edit its geometry.
- If the *Select and Edit Walls* tool is activated in the toolbar, the geometry of walls can be adjusted with a middle mouse click.

### Right Mouse Click

A right-click on a clickable object opens a context menu with quick editing options that vary depending on the object type.

### Left Double Click

A double-click on an object opens its properties.

## Copy and Paste

- Selected simulation objects and walls can be copied using **Right-click** > **Copy** or Ctrl+C.
- They can be pasted at another position or on a different floor using **Right-click** > **Paste** or Ctrl+V.
- The **Copy to Floor** option allows objects to be duplicated across multiple floors simultaneously.

## 2.3.4 Probability Distributions

For generating different groups of people (*personas*), probability distributions are required, for example, to assign speed, space requirements, or reaction time.

The following types of distributions are available in *crowd:it*:

- **Normal Distribution:** Values follow a Gaussian distribution (*bell curve*), meaning average values occur more frequently than extreme values.
- **Uniform Distribution:** All values within a defined range are equally likely.
- **Log-Normal Distribution:** Most values are concentrated on the left side of the distribution scale. This distribution is well suited for modeling reaction times.
- **No Distribution:** All objects receive the same fixed value.

## 2.3.5 Inserting a Background Image

A background image placed behind the scenario can sometimes be useful. Since it is unlikely that the dimensions of your image and the scenario match exactly, you can calibrate the size, position, and rotation of your image. To do this, the image and the scenario are displayed in two windows, each overlaid with a “calibration line.”

There are two ways to overlay an image onto your scenario. The faster method is to adjust the calibration line directly within the scenario. Since the image updates in real time, you can roughly calibrate it by eye. However, this method only provides a coarse calibration. If you require a precise calibration, the second method is better suited. Here, you move the ends of the calibration line (red and blue) to a “real” line or corner in the image. It is helpful if the two chosen positionVectors are far apart. Next, move the corresponding ends of

the calibration line to the same line or corner in the scenario. The goal is to match the exact same positionVector in both the image and the scenario as accurately as possible.

Finally, adjust the transparency of the image to avoid “visual competition” between the background image and the scenario. Once satisfied, select Apply.

### 2.3.6 Logo

You can add a logo to your simulation via **View > Logo....** Your logo will remain in the same position on every floor. By default, it is placed in the upper right corner.

You can resize your logo by selecting it. This will make blue rectangles appear in each corner of the logo. Hover over one of these rectangles and drag the logo to the desired size. You can also reposition your logo by selecting it and moving it with the mouse.

Once you are satisfied with the size and position of your logo, exit edit mode by clicking anywhere outside the logo.

The logo can only be edited once, as it is a project-wide setting.


# Chapter 3

## Floors

Floors are used to define the simulation environment. This chapter explains how to import, edit, and delete floors.

### 3.0.1 Insert

After creating a new simulation file, floors must be added for the agents to move on.

- Clicking on  allows you to import a .dxf file, which is then inserted as a new floor in the project.
- Clicking on the **+** icon creates an empty floor where you can draw your own geometry (see [Create Geometry](#)).

Floors must have a **unique name**. If this is not the case, a warning is displayed, and the import is not possible.

### 3.0.2 Replace

To replace an existing floor, right-click on it and select **Replace**. A file dialog will open, allowing you to choose a new .dxf file. The existing floor will then be replaced with the new file.

### 3.0.3 Viewing and Editing Properties

Double-clicking on a floor or right-clicking and selecting **Properties** allows you to edit the following settings:

- **Name:** The unique identification name of the floor.

Under **Advanced**, the following option is also available:

- **Cell Discretization:** Determines the grid size of the floor in meters.

### 3.0.4 Move

If the order of floors needs to be adjusted (e.g., when a basement is placed above another underground floor), you can move the floor to the desired position using the **six-dot symbol** (::) next to the floor name via drag-and-drop.

### 3.0.5 Delete

A floor can be deleted in two ways:

1. **Using the trash bin icon** .
2. **Via the context menu** (right-click on the floor).

If the floor contains simulation objects, a warning will be displayed before deletion.

# Chapter 4

## Geometry

### 4.1 Create Geometry

In most cases, the geometry of a scenario is imported from a .dxf file, which serves as a basis for modeling. A detailed guide on this can be found in the chapter [Importing Geometry from a CAD File](#).



Alternatively, geometry can be created directly in *crowd:it*. The **Draw Walls** tool in the toolbar is available for this purpose:



This tool allows you to draw both open and closed walls.

### 4.2 Edit Geometry

Existing geometry can be changed as follows:

- : Changes the geometry of walls
- : Changes the geometry of simulation objects


... or with a **middle mouse click** on the object.

The following steps can now be carried out:

- Select the object to be changed.
- After selection, the corner positionVectors of the object are displayed, which can be moved by dragging.
- You can also select and move several corner positionVectors.
- With **Ctrl + A** all positionVectors can be selected to move the entire object.


- The [Information panel][Info panel] also provides options to delete or add new positionVectors between existing ones positionVectors.

### 4.3 Simplify Geometry

The toolbar includes the following tool , which can be used to simplify existing geometry.

This tool performs the following simplifications:

- Removes inaccessible walls.
- Closes open walls.
- Merges redundant walls.
- Replaces walls with simpler geometric shapes.

The level of detail can be adjusted in the advanced settings. Further information can be found in the user interface of the tool via the help button .

### 4.4 Tips for Creating Geometry

To ensure that two lines of a polygon are parallel:

1. **Rotate the polygon** so that one of the parallel lines is either horizontally or vertically aligned.
2. **Check the coordinates** of the endpoints in the table on the right:
  - If the line is **horizontal**, the y coordinates of the endpoints should be identical.
  - If the line is **vertical**, the x coordinates of the endpoints should match.
3. **Adjust the coordinates** of the two positionVectors that should be parallel to the first line.
4. **Rotate the polygon back** to its original position.

# Chapter 5

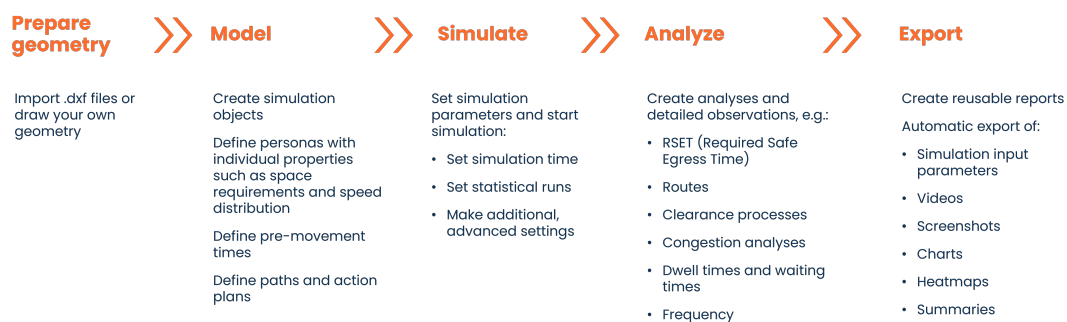
## Microscopic simulation

The *crowd:it* software uses an agent-based model to simulate crowds – the so-called *Optimal Steps Model*. Further literature on the model can be found on our website at the following link:

[Simulation model](#).

### 5.1 The microscopic modeling process step by step

The modeling process works as follows:



#### 5.1.1 Step 1: Prepare the geometry

Before modeling can begin, a viewing area must be defined and a geometry created. This can either be imported as a .dxf file or drawn directly in the software (see also [Floors](#) and [Prepare Geometry](#)).

## 5.1.2 Step 2: Modeling

In this step, all input data for the simulation is defined.

### Required input data:

- The prepared geometry or the area of interest (see also [Floors](#))
- Origins and destinations where agents enter and leave the simulation (see also [Origins](#) and [Destinations](#))
- Number and type of persons or population to be simulated (see also [Personas](#))
- Paths for controlling the agents' movement from the origin to the destination, provided that the option **Run simple assessment** has not been selected (see also [here](#))

### Optional input data:

- Action plans (see also [Paths](#)): How do the agents move through the simulation? Which stations do they pass?
- Further interaction objects on the way between source and target, e.g. waiting areas or queues
- Floor connectors such as stairs, escalators or elevators for multi-storey simulations

## 5.1.3 Step 3: Simulate

Important settings can be made in the simulation menu, e.g. simulation time or statistical runs.

After starting the simulation, the computer takes over the calculation.

## 5.1.4 Step 4: Analyze

After the simulation has finished, the results can be viewed. Various visualization and analysis tools are available for this purpose (see also [Visualize](#) and [Analyze](#)).

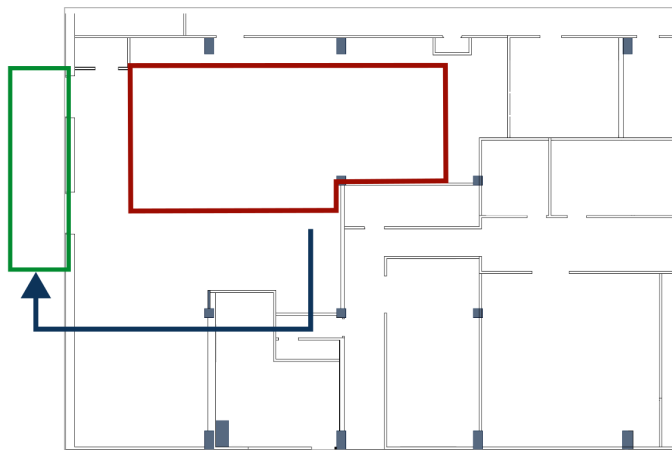
## 5.1.5 Step 5: Export results

Once the analysis is complete, all diagrams, statistics, screenshots and videos can be exported and report templates can be created (see also [Chapter Export](#)).

## 5.2 Quick intro

Our **Quick Intro**, which can be accessed via the **Welcome Screen** or the **Help** menu, introduces the basic *crowd:it* functions:

- Creating an **Origin**
- Creating a **Destination**
- Create a **Path**
- Run simulation



### Quick Intro

Create your first simulation in 5 steps!

#### Step 1: Create Origin and Agents

- Double-click on 
- Select **origin**
- Go to **Number of Agents**, enter 50
- Press **Apply**

#### Step 2: Create Destination

- Double-click on 
- Choose **destination**
- Press **Apply**

#### Step 3: Create Path

- Right-click on **Origin**
- Choose **Start Path with origin**
- Click on the **Destination**
- Choose **End Path with destination**
- Press **Apply**

#### Step 4: Run Your First Simulation

- Save your file: **File** -> **Save**
- Go to the **Simulation** menu
- Select **Run simulation**
- Press **Apply** (keep default settings)

#### Step 5: Watch you simulation results

- Press the **Play** Button 

## 5.3 Prepare Geometry

To simulate agents in an environment, a floor plan of the area to be simulated is first required. The plans should be created in CAD software and then imported into *crowd:it* or drawn directly in *crowd:it*.

### 5.3.1 Drawing Geometry yourself

[Here](#) is explained how to draw and edit geometry on floors.

## 5.3.2 Importing Geometry from a CAD File

### dxg Format

Most CAD programs allow exporting .dxg files. These can then be imported into *crowd:it*, ensuring compatibility with many different CAD programs.

#### Important!

To ensure that the exported files remain usable for the simulation, the identifiers of the objects must remain unchanged, even during multiple exports or adjustments. This is supported by most CAD programs.

Compatible and tested CAD software is listed [here](#).

Elements with a special function in the simulation (e.g. origins, destinations, stairs, etc.) can already be marked as generic “simulation objects” in the CAD plan and assigned their specific function in *crowd:it* (see [Creating Simulation Objects](#)).

These generic simulation objects can be created in several ways:

- Add the prefix “crowdit” (without hyphen) to the layer name in the .dxg file, e.g., crowdit-stairs for stairs.
- Set the color of the layer or object to RGB 232/78/15 (Hex: #e84e0f).
- Directly create simulation objects in *crowd:it* using the [Toolbar](#). You can create a general object and assign it later or select a specific object type (e.g., origin or destination) directly ( see [Creating Simulation Objects](#)).

### Allowed Element Types

*crowd:it* recognizes the following CAD element types:

- *Lines, Polylines, Circles, Arcs, Ellipses, Solids, Splines, LWPolylines, and MLines.*
- Elements within “blocks” are also imported.
- Elements like *Hatch, Text, or Image* are ignored.

### Layer Organization

CAD files are divided into layers to structure elements and floors.

For better clarity, separate layers should be created for each type of simulation object and each floor, e.g.:

floor0-crowdit-origins, floor0-crowdit-destinations.

**Note:** Frozen layers are not imported into *crowd:it*. This can be used, for example, to manage all floors of a building in a single file and only export relevant floors by freezing unneeded layers.

A sample video on layer management in AutoCAD is available [here](#).

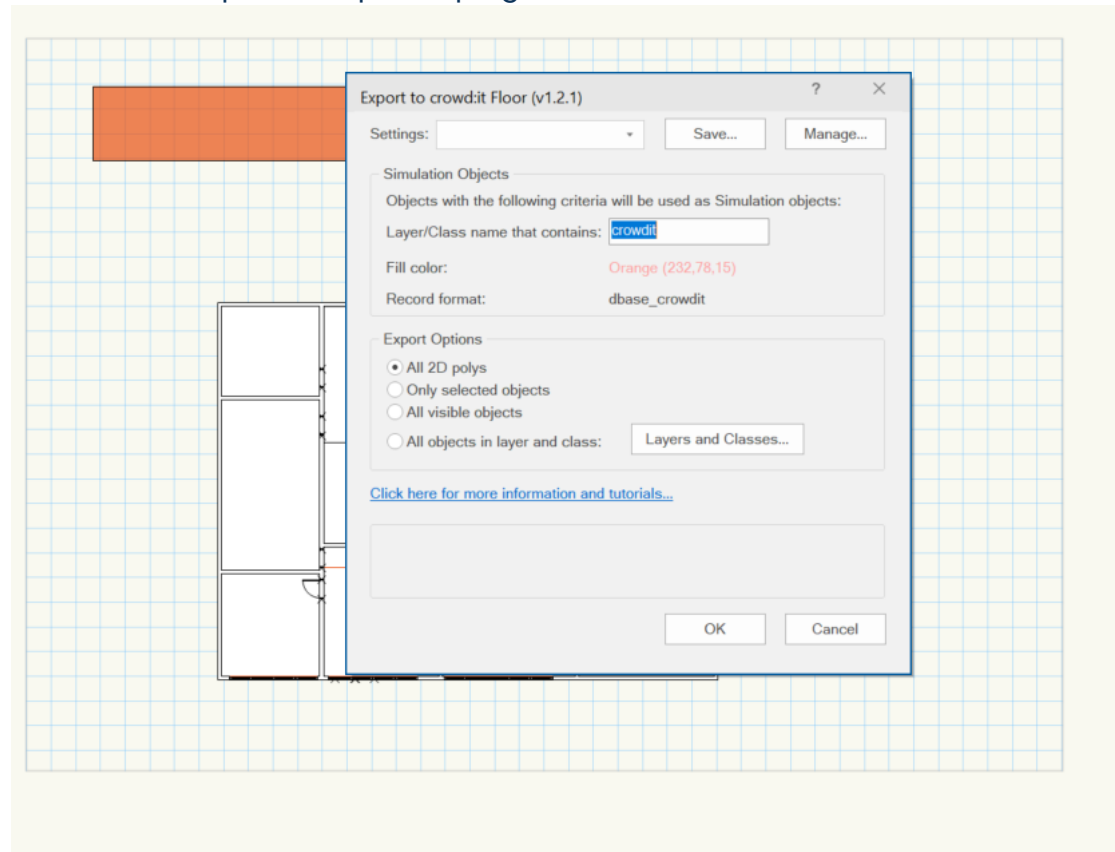
### **Check Units and Scaling**

To ensure consistent geometry and realistic simulation, plans must be to scale. Check the units before importing.

To avoid problems, it is best to move the floor plan in your CAD program to the origin. This ensures that coordinates are not too large when importing.

### **5.3.3 Preparing Geometry with Vectorworks**

Since the .dxf export from Vectorworks does not reliably work when re-importing modified floor plans, a special plugin for Vectorworks is available.



The plugin is installed via the menu in Vectorworks.

Go to **Help** and open **Install Partner Products**. Here, crowd:it can be selected

and installed.

The crowd:it plugin is automatically added to the menu under **Tools -> Partner Products -> Crowd:it**.

The plugin can also be added via **Tools - Configure Workspaces**.

Instructions for installing and using the plugin can be found [here](#).

Preparing a floor plan for simulation is similar to the approach in other CAD software.

Simulation objects can be created in the following ways:

- By adding *crowdit* to a class name.
- By using a Vectorworks database.
- By setting the color of the object (see also [Creating Simulation Objects](#)).

### Creating Simulation Objects with a Database

Using databases means drawings do not need to be reorganized, as would be necessary with class naming or color adjustment.

### Using the Plugin

The plugin adds a menu item **Tag as crowd:it simulation object**.

Select the desired geometry objects and click this menu item to automatically make all necessary database entries.

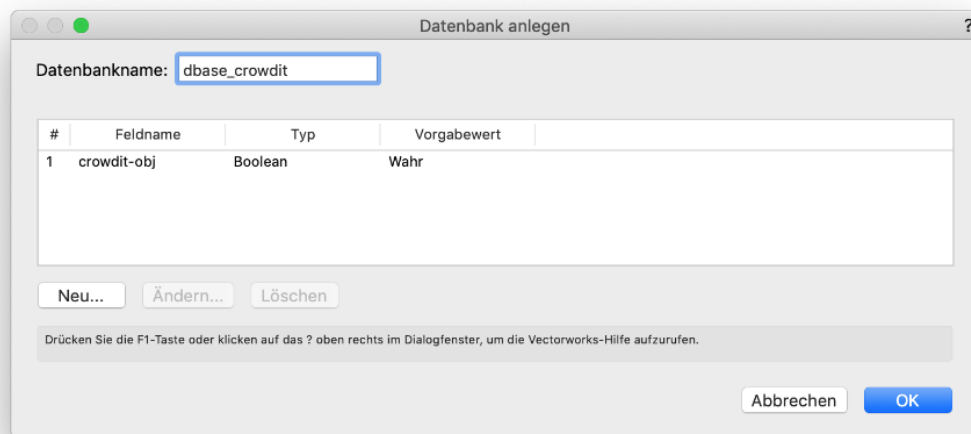
### Assigning Objects to Classes

All objects placed in the *crowdit* class are automatically exported as simulation objects.

### Manual Creation

Create a database named *dbase\_crowdit* with a boolean value *crowdit-obj* set to **true** by default.

Every object in this database is recognized as a simulation object.



### Creating a Floor Plan from a 3D Model in Vectorworks

A 3D model can also serve as a basis for a simulation in *crowd:it*, either through multi-layer export or sections.

#### **Multi-Layer Export**

In a 3D model, all floors can be exported at once. The “Level Type” and “Storey” must be entered for layers.

##### *1. Considering Building Floors and Layers*

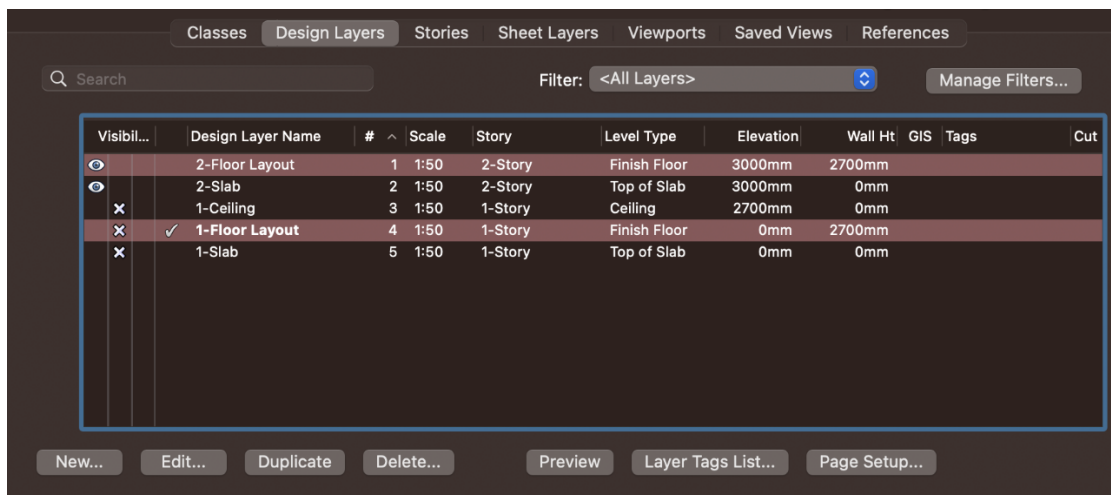
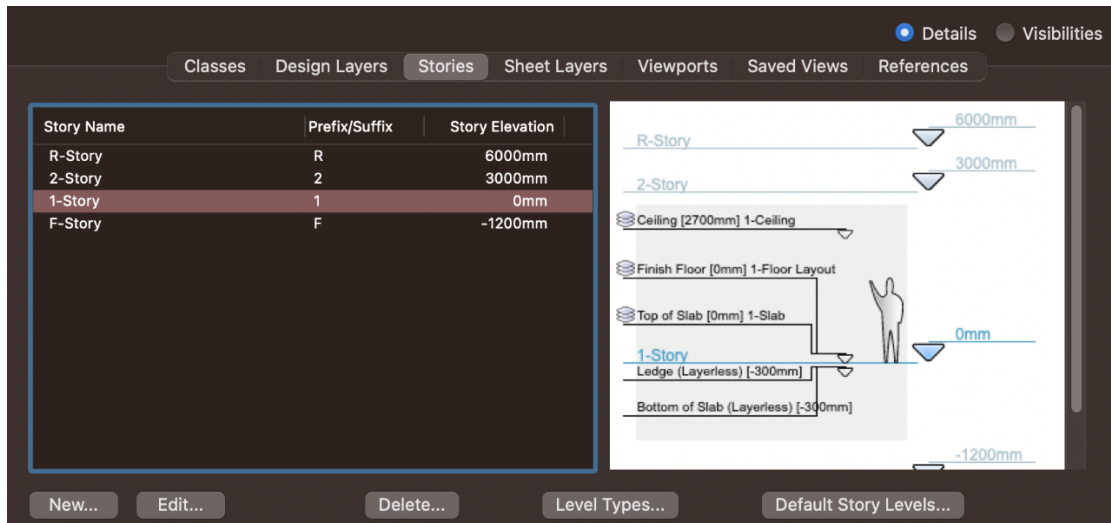
- Use storeys to define correct building floors.
- Layers should be assigned to a storey to enable precise allocation.

**Example:** All objects on layer “1-Ceiling” belong to “1-Storey” and are exported with the height of layer “1-Floor Layout” since this represents the finished floor height.

##### *2. Handling Cross-Floor Objects*

- Objects penetrating multiple floors must be matched against the exported finished floor heights using their bounding box.
- If a collision occurs with multiple floors, the object is exported multiple times.

- Doors are recognized as wall openings. Special openings must be defined as wall openings using the plugin.



Simulation objects are inserted as described above.

The multi-layer export can then be performed. Via the plugin – Export as crowd:it floor, all floors are exported at once.

### Export with Horizontal Slices

If projects were created without storeys and level types, they are not suitable for multi-storey export.

1. Use *Clip Cube* to create a horizontal slice.

2. Right-click on the section and select *Create Slice...*
3. In the new section, mark all objects and select *Modify -> Convert to Group* to convert intelligent objects to simple geometry.
4. Copy these objects into a new document. Only construction layers are exported to the *crowd:it* file.
5. Proceed as usual: create *crowdit* classes, add simulation objects, etc.

### 5.3.4 Simplify Floorplan

The cleaner a floor plan, the easier the modeling and faster the simulation runs.

Every line in the file affects computation time. Simplification is particularly useful for large simulations with many people.

After importing into *crowd:it*, geometry can be further optimized using the **Simplify Floor Geometry** tool

 (see also [Simplify Geometry](#)).

## 5.4 Creating Simulation Objects

After importing a floor plan file, all imported simulation objects are initially marked orange.

A simulation object can be assigned as follows:

- **Double click** an element opens a selection dialog.
- **Right click** an object opens a context menu, where the dialog can be initiated under **Create...**

If multiple simulation objects overlap, a menu will appear for selecting the desired object. To manage several objects simultaneously, you can select them as described [here](#).

Additional objects can be individually added by left-clicking while holding the CTRL key.

(See also [Selection Processes](#)).

The following simulation objects can be assigned:

- **Origin**: Generates agents within the simulation—either immediately or over time.
- **Destination**: Removes agents from the simulation.

- **Scaled Area:** Slows down or speeds up agents.
- **Directed Scaled Area:** Changes agent speed based on their direction.
- **Waiting Zone:** Defines areas where agents wait.
- **Stairs:** Creates stairs connecting two floors.
- **Escalator:** Creates escalators to connect two floors.
- **Elevator:** Creates elevators connecting multiple floors (available for two or more floors).

Additionally, the following simulation objects can be created from lines:

- **Queue:** Marks the starting positionVector of a queue.
- **Portal:** Connects two floors as a type of “teleporter.”

For hybrid simulations, there is also:

- **Macroscopic Areas:** Aggregates agents into flows within a specified area.

You may retroactively change the type of simulation objects by right-clicking it and selecting **Change type...** from the context menu.

### 5.4.1 Simulation objects settings

After selecting one of the above objects, a settings dialog appears. It contains the following general information:

- **Name:** A meaningful name is advisable as it is reused as an identifier in the results.
- **Description:** Space for additional information, notes, or comments.

...plus additional individual settings specific to each simulation object type. Each dialog is described in detail in the following sections.

### 5.4.2 Origin

An origin generates agents within the simulation. Various parameters must be set for this.

The dialog is divided into tabs for clarity. Typically, the first tab contains all necessary settings.

### Generation Intervals Tab

Each origin can have multiple time intervals during which agents are generated.

The following information must be provided for each interval:

- **From (s)**: Time of the first agent generation (in seconds).
- **To (s)**: Time of the last agent generation (in seconds).
- **Number of Agents**: Total number of agents generated in this interval.

### Personas Tab

Different pedestrian types (*personas*) can be defined here and their percentage share in agent generation can be determined. New *Personas* can also be created. An overview of all personas can be found on the right in the [Information area](#). Here, all existing Personas are listed and can be edited.

### Premovement Time Tab

Agents may have a premovement time – such as a delay before responding to alarms.

During this time, they only avoid other agents (see also [Waiting Behaviour][Tab Advanced Settings]).

Settings options:

- **Distribution**: Probability distribution for reaction time can be defined ( see [Probability Distributions](#)).
- **Same for All Agents**: When active, all pedestrians get the same reaction time drawn from the distribution. More info available by clicking the hint button (🔔).
- If agents should move immediately, set reaction time to 0.

### Advanced Settings Tab

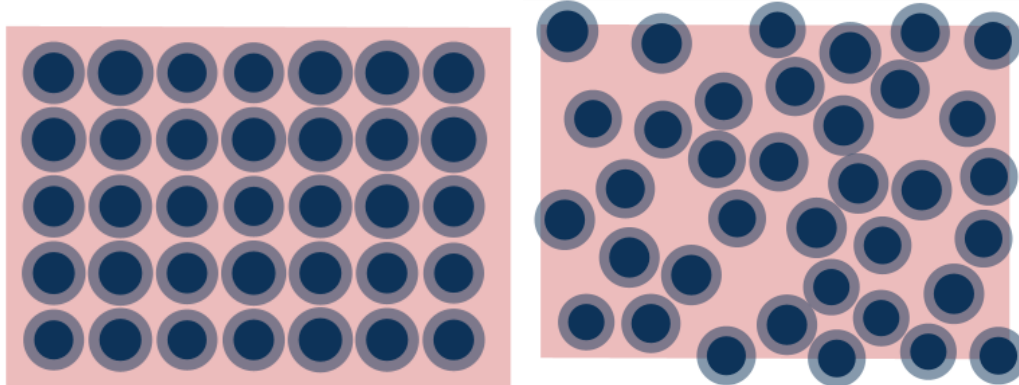
#### **Positioning**

There are two agent placement modes selectable via the **Sorted** button:

1. **Random Placement**: No fixed arrangement (default).

2. **Sorted Placement:** Agents are distributed on a regular grid evenly spaced.

*Example:*



### ***Waiting Behaviour***

Similar to Waiting Zones, various placement options for waiting agents during reaction time exist, described in [Waiting Zone][Tab Advanced Settings].

### **5.4.3 Destination**

**Destinations** remove agents from the simulation. For polygon-shaped destinations, this happens when the agents enter the area. Alternatively, it is possible to create destinations from line-shaped simulation objects. In that case, agents reach the destination when they cross the line.

### **Advanced Settings**

Optionally, custom intervals can be added (see [here](#)).

- **Enable dynamic navigation:**  
Dynamic calculation improves simulation accuracy but increases simulation duration. If you know that only certain parts of your scenario are sensitive to congestions, it can make sense to enable this option for the respective destination.

To disable dynamic navigation for all simulation objects, uncheck **Use dynamic navigation field at simulation start**.

### 5.4.4 Scaled Area

A **Scaled Area** modifies the speed of agents, e.g., to model ramps or difficult terrain.

- **Scale:** Defines how much agents are slowed down or sped up.  
*Example:* A factor of 0.7 means agents run at only 70% of their usual speed.
- **Only for navigation:** If activated, the area is considered during route selection but does not affect actual speed.  
Suitable for one-way streets or safety zones.

### 5.4.5 Directed Scaled Area

**Directed Scaled Areas** adjust agent speeds depending on their direction of movement. Useful for modeling ramps or simplified stairs (upward vs. downward).

- **Direction:** Specify the running direction as a vector with  $x$  and  $y$  coordinates.
- **Acceleration Factor:** Determine speed for movement with and against the defined direction.

To adjust the direction, use the **tool icon** (). This opens a window where:

1. Direction is set by clicking the circle in the upper half of the frame.
2. Alternatively, enter the angle of the direction vector in degrees in the lower half.

The new direction is automatically converted into  $(x/y)$  form.

The option **Only for navigation** option can also be activated for directed scaled areas. This is especially useful if you have areas that should be less attractive to agents, but agents should not be slowed down. When activated, the area is avoided but if an agent enters it, the speed is not affected.

### 5.4.6 Waiting zone

In a waiting zone, agents remain until they are allowed to leave.

The **capacity** of the waiting zone defines the maximum number of agents

allowed.

If this option is not checked, there is no capacity limit.

### **Waiting condition**

A waiting zone can be used in three ways:

1. **Waiting time:** Each agent waits for a defined duration.
2. **Schedule:** Agents are allowed to leave the waiting zone at specific times.
3. **Event:** Agents may leave the waiting zone when a specific event occurs.

While waiting, agents avoid passing pedestrians.

The avoidance behavior can be adjusted via the [Advanced Settings](#).

### **Waiting time**

Each agent is assigned a waiting time based on a predefined probability distribution ( see [Distributions](#)).

**Note:** If groups are defined, and the capacity is set to 1, the number of groups is counted, not the number of agents.  
This only works for a capacity of 1.

- **Example:** If a group of three wants to enter a waiting zone with capacity = 1, they will enter together.  
This behavior is useful for modeling queue scenarios, e.g. a family buying cinema tickets together, while three unrelated individuals buy them one after another.

### **Schedule**

Custom [Intervals](#) with variable capacities can be defined.

Agents may only enter the waiting zone during these intervals.

At the end of an interval, all agents leave the waiting zone.

This is suitable for scenarios like scheduled tours or events that start at fixed times.

### **Event**

Agents wait for a specified event before they are allowed to leave the waiting zone.

The following events are available:

- **Meeting positionVector**
- **Crowd control**
- **Passenger exchange**

#### *Meeting Point*

Agents wait until a predefined number of agents has gathered in the waiting zone, then they depart together.

The number is set via the [Capacity](#) field.

Optionally, a time limit can be defined, after which agents leave the waiting zone even if the group is not full.

This prevents agents from waiting indefinitely.

A detailed modeling description can be accessed via the help icon 

**Use cases:** Group departure for school classes or tour groups in museums, city tours, etc.

#### *Crowd control*

While waiting, agents monitor a designated area (either another [Waiting zone](#) or a [Destination](#)).

When the number of agents or the density in that area falls below a threshold, agents can walk on.

Optionally, a delay can be introduced requiring that no agents have entered the observed area for a certain time.

#### **Use cases:**

- Dynamic access control to platforms or stations to prevent overcrowding.
- Regulation of crowd size in sensitive areas (e.g. entrances at events) to avoid dangerous crowd pressure.

### *Passenger Exchange*

This event simulates a passenger exchange, e.g. at a train station.

Agents wait for an arriving train, allow others to disembark, then board the train.

The train is represented by an **Origin** (for disembarking passengers) and a **Destination** (for boarding passengers).

- The origin should cover the train cabin to ensure all exiting passengers are generated.
- The destination should be on the opposite side (away from the platform) to simulate correct boarding flow.
- Origin and destination areas may overlap geometrically.

Train schedules are defined via the **Interval** settings of the origin and destination.

The origin defines arrival timing, while the destination defines boarding time. If no intervals are defined for the destination, agents may board at any time. Optionally, a waiting period before boarding can be defined to avoid congestion.

A sample setup is available via the help icon 

### **Advanced Settings**

Advanced settings allow fine-tuning of how agents behave in the waiting zone.

**Waiting Behavior** – defines how an agent must be positioned to be considered “waiting”:

- **Fully inside:** The agent’s entire body must remain within the waiting zone.
- **Center inside (default):** Only the agent’s center positionVector must be inside the area.
- **Allowed to step outside:** Agent may step outside during avoidance movements.

**Evasion Behavior** – defines how waiting agents evade others:

- **Don’t evade passing agents:** Agent does not evade any others.
- **Allowed to step outside (default):** Agent may step outside the waiting zone when avoiding.

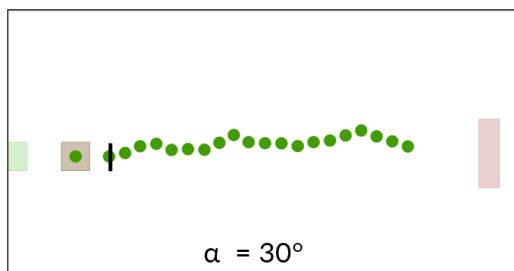
- **Evade within area:** Agent's center remains in the area while avoiding others.

### 5.4.7 Queue

Agents line up at a defined line to form a queue.

- **Sorted:** Agents queue in an orderly line. If unchecked, agents cluster behind the waiting line without structure.
- **Flip orientation:** Defines the side of the line agents queue on, indicated by an arrow in the preview.

The next element in the [Path](#) determines when agents leave the queue. Queue modeling only makes sense when followed by a capacity-limited element (e.g., [waiting zones](#) or [elevators](#)). Otherwise, agents can always leave the queue. A sorted queue looks like:



Additional adjustments:

- **Maximum queue angle deviation** can be set in [Simulation Settings](#).

More details about the queueing model can be found [here](#).

### 5.4.8 Stairs

Stairs can only be created from **rectangular objects**. If the **Stair** option is unavailable, geometry adjustments must be made either in the CAD file or using the [Geometry Tool](#).

- **Number of treads:** Adjustable within these limits:
  - **Minimum tread depth:** 0.15 m
  - **Maximum tread depth:** 0.4 m

- **Default:** 0.29 m
- **Rotate staircase:**
  - Change orientation via **Turn 90°**.
  - Arrow on staircase indicates upward direction. The agents are slowed down differently depending on whether they are going up or down the stairs. The deceleration takes place implicitly via the **Optimal Stairs Model** and is continuously validated using the latest scientific findings. The deceleration is not constant, but depends on the speed of the agents in the plane. The agents walk downstairs faster than upstairs. This is consistent with the scientific studies and the specifications from **RiMEA**. The agents are also slowed down depending on the tread depth. The smaller the tread, the more they are slowed down.
- **Connect floors:**
  - Newly created staircases do not automatically connect to other floors.
  - Select destination floor via **Connects to floor**.
- **Modeled on floor:** This shows the user where the stair resides, i.e. with which floor it is associated. This helps to avoid confusion when modeling more complex scenarios. It cannot be changed – if you want to associate the stair with another floor, you need to model it explicitly on that floor.

Once connected:

- Agents switch floors mid-stair.
- Stair becomes semi-transparent, and **Flip sides** appears to determine which floor each stair segment belongs to.

If a non-rectangular stair or a stair that can be accessed from the side is required, a **Scaled area** can be used instead.

### 5.4.9 Escalator

Like stairs, **Escalators** require rectangular objects.

- **Tread depth** is specified instead of step count:
  - **Minimum tread depth:** 0.3 m

- **Maximum tread depth:** 0.8 m
- **Default:** 0.4 m
- **Speed [m/s]:** Sets escalator speed.
- **Modeled on floor:** Description see [here](#)
- **Connects to floor:** Makes the escalator a floor connector if specified.

Adjustments:

- **Turn 90°:** Adjust step orientation.
- **Flip sides:** Specifies floor for escalator segments when connecting floors.
- **Flip travel direction:** Changes escalator movement direction. The movement direction is indicated by two arrows.

### 5.4.10 Elevator

Elevators connect multiple floors. **Boarding time** and **Capacity** can be set in the settings dialog.

- **Boarding Time:** Elevator waits after an agent enters (similar to a sensor).
  - Exception: If the elevator capacity is reached, it starts to move regardless of the entry time.
- **Floor Selection:**
  - Elevators stop on all floors by default.
  - Floors can be deselected under **Floors to stop on**.
  - The floor where the elevator was modelled can not be deselected.
- **Journey Time Matrix:**
  - Sets individual travel times between floors.
  - Include door opening/closing times for realistic simulation.
  - In the simulation, the travel time is interpreted as the **gross time** between “boarding time is over” and “first agent exits the elevator on the destination floor.”
- **Set all journey times to:**
  - Uniformly sets all journey times.

**Note:** Ensure that the elevator can reach every required floor if it is included in a path.

## Elevator Groups

If a group of elevators is to be modeled, a [queue](#) can be used to control the waiting behaviour in front of the elevators.

- Combine elevators into a [set](#).
- The **selection process** should be neutral, e.g., by using a threshold heuristic where all elevators are set to “*Keep*”.
- The functionality of selection processes is explained in detail [here](#).
- The queue ensures that agents line up and use the next available elevator.

### 5.4.11 Portal

**Portals** can either connect two floors or represent a door/barrier within a floor. They are modelled as lines. They can be used to:

- Define the **end of a floor**.
- Create a **boundary within a floor** (e.g., one-way streets, train doors, or supermarket checkouts with a fixed walking direction).

The **Connects to floor** field specifies the floor to be connected. If left empty, the portal functions, for example, as a train door or one-way street.


The **Flip orientation** option determines from which direction agents can cross the portal. If they come from the other direction, the portal acts like a wall.

### 5.4.12 Macro Area

A **macro area** aggregates agents into density flows and is suitable for areas with unidirectional traffic where **no congestion** is expected.

This can speed up calculations.

#### Creating a Macro Area

1. Select the **symbol in the toolbar** .
2. Create a **rectangular shape** (the shape can later be converted into a polygon using **Edit Geometry**).
3. A **pop-up window** will appear to enter a name and description.

### **Defining an Entry Edge**

- A **macroscopic area is unidirectional**, so an **entry edge** must be defined.
- Clicking on the **red-marked lines** changes the entry edge.
- If **no line is marked in red**, the geometry does not contain parallel lines — this must be adjusted.  
(Tips on adjusting geometry can be found [here](#)).

### **Important Aspects of Creation**

- **Entry and exit edges must be parallel.**
- The **exit edge must be at least as long as the entry edge** to avoid capacity reduction and congestion.
- The zone should be **bounded by obstacles** on all other edges.
- **No obstacles should be inside a macroscopic area**, as they would reduce the flow.
- Macroscopic areas **do not need to be directly on the agents' paths**—they function similarly to scaled areas.
- If a **persona has a wide range of speeds**, there may be deviations in clearance time, as the macroscopic area always assumes the **average of the speed distribution**.

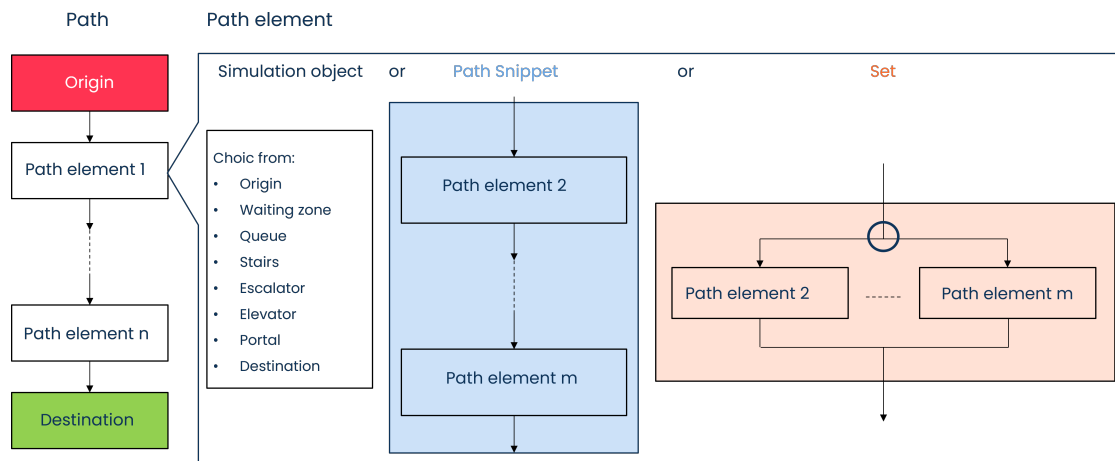
Background information on the model is summarized in [this paper](#).

## **5.5 Paths, Path Snippets, and Sets**

For agents to move in the simulation, they must be assigned **paths**. A **path** describes an agent's route from an origin to a destination and can consist of **path snippets**, **sets**, and **simulation objects**.

- **Path Snippets** are predefined route segments that can be used by multiple paths.
- **Sets** are groups of simulation objects, path snippets, or other sets, from which one is selected either randomly or based on rules.

The following diagram illustrates this structure:



The following sections provide a detailed description of the individual elements and their associated [selection processes](#).

### 5.5.1 Paths

A **path** defines an agent's route through various simulation objects.

- Each path **starts at an origin** and **ends at a destination**.
- It may include **any number of intermediate elements**, such as:
  - Waiting zones
  - Queues
  - Stairs or escalators
  - Elevators
  - Portals
  - (Intermediate) destinations
  - Origins

**Important:** An agent without an assigned path remains at its origin for the entire simulation.

#### Composition of a Path

- **Direct Path:** A connection from origin to destination without stops.
- **Complex Paths:** Contain intermediate destinations, sets, or path snippets.

Difference between path snippets and sets:

- Path Snippets (see [Path snippets](#)) contain multiple simulation objects in a fixed order (*AND-connection*).
- Sets (see [Sets](#)) contain multiple simulation objects, from which one is selected (*OR-connection*).

**Note:** Path snippets can contain sets and path snippets alongside simulation objects, and sets can also contain path snippets and other sets – enabling the modeling of complex scenarios.

Each path is assigned a **probability**:


- If only **one path** starts from an origin -> **100%** of agents use it.
- If **multiple paths** exist -> Ratios must sum to **100%**. This is set via the [Path Table](#).

### Key Rules:

- Paths **must start at an origin** and **end at a destination**.
- If a path starts with a set, the set **must contain only origins**.
- If a path ends with a set, the set **must contain only destinations**.
- A [queue](#) must be followed by an element (simulation object, set, path snippet) with capacity constraints to allow agents to queue and wait for availability.

## Creating a Path

### *In Floorplan*

1. Select the **Path Tool** in the toolbar .
2. **Left-click on an origin:** The path begins.
3. Add additional elements:
  - **Left-click on the object** and select **Add to path...**
  - If the object is part of a set or path snippet, you can add **the object, the set, or the path snippet**.
4. End the path at a destination and select **End path with...**

### *In the Paths tab*

1. Make sure the "Path" tab is open in the right-hand information panel.

2. Enter a unique name in the field “*Create new path*” at the bottom.
3. Click on the Plus button or press Enter.
4. Now you may add elements to the path, see [Editing a Path](#).

Alternatively, you can duplicate an existing path by right-clicking on it and selecting **Duplicate**.

### **Path Ratio**

Upon completion, a path ratio can be set. The default is 100%.

If multiple paths originate from an origin, the **sum of probabilities must be 100%**. This can be adjusted in the [Path Table](#).




### **Multi-Floor Connections**

If a path spans multiple floors, agents can:

- **Explicitly use stairs, escalators, or elevators** by adding them to the path.
- **Use automatic floor connectors:** The system selects the **nearest connection element** to the destination. This setting is configured in the [Simulation Menu](#).

### **Editing a Path**

The **Paths** tab in the right-hand information panel provides an overview of all paths. Clicking on a path opens an editing panel below with multiple tabs.

- **Members:** Move elements within the path by dragging them over . Use the search field to find sets, path snippets, and simulation objects and add them with . Select elements and click  to remove them.
- **Settings:** Rename the path or open the [Path Table](#).

### **Path Table**

The **Path Table** provides a tabular overview and allows:

- Editing all paths in text format.
- Filtering for specific elements.
- Adjusting path probabilities.

**Note:** It is recommended to adjust only probabilities in the Path Table and make other changes through the [Edit Path](#) section.

Each column contains:


1. **ID:** Unique and descriptive name.
2. **Ratio [%]:** Must sum to 100% per origin.
3. **Members:** List of elements traversed.

## 5.5.2 Path snippets

A **path snippet** is an ordered sequence of waypoints that can be reused across multiple paths.

- **Difference from a Set:**
  - A **path snippet** is traversed in a fixed order (*AND-connection*).
  - A **set** selects one of its elements randomly or based on rules (*OR-connection*).

**Creation:**

- Use the **Path snippet Tool** .
- Add elements by left-clicking.

All path snippets are listed in the **Path snippets** tab in the right-hand panel. Here you may also create a new, empty path snippet. Clicking a path snippet opens an editing area below with tabs:

- **Members:** Identical to path editing (see [here](#)).
- **Probabilities:** Set individual probabilities for path snippet elements.
- **Settings:**
  - **Capacity:** Limits the number of agents assigned to the path snippet. Default is unlimited (*-1* in the table).
  - **Order:**
    - \* **In order:** Agents follow the elements sequentially.
    - \* **Permute:** Agents traverse elements in a random order.
- **References:** Shows which paths and sets use the path snippet.

Alternatively, you can duplicate an existing path snippet by right-clicking on it and selecting **Duplicate**.

### 5.5.3 Sets

A **set** is a group of simulation objects, from which one is chosen for the path. Sets can combine simulation objects, sets, or path snippets, allowing complex nesting – one element is then selected based on the [selection process](#).

Applications:

- Agents choose between different route options (e.g different destinations or waiting zones).
- Agents can select among multiple path snippets.
- Multiple path options merge into one path.

#### Creating a Set

Right-click on a simulation object and select **Set: Add...** In the dialog, create a new set by entering a unique name under **Create New Set** and clicking **+**. Alternatively, you can duplicate an existing set by right-clicking on it and selecting **Duplicate**.

#### Add Elements to a Set

##### Context Menu (Right-click on Object):

- **Case 1:** Object is not yet in any set -> **Add to set...**
- **Case 2:** Object is part of a path snippet -> **Set: Add...**, then choose between object or path snippet.
- **Case 3:** Object is part of a set -> **Set: Add...**, then choose between object or parent set.

##### Via the "Sets" Tab in the [Information Panel](#) on the right:

If no **Set** is selected, there is an option at the bottom to **Create new set**. Enter a unique name into the field and click the plus icon **+** to create an empty set, which will then be displayed in the overview. Elements can be added by clicking on the set.

#### Edit Sets

Through the **Sets** tab in the [Information Panel](#) on the right side of the user interface, you can view all Sets created in the simulation. A left click on a Set will display various tabs at the bottom, including:

- **Members:** The *Members* tab is identical to path editing (see also [here](#)) and allows adding and removing elements from the selected set.
- **Settings:** Allows you to change the name and description of the set.
- **Selection Processes:** Lists all paths, path snippets, and sets where the selected path snippet appears directly or indirectly. A detailed description of the available heuristics can be found [here](#).
- **References:** Lists all paths, path snippets, and sets where the selected set appears directly or indirectly.

For a hierarchical representation of the set, see [here](#).

### 5.5.4 Selection Processes

When an agent reaches the last path element before a set, they choose an element from that set as the next destination. A heuristic determines how available elements are selected.

Available heuristics include:

- **Closest:** Selects the spatially nearest element.
- **Farthest:** Selects the spatially farthest element.
- **Shortest queue lane:** Selects the queue with the fewest agents. If multiple queues have the same number of agents, a random queue is chosen.
- **Shortest and closest queue:** Selects the shortest and nearest queue.
- **Distributed:** Selects all elements equally.
- **Distributed and empty:** Selects only empty elements (e.g., unoccupied waiting zones) equally.
- **Less crowded:** Initially selects the nearest element; if the person density exceeds 1.2 persons/m<sup>2</sup>, a farther element is chosen.

### Creating Custom Heuristics

By clicking on *"Edit"*, you can define custom selection processes for the elements of a set.

A new window appears where you can define the selection mode and selection processes.





The following relationships exist:

- A selection mode determines how the elements of a set are chosen.

- A selection process is a combination of selection modes applied sequentially.
- Multiple selection processes can be defined for a set, each active during a specific time frame. The time frames cannot overlap, meaning only one selection process is active at any given time.

### Selection Processes

The following actions are available:

- **Add Selection Process:** Enter a name in the field “*Create New Selection Process*” and press **+** to create a new selection process.
- **Delete Selection Process:** To delete a selection process, select a process and press .
- **Duplicate Selection Process:** To duplicate a selection process, select a process and click .
- **Rearrange Selection Processes:** Clicking  opens a window where processes can be reordered. Note: The times (“*Active Until*”) of the selection processes remain unchanged; only the processes themselves are rearranged. To return to edit mode, click the wrench icon .

For each selection process, the following can be set:

- **Name:** Double-click in the table field to change the name.
- **Active until:** Specifies the time until which the selection process is valid. If only one selection process is defined, this field defaults to “End of Simulation” and cannot be changed. Additional processes enable time adjustments. The processes must cover the entire simulation period; otherwise, agents may stop moving when the process expires.


### Selection Mode

To define selection modes during a selection process, first left-click on the table row.

All defined modes appear as a list at the bottom.

#### Delete a Selection Mode

Click  to remove a mode from the list.

**Reorder Selection Modes** Click the double arrow icon at the bottom left  to open a new window where selection modes can be sorted. Click the six dots

⋮ next to a mode name to move it. Click the wrench icon  to return to edit mode.

### Add a Selection Mode

The following selection modes are available:

- **Best:** Selects the best element based on specific criteria.
- **Demand:** Selects the element with the lowest demand value.
- **Threshold:** Selects elements that do not exceed a specified threshold.

To add a mode to the list, press the corresponding button.

#### **Selection Mode: Best**

The **Best** mode selects the “best” element based on chosen criteria and discards all others, typically leaving only one best element.

Available options:

- **Number of agents:** Selects the element with the fewest assigned agents.
- **Distance:** Selects the nearest element. Choose “Farthest” to select the farthest element instead. Three measurement methods are available:
  - Linear distance: Measures the distance to each set element (default).
  - Walking Time: Evaluates how long an agent needs to reach each element (e.g., to include scaled areas).
  - Distance to projection on object: First calculates the projection of each set element to the object’s outer edge where the agent is located, then measures the distance from the agent to these projection positionVectors. This is useful when target elements are unevenly spaced and agents should pick the one in their movement direction.
- **Delay Time:** Selects the element that delays the agent the least.
- **Priority:** Assigns a priority/order to each element. In “Best” mode, elements with the lowest priority are chosen. For example, elements with priority 1 are preferred over those with priority 2.

#### **Selection Mode: Demand**

The **Demand mode** operates similarly to the **Best** mode: it selects a single element that is the best according to the specified criteria. However, the demand mode allows these criteria to be set individually for each element. To

compare the elements, a demand value is determined for each member. The demand value represents the distance to the defined threshold of the element. The element with the lowest demand is selected. *Right-click on a member and select a demand type from the popup to define the criteria by which the demand should be determined.*

The following demand types are available:

- **Default:** Elements in the standard category are either discarded or not discarded by the mode.
- **Number of Agents:** Elements in the demand category are evaluated based on the number of agents. The demand of an element reaches 100% once the number of agents assigned to the element, as specified in the "Threshold" column, is reached, and it is then discarded.
  - "At the same time" / "Total": This option allows you to specify whether only the agents currently assigned to the element (at the same time) or all agents who have ever visited it (total) should be counted for the demand.
  - "Absolute" / "Relative": With the "Absolute" option, the number of agents is specified directly. With "Relative," the mode is calculated as a percentage.
- **Delay Time:** The demand reaches 100% once the agent is delayed by the element for the time specified in the "Threshold" column.
- **Density:** The demand reaches 100% when the current person density in the element (in agents per square meter) reaches the value specified in the "Threshold" column.

#### **Selection Mode: Threshold**

Unlike the Best and Demand modes, the threshold mode does not compare elements with each other but against a threshold value. Each element can be assigned a category and a threshold. The mode then discards all members that exceed their threshold. As a result, multiple elements may remain, which can then be selected through another selection process or randomly. The same categories are available as in demand mode.

Additionally, there is the "Distance" category. By default, with the setting "nearest," it discards all elements that are farther than the specified threshold. Select

“farthest” to choose elements that are farther away than the given threshold while discarding the nearer ones.

The threshold mode typically discards all elements that exceed their assigned threshold and returns the remaining elements. However, if all elements exceed the threshold, no element would be returned. Therefore, you can specify what should happen in this case. By default, the mode simply returns the element closest to the threshold, even if it exceeds it. Other options include selecting all elements (and letting the next mode decide which element to choose) or discarding all elements (causing the agent to be unable to select a valid element and thus stop moving).

*Note:* Some categories may be missing if the set contains members for which the category is not applicable.

### **Selecting Multiple Best Elements**

The “Best” and “Demand” modes offer the option to choose “Choose the best N elements” at the bottom of the window. Normally, these two modes select a single “best” element based on specific criteria. However, sometimes it is beneficial to select multiple best elements, such as the three closest ones. Use this setting to specify the number of best elements to be selected. Select “more...” in the dropdown list to enter any number greater than zero.

### **Agents to Be Considered**

All three selection modes offer the option to measure the number of agents assigned to an element. Use the “Consider ...” section to set the condition specifying which agents should be considered. By default, all agents currently assigned to an element are considered. Other options include considering only agents who have reached an element or only those currently approaching an element. The “Time Period” setting in “Demand” and “Threshold” modes must be set to “at the same time” to enable the “Consider...” settings.

## **5.5.5 Open Hierarchy Window**

In the tabs [Paths](#), [Path snippets](#), and [Sets](#) in the [Information Panel](#), you have the option to open an additional window with a hierarchical view via the context menu (“*Open hierarchy window...*”). This can be used to compare your paths, path snippets, or sets if they are long and complex.

## 5.6 Population Settings

In the right [information panel](#) under the **Settings** tab, at the bottom, you'll find the **Personas\*\*** and **Groups** tabs, where you can make population settings.

### 5.6.1 Personas

Within the simulation, it is important to model the correct agent population. To this end, the modeler must create one or more personas. Each persona describes a group of people with similar characteristics. Classic personas could be, for example, elderly people or students.

In the persona settings dialog, personas can be edited, created, and removed. In crowd:it, a so-called **DEFAULT** persona is preconfigured, which is based on values from the literature as an average population.

The values were derived from [Weidmann](#) as well as from the [RiMEA Guidelines](#). After creating the personas, the corresponding persona distribution can be set for each [origin](#).

A persona summarizes following characteristics of people, which will be explained below.

#### Torso diameter

The torso diameter describes the space requirements of a pedestrian. Each pedestrian is modeled in crowd:it as a circle. To specify the required area, the diameter must therefore be known.

To ensure that not all pedestrians within a persona are the same size, an appropriate size distribution can be used. A more detailed description of the distributions can be found [here](#).

#### Speed

Speed describes the walking speed of a pedestrian on level ground. It represents the speed of the person without the influence of obstacles, other pedestrians, slopes, etc.

To ensure that not all pedestrians within a persona walk at exactly the same speed, an appropriate distribution can be used. A more detailed description of the distributions can be found [here](#).

## Behavior

- **Accessible simulation objects:** This allows various types of simulation objects to be excluded for a persona. If a simulation type de-selected, agents with this persona will not use simulation objects of this type.
- **Social distancing:** The COVID-19 pandemic has prompted us to incorporate social distancing into our simulation model. Therefore, we have integrated a social distance model that keeps agents a minimum distance from one another, if possible. The value can be set between 0 and 2.5 meters. More information about this model can be found at [the following link](#). In our implementation, agents try to maintain distance and only violate social distancing if it gets too crowded or a bottleneck has to be passed. If no social distancing should be maintained, the value can be set to 0.

### 5.6.2 Groups

Here, we find the distribution of the groups:

- **Size** shows the size of the group
- **Ratio** describes the proportion of agents in a group of that size. The ratios must sum to 1.0.

## 5.7 Simulation Calculation

### 5.7.1 Simulation Menu

Under the **Simulate** menu, the following options are available:

As long as no simulation results are available, the following functions can be used:

- **Check simulation:** Opens a window with analysis results from plausibility checks. This check is particularly helpful for complex simulations to avoid fundamental errors.
- **Run simulation...:** Starts a simulation. A window opens with configuration options, which are described [here](#).
- **Run simple assessment...:** Performs a simplified simulation without paths (details see [here](#)).

- **Custom kernel...:** Allows selecting a specific simulation kernel. This option should only be changed in consultation with an accurate employee if special simulation requirements exist.

If simulation results are already available, additional options appear:

- **Delete Results:** Removes the current simulation results.
- If **statistical runs** (see also [here](#)) have been performed, you can switch between different simulation runs here to load the respective results.

## 5.7.2 Start Simulation

To start a simulation, click **Simulate > Run simulation...** A window opens where you can configure the simulation settings.

### General Settings

- **Simulation duration:** Specifies the total duration of the simulation in the time format hh:mm:ss. The simulation ends when either
  - all agents have reached their destination,
  - the specified time has elapsed, or
  - you select *Cancel*.
- **Number of simulations to run:** Specifies how many times the simulation should be executed. This allows statistical runs to be started, each performed with different starting conditions (see also [here](#)).
- **Agents to be generated [in %]:** Defines how many percent of the agents specified in sources will be simulated. This setting is particularly useful for test simulations to verify the modeling.

### Optional Settings

- **Use dynamic navigation field:** Activates an algorithm that makes the simulation more realistic but significantly increases computation time (default setting: Disabled). If dynamic flooding is enabled in the settings of individual [destinations](#), it is applied for them regardless of the general setting.
- **Automatically find floor connectors:**

- Floor connectors such as [stairs](#), [\[escalators\]](#)[\[Escalators\]](#), and [elevators](#) can be placed on a [path](#) to allow agents to use the respective connection element.
- If an agent's next destination is on a different floor without an explicitly defined floor connector, the agent will automatically choose the nearest connector.
- The object chosen is always the one that leads to the destination via the shortest global path.
  - \* Even stairs and escalators that are not floor connectors are automatically accessed by agents.
  - \* In rare cases, the automatic selection of floor connectors may lead to undesirable results.

### Advanced Settings

#### *System settings*

- **Maximum memory [GB]:** Defines the maximum usable RAM for the simulation.
  - By default, a maximum of 3/5 of the available memory or 100 GB is used – whichever is smaller.
  - The value should be specified as a multiple of 2<sup>x</sup>.
- **Compress output:** Saves simulation results in a zip archive to save storage space. This is enabled by default.

#### *Model settings*

- **Update rate of navigation field [Simulation Steps]:** Specifies after how many simulation steps the navigation field is updated. By default, updating occurs after each step.
- **Distance between two evaluation positionVectors on OSM Circle [m]:**
  - Determines the distance between two positionVectors on the OSM circle disk, which are evaluated as the next possible position for an agent.
  - This value should only be adjusted if agents get stuck in narrow seating rows, for example, as reducing it increases computational effort.

- **Default value: 0.15 m.**
- **Maximum queue angle deviation [°]:**
  - Defines the largest angle within which a queue can form around an object (**0–180°**).
  - **Default value: 30°.**
- **Cancel simulation automatically:**
  - Ends the simulation automatically if **unresolvable traffic jams** occur.
  - **Default value: Disabled.**

Once all settings are configured, click **Apply** to start the simulation. A progress bar indicates the current status.

- Under **Details...**, additional information about the simulation progress can be viewed.
- With **Cancel**, the simulation can be stopped at any time.
- After completion, the finished simulation results will open automatically.

**Tip:** While the simulation is running, you can open a second instance of *crowd:it* to see the first results.

### 5.7.3 Run simple assessment

As an alternative to the full simulation, a **simplified assessment** can be performed.

- **No paths** need to be created.
- Agents automatically choose the shortest route to the nearest destination.
- Only **origins, destinations, and floor connectors** need to be defined, but not connected – agents will independently find the best path, even across multiple floors.

### 5.7.4 Further Editing

After executing a simulation, simulation objects and paths can no longer be edited.

- Subsequent editing would lead to a discrepancy between the simulation results and the original geometry.
- If changes are necessary, a warning will appear with the following options:
  1. **Delete results** – Deletes all results and allows editing the file.
  2. **Copy project** – Creates a new project file without simulation results.
  3. **Cancel** – Cancels the process.
- Regardless, simulation results can be removed at any time under **Simulation > Delete results**.

## 5.8 Visualize

In the **Visualize** menu, you can customize the colors for both the simulation results and the display of objects and geometry.

### 5.8.1 Agents

When selecting **Agents...**, a dialog appears that allows you to adjust the color of agents and their movement paths. The reset button restores the default values. Changes are only applied after clicking **Apply**.

#### Agent Color

Agents can be colored based on specific characteristics, including:

- **Origin**
- **Destination**
- **Path**
- **Persona**
- **Navigation State:** Colors the agents based on their current navigation state, i.e. walking, waiting, on stairs, etc. This is particularly useful for debugging and analyzing the behaviour of agents.
- **Next Destination:** Colors agents according to the next simulation object they will reach.
- **Congestion:** Green if not in congestion, red if in congestion.
- **Velocity:** The faster the agent, the brighter its color.
- **Social Distancing:** Colors agents based on their proximity to others (appears only if [Social Distancing](#) is activated)

- **Group:** Displays lines connecting each individual within the group.
- **Group Size:** Represents *group size*, not the group itself.
- **Time of Exit:** The earlier an agent leaves the simulation, the brighter its color.
- **Custom:** Choose a custom color to color the agents.

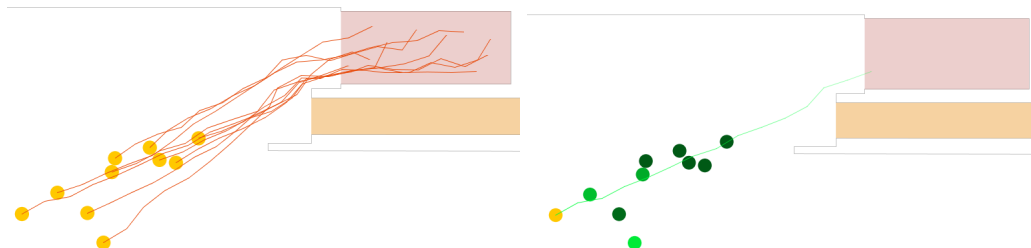
A legend in the lower-left corner may appear to explain the color coding.

### Agent Traces

To assist your analysis, movement paths of agents can be visualized in real-time. Enable **Draw agent traces** to display them.

The following display options are available:

- **Color:** Choose whether movement path colors should match the agent's color or define a custom color for paths.
- **Style:** Select **Fading** to allow movement paths to fade over time. Otherwise, the full path remains visible.
- **For:** Choose **Active** to draw movement paths only for agents still active in the scenario. If you have selected specific agents via [Identify Agents](#), the option **Selected** will be enabled.



## 5.8.2 Display Evaluations

Under **Visualize > Show Evaluations...**, you can toggle evaluation objects on or off.

## 5.8.3 Colors

You can adjust the color palette of your simulation under **Visualize > Set Colors...**. This allows you to change the colors of simulation objects, simulation object types, the background, walls, and evaluation objects. To change the color of

individual elements, enter the element name in the provided field. Additionally, all elements can be hidden similarly to CAD programs.

Attention! Hidden simulation objects are not displayed but still affect the simulation.

You can also change the color of a simulation object by right-clicking on it and selecting **Set fill color...**. To change the outline, you can select **Set outline color...**

### 5.8.4 Object Labels

With **Visualize > Object label...**, a dialog window opens where you can enable or disable labels. These labels apply to all object classes, from walls and unassigned simulation objects to origins and stairs.

### 5.8.5 Set Start Time

Under **Visualize > Set start time...**, you can define a start time in the format hh:mm:ss. This setting is reflected in both evaluations and the playback timeline. This feature is especially useful for process simulations, such as simulating a train station with a timetable.

**Attention:** If you set the start time, the times of the time-dependent [Selection processes](#) must also be adjusted.

### 5.8.6 Playback Speed

With **Visualize > Set playback speed...**, you can accelerate or slow down the simulation runtime.

## 5.9 Analyze

Once your simulation is complete, and any visualisation settings defined, you can evaluate your results.

### 5.9.1 Agents

In *crowd:it* it is possible to analyze the behaviour of individuals and groups of individuals. The following menu entries describe how.

## Identify Agents

In order to consider individual agents and their personal details, select the **Identify agents** tool from the toolbar and click on any agent you wish to know more about.



A table in the right panel displays the key data of the agent and a trace displays the path so far. To identify multiple agents, drag a rectangle over the agents you wish to highlight, or hold the Ctrl-Key and individually select additional agents. De-selection is similarly available. When selecting multiple agents, three buttons display underneath the agents' key data: <-, Statistics and >-. To navigate between each agent's key data use the <- and >- buttons. To display statistics about all selected agents, select **Statistics**.

## Search Agents

If you are interested in a particular agent, and know its ID, use **Search Agent...** to locate it and display its details and Path as in [Identify Agents](#).

### 5.9.2 Evaluation objects

To evaluate areas of your scenario, *crowd:it* provides various evaluation objects, as described in the following menu entries.

#### Tripwire

After drawing, a dialog will appear with the following options:

- **Name:** The name of the counting line, which will be displayed in the legend and in the charts.
- **Color:** The color of the counting line. This is helpful for quickly identifying the counting line in the plan.
- **Count each agent only once:** This option is selected by default. You can deselect it if, for example, you want to count the total number of agents in a process simulation, even if an agent has crossed this line multiple times.
- **Unidirectional:** You can choose to count only agents moving in a specific direction across the counting line, or you can count in both directions. If

you select this option, you can adjust the direction in the lower window by clicking **Reverse orientation**. An arrow will appear on the counting line to indicate the direction in which agents will be counted.

Once the counting line is created, the following options are available via a right-click:

- **Properties...:** This will open the dialog described above so you can modify settings.
- **Summary...:** Opens an exportable table summarizing key positionVectors, such as the number of agents that have crossed the counting line and the times when the first and last agent crossed the line.
- **Chart...:** Here you can display a chart directly. The available charts are described below.
  - **Number of agents:** The number of individual agents that pass the counting line and at what time.
  - **Speed:** Provides statistics on the speed of agents when they cross the counting line (minimum, mean, median, maximum).
  - **Evacuation time curve:** Displays the number of agents crossing the counting line over time.
  - **Weidmann (Speed/Flux):** This chart plots the Weidmann curve as well as the fundamental chart for speed and flow based on agents crossing the counting line.
  - Other custom charts can also be created. This is done via **Analyze > Create chart...** and is described [here](#).
- **Export data as .csv:** This allows you to export the raw data for further use in .csv format.
- **Copy to floor...:** This allows you to copy the counting line to other floors. After applying, new simulation objects will be created on the selected floors. These objects will have the same properties as the original simulation object.
- **Edit geometry:** Move the counting line using the blue circles at both ends. Once you are satisfied with the position of the counting line, click anywhere outside the line to end the action.
- **Delete:** Deletes the counting line.

## Rectangle

With the **Draw evaluation rectangle** tool  you can draw a rectangular box on the plan. As with the **Tripwire**, you can assign a name and color.


A right-click will show a context menu, as described here.

In addition to the predefined charts available for counting lines, rectangles have the following:

- Base charts:
  - **Local density**: Standard density calculations (agents per square meter) within the rectangle area.
  - **Local Voronoi density**: Voronoi density calculations within the rectangle area. Caution: this value may be inaccurate if the evaluation area is too small or lies over obstacles.
  - **Time spent in area**: Creates a data positionVector each time an agent enters the rectangle and records the duration of each visit.
  - **Aggregated time spent in area**: Provides a data positionVector per agent and aggregates the time they spent inside the rectangle.
    - \* **Maximum individual congestion duration**: This counts how long each agent is in congestion within the area. The criteria for when an agent is considered to be in congestion can be adjusted in the **Evaluations** tab next to the floor plan view. A more detailed description can be found [\[here\]](#)[\[Congestion\]](#).
    - \* **Agents in congestion over time**: Displays the number of agents in congestion at a given time.
- Weidmann charts:
  - **Weidmann (Density/Flux)**
  - **Weidmann (Density/Speed)**

Additional custom charts can also be created. This is done via **Analyze > Create chart...** and is described [here](#).

## Polygon

With the **Draw evaluation polygon** tool  you can create a closed shape that behaves similarly to the evaluation rectangles, but is not rectangular. When creating your polygon, you can add as many vertices as needed. The Del key

will remove the last added vertex. To close the polygon, click near the first positionVector or press Enter.

A right-click will show a context menu, as described here. The same charts are available as for the [Rectangle](#). The creation of custom charts is described [here](#).

**Warning:** Polygons must be convex; otherwise, no evaluations will be calculated. (A convex polygon does not contain any inward-facing corners.)

### 5.9.3 Evaluation Settings

Some evaluations can be more precisely adjusted by the user. For example, criteria for detecting congestion can be defined by the user. These settings apply to all evaluations of this type. All globally valid settings can be set in [Info Panel][Info Panel] area under the **Evaluations** tab. The tab is only visible if simulation results are present.

#### Congestion Settings

The second section contains all settings related to congestion evaluation:

##### Max speed limit...

- **... in the plane [m/s]:** Describes the maximum speed below which a pedestrian is considered to be in congestion.
- **... on stairs [m/s]:** Describes the maximum speed below which a pedestrian is considered to be in congestion when moving on stairs.
- **... in relation to free flow speed [%]**
- **Min. time threshold [hh:mm:ss]:** Describes the minimum time an agent must stay below the speed limits before being considered to be in congestion.

Note that an agent's speed must fall below both the absolute and relative thresholds to be considered in congestion.

Agents that have not yet moved (see also [premovement time](#)), i.e. whose pre-movement time has not yet expired, are not taken into account in congestion detection.

### **Social Distancing Settings (only visible if social distancing is enabled)**

Under **Analyze**→**Social Distancing...** settings for the evaluation of social distancing can be found:

**Social distance [m]:** Indicates the distance (center to center) at which an agent is considered to be violating social distancing behavior.

**Exposure time threshold for calculation [hh:mm:ss]:** Indicates the number of seconds that an agent must violate social distancing behavior before it is considered a violation of the rule.

**Exposure time threshold for visualisation [hh:mm:ss]:** Sets the time threshold after which the circles around the pedestrians turn red.

**Exclude agents on same path:** If you want to exclude agents who are on the same path, you can check the option below. All agents assigned to the same path will be ignored in the calculation of congestion.

## **5.10 Result Presentation**

### **5.10.1 Charts**

Use charts to quickly and clearly present the results of your simulation. You can select a pre-defined chart (see [Tripwire](#), [Rectangle](#) or [Polygon](#)) or create a custom chart.

For each simulation object, pre-defined charts are available via a right-click on the object. The appearing context menu offers **Chart...** Here, some most common charts are offered. If you want to create a custom chart, check [here](#) for more information.

#### **Creating a Chart**

Use **Analyze** > **Create chart...** to create a custom chart. A dialog will then appear to guide you through the process.

In the first step, you must decide whether to create a chart from a template for a specific area type or a completely custom chart. If you choose a custom chart, you will be directed to the customization dialog (see next chapter). Under chart templates, you can select a template that matches your area type and the area(s) to be analyzed.

## Chart Customization

In the customization dialog, various options are available to tailor your chart:

- **Select chart type:** Choose from scatter plot, bar chart, or line chart.
- **Select data:** Choose from the available data series.
- **Refine data:** Apply functions such as min, average, max, or median to your data series.
- **Design data:** Customize the data series (color, name, icons, etc.).
- **Design chart:** Set the layout and title of the chart.

Clicking **Calculate** in the right-hand window shows how your selection affects the result. This allows you to see immediately whether the chart meets your expectations. It is recommended to use the **Calculate** button frequently, as there is no reset button. Frequent calculations make it easier to manually revert changes if needed.

You can switch between different configuration levels by clicking the desired field on the left side of the window. To finalize and display the chart, click the checkmark button.

### **Select chart Type**

In the first configuration field, choose the type of chart you want to create.

Available options:

- **Line, Scatter, and Mixed:** Best for displaying numerical data series. You can choose between line, positionVector, and mixed charts. Line charts connect all data positionVectors with a line, while positionVector charts display each data positionVector as an individual small circle. The mixed option allows combining line and positionVector data series.
- **Area:** Best for representing numerical data series. Choose between standard and stacked area charts. If stacked is selected, it is recommended to color the areas under the individual data series to avoid misinterpretation.
- **Bar:** Best for data series that map categories to numerical values. Choose between standard and stacked bar charts.

### **Select data**

This is the most complex panel, where you select the data series. A description of available data series can be found in the [Data Series](#) section.

To quickly find the desired data series, filter by evaluation area type (floor, source, elevator, etc.) or x- and y-axis dimensions.

The panel is structured as follows: on the left are all possible data series, while on the right are the data series already added to the chart. A more detailed description of individual data series and their use cases can be found in the “Data Series” section.

To add a data series to the chart, select it in the left section and either double-click it or press the button **+**.

**Note:** Only data with matching area and series types can be displayed in the same graph. Once a data type is selected on the right side, only compatible options remain on the left. For example, if the selected x-dimension is “Time,” you cannot select data where the x-dimension is “Density.”

### **Refine data**

In the next step, you can apply functions to the data series, such as summing values over a specific window, calculating the average, or displaying minimum and maximum values.

There are two ways to set aggregation time intervals: **Simple** and **Advanced**.

- The **Simple** option allows users to define the aggregation window and interval size.
- The **Advanced** option lets users set the start and end positions of each interval.

### **Design data**

This section allows you to define visualization settings for individual data series, including labeling, coloring, and representation. Options vary depending on the selected chart type.

### **Design chart**

In the final section, general visual aspects of the chart are set, such as the title, axis labels, and chart size.

### **Calculating and Displaying the Chart**

To finalize and generate the chart, simply press the **Checkmark Button** (✓) at the bottom.

The final chart will then be displayed. If *crowd:it* has to process a large amount of data, the display may take some time. Don't worry—the results will appear shortly!

Once generated, the chart can be exported using various methods:

-  **Copy to Clipboard**
-  **Save as File**

Available export options:

- **.png**: Saves the chart as an image.
- **“.csv - raw data”**: Saves the base data without aggregation.
- **“.csv - calculated data”**: Exports the result data series after applying the aggregation function.

Alternatively, you can save the configuration using the **Star Icon** (see [Reports][Berichterstattung]).

### **Data Series**

#### **Number of Agents**

**Number of agents over time**: This represents the number of people in a specific geometry (scenario, floor, rectangle, polygon, or tripwire) over time. Using the *average* and *max* functions (see [Refine data](#)), the average and maximum number of people over time (interval in seconds) can be determined.

#### **Walking times**

Travel times can be determined by creating tripwire at the starting position-Vector (A) and endpoint (E). A tripwire chart shows how many people cross each tripwire at a given time. To determine how long people take to travel from A to B, a counting line can be created at both A and B, and a counting line tour table can be generated. Additionally, the times of the last agent movement per floor can be displayed via floor evacuation times.

### **Speed**

Speed over time: This represents the speed of agents in a specific geometry (floor, rectangle, polygon) over time.

### **Dwell Time and Congestion Analysis**

- **Time spent in area:** The duration an agent remains in a specific area (floor, rectangle, polygon).
- **Aggregated time spent in area:** The total accumulated time an agent spends in a specific area (floor, rectangle, polygon).
- **Max individual congestion durations:** The longest single time period an agent moves at a speed below the congestion threshold speed.
- **Total congestion duration:** The sum of time periods in which an agent moves below the congestion threshold speed.
- **Agents in congestion over time:** The number of agents experiencing congestion at a specific time.

### **Density**

- **Local density over time:** Number of people per square meter over time.
- **Local voronoi density over time:** The available space per person over time. Density heatmaps illustrate where bottlenecks occur.

### **Fundamental Diagrams**

Fundamental diagrams based on Weidmann can be displayed for simulation output comparison:

- **Flux over Density:** Represents people flow compared to density.
- **Density vs. Speed:** Represents average density compared to pedestrian speed.
- **Flux over Speed:** Represents people flow compared to speed.

### **Social Distancing**

*Only available if results include Social Distancing*

- **Number of Agents by Maximum Exposure Duration:** The number of people compared to their maximum contact time when violating the minimum

distance with another person. A distance violation occurs when a person stays below the adjustable minimum distance to another person for longer than the configurable minimum exposure duration.

- **Number of Agents by Total Exposure Duration:** The number of people relative to their aggregated duration of distance violations with others.
- **Agents Who Violated Distance:** The number of people over time who violated the required distance from another person.
- **Agents Who Stood Too Close Over Time:** The number of people over time who failed to maintain the required minimum distance.
- **Duration of Violations:** The maximum duration of individual distance violations over time.

### ***Elevator***

The following data series are available for elevators:

- **Agents in the elevator:** Displays the agents currently in the elevator.
- **Number of waiting agents per floor:** Displays the number of waiting people per floor over time.

### ***Queue***

The following data series are available for queues:

- **Agents in queue:** Shows the queue length over time. It can be used to determine when the queue is longest during the simulation.
- **Length per stay in queue:** Similar to dwell time, this data series represents the waiting time relative to the arrival time.

## **5.10.2 Summary Tables**

The menu option **Analyze > Create Summary...** opens a dialog where you can choose between multiple automatically generated tables. All options are categorized and explained below.

### **Area Summary**

Provides a summary for each evaluation rectangle and polygon (area). Each area is described by:

- The average number of people in the area.
- The maximum number of people in the area at the same time.
- The minimum, average, and maximum time spent in the area by all agents.
- The number of different people who have been in the area.
- The number of arrivals in the area.

### **Tripwire Summary**

Provides a summary for a counting line, including:

- The total number of counted agents.
- The time when the first agent was counted.
- The time when the last agent was counted.

### **Path Summary**

Each path is described in detail:

- The number of people assigned to the path.
- The minimum, maximum, and average time in seconds to traverse the path.

As with [Tripwire Summary](#), use **Edit** to define the evaluation period.

### **Origin-Destination Summary**

Provides a detailed overview of each origin-destination combination:

- The number of people moving from the respective origin to the corresponding destination.
- The minimum, maximum, and average time in seconds to complete the journey.

As with [Tripwire Summary](#), use **Edit** to specify evaluation settings (e.g., including lead time in travel time).

### Elevator Summary

This table provides a detailed report on each elevator.

The following information is displayed:

- **Total number of agents who used the elevator during the simulation.**
- **Average and maximum waiting time for the elevator** (measured from arrival at the elevator to when the pedestrian starts entering the elevator. Important: all other pedestrians must exit the elevator first).
- **Average and maximum time to enter the elevator** (measured from when the pedestrian starts entering until the pedestrian's torso is inside the elevator).
- **Average and maximum ride time in the elevator** (measured from when the pedestrian's torso is inside the elevator until the pedestrian starts exiting, including stops at other floors).
- **Average and maximum time to exit the elevator** (measured from when the pedestrian starts exiting until their torso has fully left the elevator geometry).

### Elevator Stop Summary

This table indicates how often each elevator stops at a specific floor.

**Important:** If an elevator does not open its doors at a floor, the floor is not counted, even if the elevator passes through it.

### 5.10.3 Critical area detection

An automatic calculation of all critical areas can be triggered with **Analyze > Critical area detection...**

The following settings can be made:

- **Congestion threshold value [hh:mm:ss]:** Defines the minimum duration that an agent must spend in congestion in an area for this area to be classified as critical.
- **Use bounding box:** If this option is activated, a [rectangle](#) is automatically drawn around all critical areas. Otherwise, a [polygon](#) is drawn around the critical areas for each critical area.

The same calculation is started in the background that is also used for the **Max congestion duration** heatmap. The area is divided into one meter cells and in each cell the maximum time is measured, during which an agent has been stuck already. If this time exceeds the set congestion threshold value, the cell is classified as critical. All cells are iterated over and grouped into clusters. An evaluation object is created for each cluster and displayed in the floorplan. The settings for when an agent is considered to be in a congestion can be made [here](#).

The critical area detection can be used in a congestion analysis to identify the congestion areas and to analyze these locations in more detail.

#### 5.10.4 Heatmaps

Heatmaps allow for the interpretation of simulation results by dividing the scenario area into square tiles. These tiles are then colored based on a selected criterion, helping identify congestion areas that can be further analyzed using evaluation objects.

The Heatmap dialog can be opened via the **Analyze > Heatmap...** menu.

The menu is divided into three sections:

- **Measurement:** Here, you select the coloring criterion, the time period to analyze, and the tile size. Click **Advanced Setting\*\*** to unlock all available options.
  - **Criterion:** Defines the coloring criteria (options are detailed in the Coloring Criteria section).
  - **Start time** and **End time:** Specify the time in seconds over which you want to calculate the heatmap values.
  - **Moving average:** Define the time span for averaging values. Example: If you choose 10 seconds, the next 10 time steps will be considered and averaged.
  - **Tile size:** Sets the size of measurement tiles in meters.
  - **Consider agents while in pre-movement:** Enable this to exclude agents who have not yet started moving (see the Reaction Time section).
- **Coloring:** Displays a preview of the heatmap legend. Click the button next to the preview to adjust colors.
- **Floors:** Define which floors the heatmap should be calculated for.

Click **OK** to generate and display the heatmap.

To close the heatmap, select Delete from the menu above. To modify the heatmap, select **Heatmap...** again.


### Coloring Criteria

The heatmap tiles can be colored based on different aspects. The following options are available:

- **Mean length of stay:** The average time agents spent within the tile is calculated per tile.
- **Time within social distance to any other agent (Max per Tile)** - only available if simulated with Social Distancing: The time agents did not maintain social distancing is calculated per tile. The settings are defined under [Personas](#) and [Social Distancing](#). The average is calculated from all times and all pedestrians.
- **Time within social distance to same agent (Max per Tile)** - only available if simulated with Social Distancing
- **Agents:** Indicates how many agents entered a tile over the observed period. This can help determine path utilization.
- **Evacuation time:** Determines the last time an agent was present in each tile. This helps identify which areas were completely evacuated at which time.
- **Max congestion duration:** Counts the seconds in which an agent was in a congestion (even before entering the tile) until he leaves the tile. The tile value is calculated as the maximum over all agents who were in congestion. This heatmap is used for the [critical area detection](#) as well. The congestion settings can be adjusted [here](#).
- **Density:** Records the density of each tile across all time steps. This means the number of agents in each tile at a given moment is averaged. (Mean or maximum density can be used as comparison criteria between tiles). Density heatmaps can be colored according to Fruin's defined service levels.

- **Speed:** During the simulation, the speeds of all agents entering a tile are averaged. (Minimum, mean, or maximum speeds can be used as comparison criteria between tiles).

## Color Settings

Depending on the use case and audience of the heatmaps, it may be necessary to adjust the color assignment. With a click on the , a color settings dialog opens with the following settings:

- **Scheme:**
  - Default: The default color scheme is white, blue, and black.
  - Custom: A user-defined color scheme. If you change any of the following settings, the color scheme becomes custom.
  - **Service Level:** These color schemes can be used for heatmaps with high density:
    - \* Sidewalk: Service levels for sidewalks.
    - \* Staircase: Service levels for using stairs.
    - \* Queue: Service levels for queues.
- **Gradient:** Specifies whether color changes are continuous or discrete.
- **Color Values:** Here, you can define the color values. First, specify whether the color values should be relative (where the maximum value is set in the dialog) or absolute (where the maximum value is determined by the highest achieved value). Next, set the values. Values can be added, removed, or edited.
- **Legend Preview:** This section displays a preview of the heatmap legend.

## 5.11 Export

After visualizing and analyzing your simulation, you may want to export a video or screenshot of the results. These videos or screenshots can be used to get an overview of the results, highlight specific time steps or critical positionVectors, or create outputs such as heatmap displays and agent movement paths.

### 5.11.1 Screenshot

The menu item **Analyze > Export Screenshot...** allows you to create screenshots of multiple (or all) floors at the current time or at different intervals.

In the window that opens, you first need to specify the filename under which the screenshots should be saved. Then you can select which floors and settings you want to “photograph.” Additionally, you must choose the time at which the screenshots should be taken.

Click the **Advanced Settings** button to unlock a range of additional options. With the **Image Size** settings, you can define the desired resolution of the screenshot. The width and height are bound to the aspect ratio of your floor. Alternatively, you can set the resolution by specifying how many pixels in the resulting image should represent one meter in your floor. If you change the **Pixels per Meter** value, the width and height will automatically adjust accordingly—and vice versa.

To get an impression of your current settings, you will find a live preview on the right. If you have selected a heatmap in the **Advanced Settings** section, you can click the **Recalculate Heatmap with Current Settings** button to generate a preview of the heatmap.

Once configured, you can save this report as a template (by clicking ★) and reopen it later (see section [Report Generation](#)) or directly export it by selecting a folder and then clicking **Apply\*\***.

### 5.11.2 Video

With **Analyze > Export Video...**, you can create videos of the simulation. Here, you can specify the filename for your video(s), select which floors should be displayed, choose the video’s time span, configure visualization settings, and set the desired resolution—similar to the [screenshot dialog](#). Additionally, there is the **Follow Marked Agent** option. This option is only available if you are tracking a single agent in the visualization window (see [Select Agents](#)). If this option is enabled, the video will follow this agent across floors. For example, if the tracked agent is on Floor 1 for the first thirty seconds of your simulation and then moves to Floor 2, the video will display Floor 1 for the first thirty seconds and then switch to Floor 2.

Save your configuration as usual with the **star icon** (★) (see also [here](#)) or generate the videos directly by clicking **Apply**. A target folder can be set via **Folder**. *crowd:it* will be locked during the export process.

After the export, you can open and play the finished videos in a media player.

### 5.11.3 Reporting

Often, you will need to simulate multiple scenarios of the same situation, outputting the same reports. In order to save time, *crowd:it* allows you to save your screenshot, video and chart settings. This feature is named **Reporting**. **Reports** can be viewed, re-opened, and re-exported across multiple scenarios.

#### Save report

The star button ★ can be found in all dialogues in which the **Reporting** function is available. This button allows you to save the current setting as a **report**. In order for a **report** to be recognised, a corresponding name must be entered by the user. The **report** is then added to the saved **reports**.

#### Saved Reports

All saved reports can be viewed, re-opened, re-exported and deleted under **Analyze > Saved reports...**

You can select a single or multiple reports to export or view by using the checkboxes. A double click will open the relevant dialogue menu for that report.

The list of reports can be filtered using the search bar on top of the dialog.

A pop-up menu can be opened by right-clicking in the table so that existing reports can be renamed or duplicated.

Furthermore, created reports can now be saved or imported as JSON. In this way, saved reports from one project can be integrated into another. Above all, this can be applied to reports using evaluation polygons/rectangles, too. It is important for this that all designations for evaluation polygons and floors are the same from project to project. Excluded from this feature are video reports following an agent.

See [advanced tools](#) for more information.

## 5.12 Quick Edit

With **Quick Edit...**, you can select and edit multiple objects simultaneously in an additional window.

The selection process works by applying various filters to your objects. The first filter is the object type:

- **Assigned simulation objects** (e.g., source, destination, etc.)
- **Unassigned simulation objects**
- **Walls**
- **Sets** (groups of simulation objects)
- **Subpaths**
- **Paths**
- **Evaluations**

Sets, subpaths, and paths can be renamed or deleted. Evaluations can also be deleted. Additionally, simulation objects, subpaths, and sets can be added to a set.

For unassigned simulation objects, you can use the Assign Type option to convert the selected objects into, for example, sources. The same dialog menu as in [Assign Simulation Objects][Simulation Objects] will appear.

For simulation objects with an assigned type, the following actions are available:

- **Reset**
- **Create Evaluations:** Generates evaluation polygons that precisely match the geometry of the selected simulation object.
- **Show**
- **Hide**
- **Add to Set**
- **Rename**
- **Delete**
- **Select**

## 5.13 Advanced Tools

### 5.13.1 Automatic Report Generation via Console

#### Installed JARs

After installation, the JAR files are located in the following directory on Windows:

**AppData > Roaming > accu-rate > crowd-it > bin**

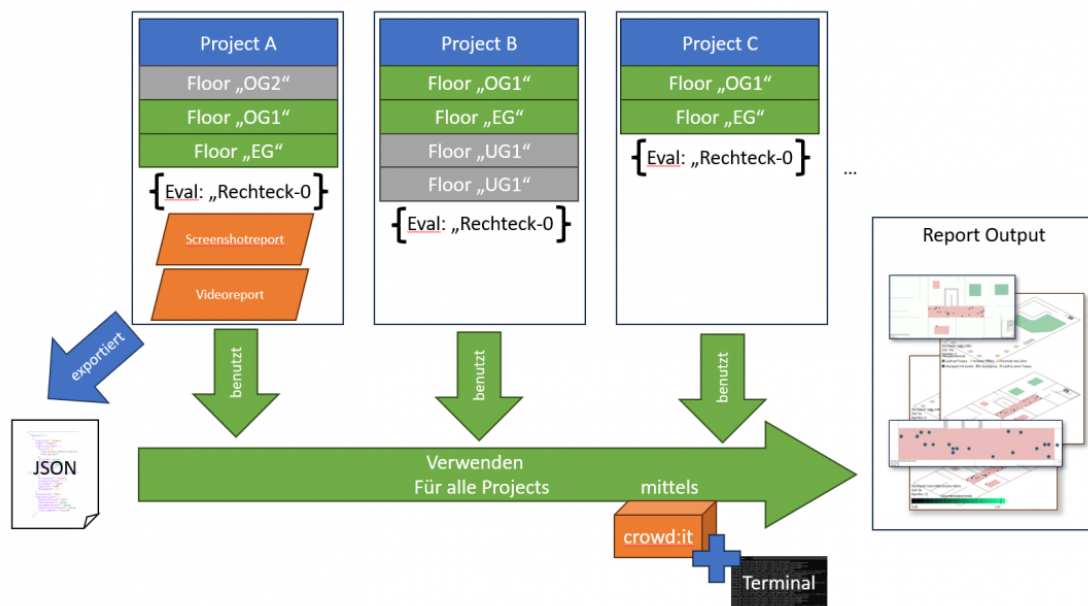
They are divided into the subdirectories **gui** and **kernel**.

## Workflow

The process is as follows:

1. Open the project in crowd-it
2. Create saved reports
3. Under **Analyze > Saved Reports...**, select the desired reports
4. Right-click > **Export as JSON...**, select a folder, and save the JSON file
5. Open the console

The following graphic provides an overview of the process:



If the GUI JAR is accessed directly via the console, the [automatic report generation](#) feature can be used.

```
""bash java -jar -filename= -jsonFile= -exportReports
```

## 5.14 Troubleshooting

### 5.14.1 Slow simulation calculation: Performance tips and tricks

Sometimes the calculation of simulations seems much slower than usual, but there are usually simple explanations for this.

To get your results faster, you can check / adjust the following things.

### **Check geometry**

For simulation, each additional line in your CAD tool affects the simulation time. This also applies to lines that are located in areas that cannot be accessed (e.g. drawing cables in walls).

### ***Reduce lines within the simulation***

Usually, the fastest way to speed up simulations is to move all objects that are not needed / located in inaccessible areas to hidden layers in the CAD tool. This way they remain in the CAD tool but do not influence the simulation time anymore. With our [Simplify Geometry](#) tool, you can also remove all objects that are not needed for the simulation.

### ***Check line duplication***

Often duplicated lines are found in CAD plans, which lie on top of each other and thus offer no further added value. Tip: Most CAD tools offer simple commands to remove such objects.

### ***Check coordinate system origin***

Within crowd:it the same coordinates are used as in the CAD plan. This means that a polyline in your CAD tool has exactly the same coordinates as in crowd:it. This simplifies the search for certain objects in the CAD program, but also has a decisive disadvantage. If the coordinates in your CAD program are very large, this also means large coordinates in crowd:it. Since the simulation file contains much more information than the CAD file (agent positions), large coordinates can result in a significantly higher memory requirement for your simulation file. In addition, very large coordinates can also lead to rounding errors in the simulation calculation.

Therefore, we recommend moving the geometry in the CAD tool to the coordinate system origin.

### **Check simulation settings**

This section should be viewed with caution, as an incorrect setting of the simulation settings can have a strong influence on the simulation results.

Nevertheless, it should be mentioned, as a lot of performance can be gained here.

### **Draw boundary lines**

To avoid long calculation times, you can draw boundary lines around the simulation area. To do this, use the [Walls](#) tool. Make sure that all simulation objects lie completely within the boundary lines and draw the boundary lines as *open walls*.

### **Floor discretization**

Especially in large viewing areas, it is usually not necessary to use very fine granular navigation because there are no narrow paths such as between stage seats.

In these scenarios, the discretization of a floor can be increased from the default value of 0.1 meter to 0.5 meter. This setting is called **cell discretization** and can be changed [\[here\]](#) [\[Show and edit properties\]](#).

This can speed up the calculation of the navigation extremely but might lead to stuck agents in very narrow areas.

### **5.14.2 Out of memory error**

Especially with larger simulations sometimes the error “Out of Memory” can occur.

Reasons for this can be endless, but the most common reason is that the calculation doesn't get enough memory assigned.

If you experience an out of memory error, please contact us ([product-support@accu-rate.de](mailto:product-support@accu-rate.de)) and we will look for a solution together.

# Chapter 6

## Macroscopic Calculations

crowd:it offers several calculation modes. In addition to a microscopic simulation, macroscopic calculations can also be performed.

Currently, the following macroscopic calculations are possible:

- **Predtetschenski & Milinski (P&M):** The Predtetschenski & Milinski method can be used to quickly and approximately determine evacuation times and bottlenecks. It primarily works with pedestrian flows. For a detailed explanation of the P&M method, please refer to the relevant literature.

### 6.1 Workflow

To perform a macroscopic calculation, the following steps are required:

Optional:

1. **Import geometry, e.g., as a background image, as a template for modeling**
2. **Scale the background image to the correct size**

Mandatory:

3. **Define way elements**
4. **Connect way elements**
5. **View and export results**

Unlike microscopic simulation, macroscopic calculation does not require an exact geometry. To determine the necessary data, way elements are drawn and corresponding parameters are assigned. To facilitate the process, it is






recommended to import an existing floor plan, including dimensions, as a background image ([see also here](#)) and scale it to the correct size. A quick tutorial can be found [here](#). This allows required way lengths and widths to be conveniently measured from the plan.

## 6.2 Quick Introduction


A quick introduction can be found [here](#).

## 6.3 Define Way Elements

To build a scenario, the first step is to define way elements. The following options are available in the toolbar:

- : Defines a [start element](#). Every scenario must include at least one way element of this type.
- : Models a [corridor](#).
- : Models a [stair up](#).
- : Models a [stair down](#).
- : Models a [bottleneck](#).

### 6.3.1 Create start element

After clicking the icon , a drawing tool opens, allowing you to define a rectangular element within the drawing space.


In the Predtetschenski & Milinski model, the start element determines the initial density, which then flows through way elements as a pedestrian flow. To calculate the initial density, several parameters must be set.

These parameters are entered in the dialog that appears after defining the area:

#### General Information

- **Name:** A meaningful name is useful since it will be reused as an identifier in the results.
- **Description:** Space for additional information, notes, or comments about the object.

### Geometric Properties

- **Minimum Width:** The narrowest part of the start element.
- **Maximum Walking Distance:** The longest walking distance within the start element (in meters), usually the diagonal of the rectangle. This can be measured directly from the drawing area (by clicking the icon ) or taken from other documents.

The **personas**, i.e. the different person types that appear in the model, can be defined in the [persona table](#). This results in the initial density, which is calculated from the sum of the total projection area of all person types divided by the available area. The resulting density is listed in the Overview.

*Note:* The walking distance and minimum width are required to calculate the walking time within the start element. An additional **corridor** element is not necessary.


### Density Calculation

- **Number of Pedestrians:** The total number of people in the start element. At least one person is required for the process to work.
- **Available Area:** The area of the start element in square meters. It is automatically calculated from the drawn rectangle but can be adjusted manually.

Once all parameters are set, the start element is created by clicking **Apply**.

### 6.3.2 Create Corridor

A corridor represents a path on a flat surface. The model calculates the capacity of the way element and the time required for pedestrian flow based on maximum walking distance and minimum width.

After selecting the icon , a drawing tool opens, allowing you to draw the corridor. A dialog then appears to define its properties.

By default, the horizontal path is suggested as a rectangle. If an arbitrary polygon is desired instead, there are two options:


- First, create a simulation object as a polygon () , then select the object by double-clicking on it and assign **Corridor**.

- After drawing the rectangle, adjust the geometry with a middle mouse click on the element (see also [here](#)).

As with the start element, a name and description can be assigned (see [General Information](#)).



### Required Properties

- **Minimum Width:** The narrowest part of the path.
- **Maximum Walking Distance:** The longest walking distance within the element.

Both values are entered in meters and can be either manually input or measured using the measurement tool in the drawing area. Simply click the icon  next to the input field, which opens the drawing area for measuring the desired distance.

### 6.3.3 Create Stair up or down


Pedestrian flow slows down on staircases. The model accounts for this delay based on the width and direction (upward/downward) of the staircase.

After selecting the icon  for an upward staircase or  for a downward staircase, the staircase is drawn. A dialog then opens for entering the required values.


Similar to the **corridor** element, the minimum width and maximum walking distance must also be set ( see [here](#)).

### 6.3.4 Create Bottleneck

Doors and other bottlenecks must be modeled separately, as pedestrian flow is reduced in these areas. The clear passage width determines the impact on pedestrian flow.

After selecting the icon , a line is drawn in the drawing area. A dialog then opens to define the basic properties ( see [General Information](#)) and the passage width.

## 6.4 Connect way elements

For calculations, the way elements must be connected. This is done using the icon  in the toolbar.

After switching to path mode, clicking on a way element creates an arrow that can be connected to another way element (except for the start element). The arrows can be deleted either by right-click via the context menu or by pressing the **DELETE**-key.

**Important:** At least one path must originate from a **start element**, otherwise, no calculation is possible.

As long as an outgoing path exists from an element, that element is not considered an exit. In other words, if a path element should act as an endpoint for analysis (see overview table), ensure that no outgoing arrows exist from that element.

### 6.4.1 Merging

When two paths merge, the pedestrian flows in the model are added together. Time delays due to staggered arrivals are not considered, which may slightly overestimate evacuation times. If this effect should be explicitly modeled and it is ensured that pedestrian flows arrive with staggered timing, two separate paths can be modeled, and the way element can be duplicated.

This ensures that the element is available at full capacity for each incoming pedestrian flow.

**Caution:** Simply reducing capacity by assigning smaller widths to duplicated elements does not work, as the correct densities would not be accounted for.

### 6.4.2 Splitting

If a way element leads to multiple destinations, pedestrian flows are distributed based on the width of the outgoing paths. If a different distribution is required, the source can be divided into multiple sub-areas. The available area must be adjusted accordingly.

The following image illustrates an example where pedestrian flows are distributed equally rather than according to adjacent escape path width:



## 6.5 Overview of all modeled way elements

In the information panel to the right of the **floor plan view**, the **Properties** tab provides an overview of all modeled way elements, including all properties used as input for calculations.

Clicking a column title sorts the respective table column, which is useful for sorting way elements by floors, for example.

Clicking a table row or hovering over a row highlights the corresponding element in the floor plan. Conversely, selecting an element in the floor plan highlights the corresponding table row.

This overview can be exported for documentation purposes as a .csv or .png file or copied to the clipboard for further use. (For detailed export instructions, see [here](#).)

## 6.6 Persona table

In the persona table, predefined person types based on Predtetschenski & Milinski can be selected and weighted by percentage. The total must sum to 100%. The following person types are available according to the model:

Category	Projected area [m <sup>2</sup> /P]
Child	0.06
Juvenile	0.09
Adult + summer clothing	0.1
Adult + Transitional Clothing	0.113
Adult + Winter Clothing	0.125
Adult + Transitional Clothing	
... with small luggage	0.18
... with suitcase	0.24
... with backpack	0.26
... with big luggage	0.39
... with child by hand	0.2
... With child on arm	0.26
... child by hand and luggage	0.32

## 6.7 Modeling Tips

- Since this is a macroscopic model, an accuracy of 10–20 cm is sufficient. Higher accuracy suggests a precision that the model cannot provide.
- Path elements should not be modeled in excessive detail. If necessary, consider microscopic simulation. For example, a staircase with multiple landings can be modeled as a single staircase.

## 6.8 Visualization Options in Macroscopic Mode

The mode-dependent menu bar contains the entry **Visualize**. Here, the following visualization settings can be made:

- **Show Paths:** Paths can be toggled on and off.
- **Set colors...:** Macroscopic elements, walls, and the background can be recolored, enabled, and disabled. [Here](#) is explained, how this works.
- **Object label...:** The configured information and the name of each macroscopic element can be displayed in the floor plan view. This is done via the **Object labels** dialog. Under **Show additional info**, the configured properties are displayed in the path element.

## 6.9 Calculation

The calculation of the model takes place in the **Calculation** tab, located to the right of the **Floor plan**. Here, you can start the calculation and view the results. In the upper area of the tab, the calculation type is specified. The model according to Predtetschenski & Milinski (P&M) offers three different calculation types:

- **Comfortable conditions**
- **Dangerous conditions**
- **Normal conditions**

For evacuation calculations, the calculation type **Dangerous conditions** is generally used. However, this can be individually adjusted by the user.

After selecting the calculation type, the results in the tables below are immediately updated.

## 6.10 Results

### 6.10.1 Overview Table

This table lists the following information for each exit or each path endpoint:

- **Exit:** Name of the exit, as defined in the name field of the way element.
- **First Person walking time:** Time in minutes when the first person reached the exit.
- **Last Person walking time:** Time in minutes when the last person reached the exit, i.e., when all individuals had left the scenario (e.g., a building). This also includes congestion time.
- **Maximum persons in congestion:** Maximum number of persons who were in a congestion in a way element on the way to the exit. **Note:** This number is determined from the congestion analysis according to P&M by multiplying the congestion length with the maximum density defined in P&M. This may lead to an overestimation of persons in the congestion, as the maximum congestion density is  $0.92 \text{ m}^2/\text{m}^2$ .
- **Maximum congestion walking time:** The congestion time is determined across all way elements of a path. The maximum congestion time refers to the path with the longest congestion time.
- **Number of Paths:** Number of paths leading to this exit.

### 6.10.2 Detailed Analysis



In the lower area of the tab, a detailed analysis of the individual paths leading to an exit is displayed. To view these, select the corresponding row of the exit in the overview table. The detailed analysis contains the following information:

- **Way element:** Section or element of the path. The name was set in the respective way elements and can be changed there.
- **Min. width [m]:** Minimum width of the way element in meters, as defined here.
- **Max. walking distance [m]:** Maximum distance covered within this way element. This value is specified by the user and can be changed under here.
- **First Person walking time:** Time in minutes when the first person reached the end of this way element.
- **Cumulated walking time:** Total time required for the first person to traverse the path including this way element.
- **Congestion delay:** Indicates whether and how long delays occurred within this way element.
- **Time for last person:** Time when the last person left this way element.
- **Number of pedestrians:** Total number of people who moved through this way element.
- **Persons in congestion:** Number of people who were in a congestion within this way element.

To switch between the respective paths, you can use the arrow keys above the table: < and > .

### 6.10.3 Exporting Results

The tables can be exported in various formats to be included in the results report.

Below each table, there is an option to select the export format (.png, .csv). After selection, the data can either be copied to the clipboard () or saved as a file ().

**Note:** When exporting as .png, only the currently visible table is exported. If the table headers should be fully readable, the table must be resized accordingly.

### 6.10.4 Calculation Parameters

To trace individual calculation steps and verify the calculation parameters used, all relevant values are summarized in the **Parameters** tab. The listed calculation parameters originate from the model and are derived from so-called calculation value tables, which represent a fundamental diagram in tabular form. Each density is assigned a speed and a pedestrian flow rate.

**Note:** In the original model, these values are listed in steps of 0.01. In **crowd:it**, the original formula is implemented, allowing an exact value to be calculated for each density. Therefore, slight deviations may occur compared to the table values.

The following parameters are listed:

- **Name:** Name of the way element
- **Type:** Type of the way element
- **q\_in:** Adjacent pedestrian flow of the way element in [m/min]
- **q\_max:** Maximum capacity of the way element in [m/min]
- **q\_adjusted:** If the adjacent pedestrian flow exceeds the capacity of the way element, the pedestrian flow is adjusted. For this, the value from the calculation value table for the maximum possible density is used.
- **Congestion:** Indicates whether a congestion formed at this way element. If  $q_{\text{adjacent}} > q_{\text{max}}$ , a congestion occurs.
- **Speed:** Flow speed in [m/min] at which the adjacent pedestrian flow moves through this way element. This value is derived from the calculation value table.
- **Density:** Density in [m<sup>2</sup>/m<sup>2</sup>] generated in the pedestrian flow.
- **Q:** Movement intensity in the way element, calculated as  $q_{\text{adjusted}} \times \text{minimum width of the way element}$ .

## 6.11 Analysis

In the **Analysis** tab, you can visualize and analyze the results of the calculation in the floor plan.

Different colorizations of the respective path elements can be selected:

- **Speed:** Each path element is colored according to the speed of the pedestrian flow, allowing a quick overview of where the pedestrian flow slows down.

- **Congestion delay:** The delay time per path element indicates where delays occur and where congestion forms.
- **Time for last person:** This option allows further selection of the target for which the analysis should be performed. After selecting the target, each path element is colored according to when the last person moving toward the target passed that element. If multiple paths lead to the target via a path element, the maximum value is displayed.

## 6.12 Export

The colored objects can be exported and saved as .png via the **Export** button. A dialog opens, similar to the **Export Screenshot\*\*** dialog in microscopic mode. The following settings are available:

- **Floors:** Specifies which floors should be exported as a screenshot.
- **Visualization:** Allows enabling/disabling the legend and/or scale as well as the floor name. Additionally, the floor boundary can be displayed when a specific perspective is selected.
- **Rendering:** Determines the size of the exported image. The preview window on the right shows how the image will look. Additionally, you can choose whether the floor should be displayed and saved in an isometric perspective.
- **File Settings:** Specifies where the screenshot should be saved.

# Chapter 7

## Glossary

**Deviation:** Standard deviation

**Agent:** Simulated pedestrian

**Disable dynamic navigation calculation:** Disables dynamic flooding for this destination.

**Directed scaled area:** A scaled area that can specify different directions to define different braking/acceleration factors.

**Heatmap:** A spatial map with color coding based on density, speed, or other aspects of the simulation.

**Capacity:** Capacity limitation. For example, how many agents can enter the waiting area simultaneously.

**Paths:** Movement paths of a pedestrian.

**Name:** Name of the simulation object.

**Path:** The connection between the source and the destination along which agents are supposed to move. May include intermediate destinations such as stairs, portals, and waiting areas.

**Portal:** Interface connecting two floors.

**Origin:** Sources where agents are generated.

**Queue:** Element representing the start of a queue.

**Reaction time:** The response time to a specific event (e.g., alarm system), which pedestrians “wait for” before they start moving.

**Set:** Group of simulation objects (sources, destinations, stairs, etc.).

**Scaled area:** Areas where agents can be slowed down or sped up.

**Simulation object:** Simple polygons in the CAD program that can later be converted into sources, destinations, stairs, etc., in crowd:it.

**Floor:** A collection of all geometries of a simulation observation area that are on the same level.

**Stair:** Object representing a rectangular staircase.

**Waiting zone:** An area where pedestrians wait for a certain period.

**Exit time:** The moment when an agent reaches the safe area or final destination.

**Destination:** Target areas that accept agents; sinks where agents are removed from the simulation.