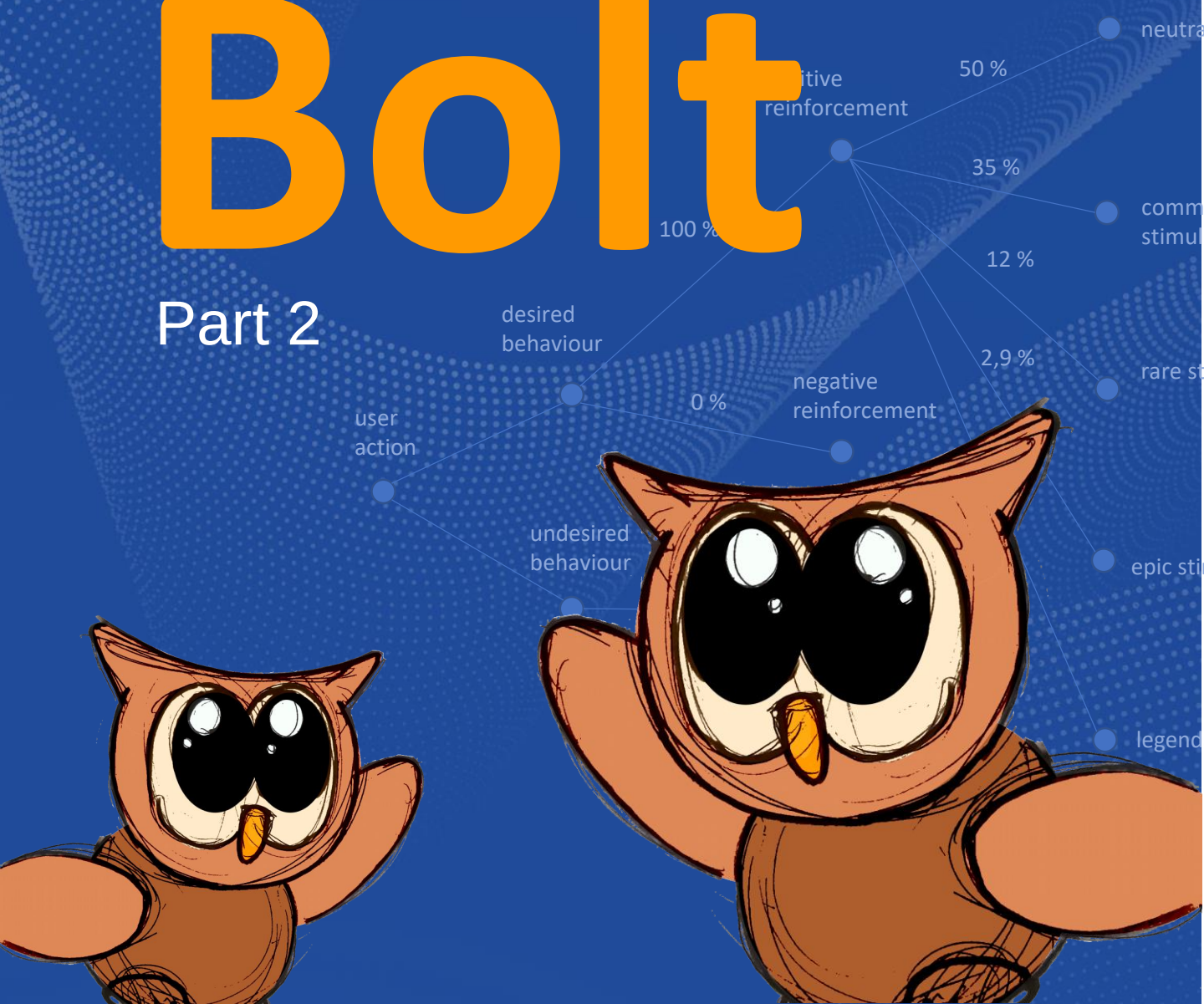
FreeCAD Beginner Assistant

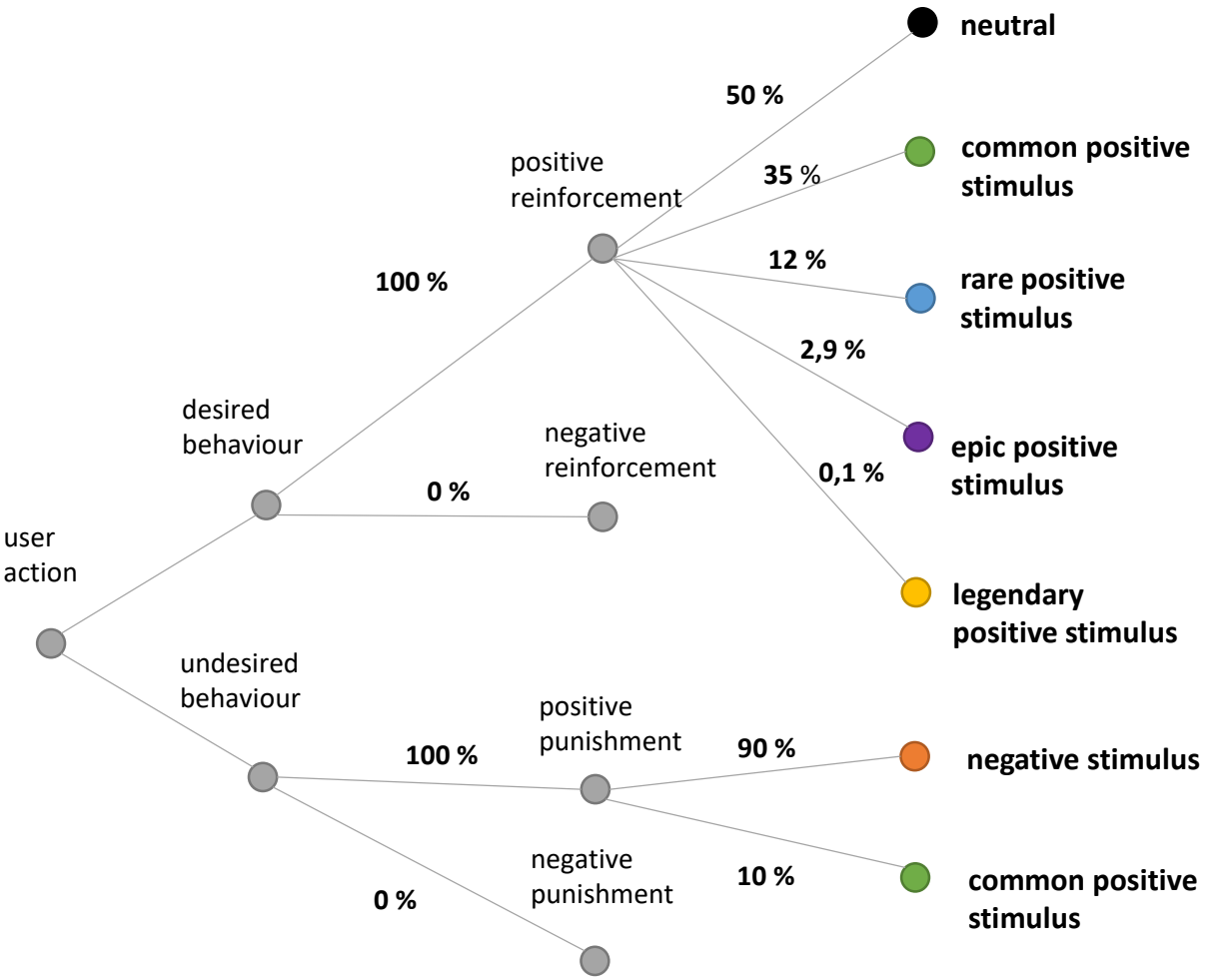
The Psychology of Bolt

Part 2



Determining Bolts‘ stimuli

Bolt used punishment and reinforcement to teach the user. Bolts‘ behaviour can be modelled by a tree diagram. The tree diagram refines the reinforcement and punishment of Bolts‘ teaching by a range of randomly selected stimulus grades.
The randomness of the stimulus keeps the user hooked, as it is unpredictable. This method in known from computer games where player get rewards based on a random chance (loot).



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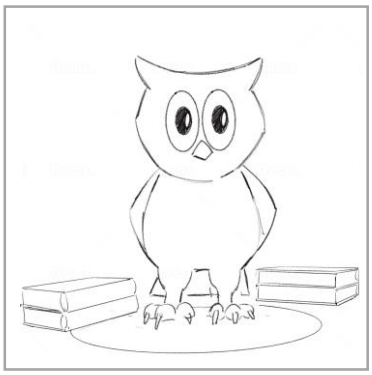
Visual and non-verbal stimuli

Each stimulus grade is represented by a different visual and non-verbal stimulus. The impact of each stimulus rises with its rarity.



neutral

common positive

rare positive

epic positive

legendary positive

negative

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Verbal stimuli

While the visual and non-verbal stimuli are mainly responsible for managing the feelings of the user, the verbal stimuli give informative feedback on the users actions.
For the initial release of the FreeCAD Beginner Assistant „Bolt“, 21 best practice checks have been defined to be compared to the users actions.
Each feedback consists of „What the user has done, „What negative or positive effect that has“ and „How to resolve it“.

ID	What the user has done	What negative (and positive) effect that has	How to resolve the issue
1	You have referenced a face of your 3D model (topological element) for your sketch.	This might lead to the sketch losing its reference, when the topological elements change.	Reference one of the Origin planes or create a new plane, that also only references one of the Origin planes instead.
2	You have created a sketch, that is under constrained	This might lead to unexpected behaviour, when you use that sketch for a feature	Go back to your sketch and fully define it using dimensional of geometrical constraints.
3	You have created a sketch, that is over constrained	This might lead to unexpected behaviour, when you use that sketch for a feature	Go back to your sketch and remove redundant constraints.
4	Your 3D model is not symmetric in relation to one of the Origin planes	Designing your 3D model symmetric to as many Origin planes as possible makes it easier to modify it in the future.	Try to create your 3D model symmetric in relation to as many Origin planes as possible.
5	You have created a complex sketch that uses a lot of geometrical elements and constraints	This might lead to performance issues and make building your 3D model slow.	Split up your complex sketch into multiple simple sketches if possible.

Verbal stimuli (2)

ID	What the user has done	What negative (and positive) effect that has	How to resolve the issue
6	You have not given your sketch a useful name.	This might lead to you getting confused when trying to reference a specific sketch in a formula or selecting it for a feature.	Rename your sketch to something that represents its purpose
7	Your file size is getting large.	This might lead to stability issues and the program crashing	Save your document regularily.
8	You have created a new document that might not be using version control.	Using version control for your documents will help you keep track of your document progress. Also, in combination with a secure hosting service, this prevents you from losing data.	Use a version control, like git or Subversion. To host your files using git you can use Github.
9	Your document contains at least one error.	This might lead to unexpected behaviour of your 3D model or the program even crashing.	Resolve all errors first, before continuing with your work on this document. Thank me later.
10	Your Part Design body contains many features.	This might lead to the 3D model being hard to understand and might potentially lead to performance issues.	Try to combine features, especially the ones dependent on sketches.
11	Your system will soon run out of memory.	This might lead to the program crashing or dramatically slowing down.	Free up some memory by closing other programs or files that you don't need right now.

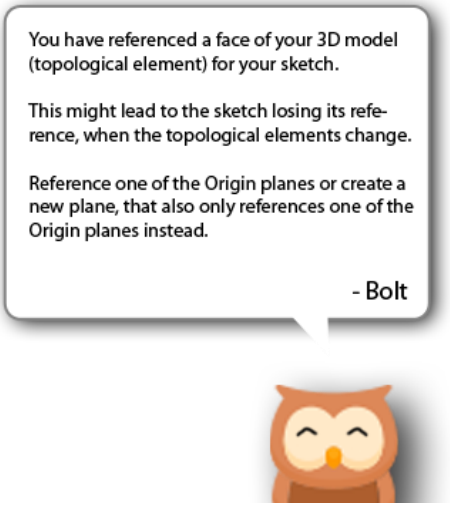
Verbal stimuli (3)

ID	What the user has done	What negative (and positive) effect that has	How to resolve the issue
12	Your system cpu usage is very high.	This might lead to the program crashing or dramatically slowing down.	Free up some cpu resources by closing other programs or files that you don't need right now.
13	Your system will soon run out of disk space.	This might lead to you not being able to save your document.	Free up some disk space by deleting or moving files that you don't need.
14	Your FreeCAD program version is not up to date.	This might not only prevent you from using the newest features but also usability and stability improvements in existing features.	Install the newest stable version of FreeCAD. You can go back to this version and use both side by side.
15	You have not saved your document in a while.	This might lead to you losing your progress if the program crashes.	Save your document now. You can also enable auto-save, to let FreeCAD take care of that.
16	You have created a sketch that does not contain a closed wire	This will lead to creating a zero thickness surface instead of a solid	When creating solids, always close your wires in your sketch.
17	You have created an additive Part Design feature after a subtractive one	This might prevent you from staying flexible in your modeling approach.	If possible, first create all additive Part Design features and then all the subtractive ones.

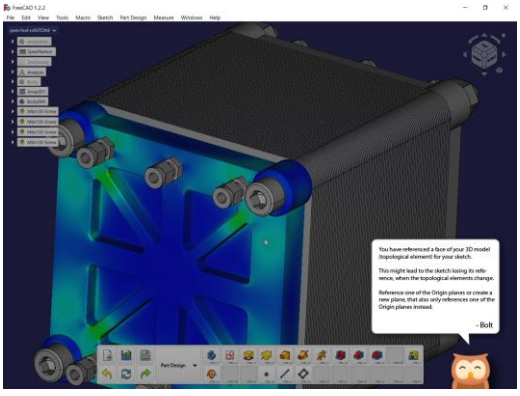
Verbal stimuli (4)

ID	What the user has done	What negative (and positive) effect that has	How to resolve the issue
18	You have created a Part Design feature after a fillet or chamfer	This might lead to you losing the referenced edge(s) from the fillet or chamfer feature	Create all fillet and chamfer features at the end of the modeling steps.
19	You have saved your FreeCAD document using a name that is not compatible with the Linux operating system	This might lead to other FreeCAD users not being able to open your file when using Linux	Rename your FreeCAD document to something that's compatible with Linux. Use underscores or hyphens for multipart names (e.g. "my-own-spaceship.FCStd"). Remember that Linux is case-sensitive.
20	Your sketch or feature geometry intersects itself, leading to invalid geometry.	This can lead to errors in operations that depend on that sketch or feature.	Modify the sketch or feature to remove or resolve the intersecting geometry.
21	You have used a non-standard file format to save your part.	This can create compatibility issues when sharing files with others.	"Export" or "Save As" your part in a standard file format.

Combining all components



Whenever a user action relates to a best practice, Bolt gives feedback using the stimuli defined in this document. The verbal stimulus is represented by a speech bubble with text. Whenever the user permorms an action that does not relate to best practices, a neutral representation of Bolt is displayed.



Bolt‘s visual representation and the speechbubble with text are located in the bottom-right corner of the FreeCAD graphical user interface.

The Psychology of Bolt, the FreeCAD Beginner Assistant (Part 2)

This overview showcases the development of „Bolt“ process from a macro perspective, starting off with established methods of learning known from psychology, to teach best practices to FreeCAD beginners. Here, Bolt gains **visual, verbal and non-verbal stimuli** for his learning method **operant conditioning**.

What‘s next?
Now it‘s time to determine, how the background story of Bolt can be incorporated into the FreeCAD graphical user interface and how the first interaction of the user and Bolt will look like.

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