

GSoC 2023 Proposal

Project Title

Unified Measurement Facility

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Synopsis

There are currently several ways of measuring objects in FreeCAD: Draft Dimensions, TechDraw Dimensions, Manipulator Caliper, Measure Distance, Measure Linear, etc. This causes confusion for users and makes the learning curve of FreeCAD steeper.

Benefits

A single method of measuring objects would smooth the workflow and reduce learning time. The user will always be able to invoke one global measure command regardless of the active workbench.

Deliverables

1. Refactor base command to be a generic interface where workbenches can hook into
2. Refactor and port existing tools to use the new interface
3. Polish existing measuring commands to get a consistent user experience regardless of the geometry type that is measured

4. Optional: Add additional measurement modes and settings
5. Optional: Communicate with developers of external workbenches to port them to the new system

Project Details

Existing Functionality

The several existing ways of measuring can be put into categories, modal/persistent measurement and annotation commands, where the first is meant to quickly check a measurement and the others use parametric objects that will stay around.

It's important to note that annotations have quite different requirements which are specific to a workbench. They might have a lot of settings like styling and specific user interaction. Having these follow some kind of interface might be rather restrictive than beneficial. Also grouping all kinds of annotations into one annotation tool might be more confusing than actually helpful. Therefore I propose that this project will be focused on measurements.

Modal Measurements

- Measure Distance: Base measure command
- Manipulator WB Caliper: Advanced measuring tool from an external workbench
- Part WB Measure Linear/Angle : Allows measurements between geometry elements (vertex, edge, face)
- Measure WB: There seems to be a measure workbench in the codebase, doesn't seem to be exposed currently?

Persistent Measurements

Todo: which measurements are persistent?

Annotations

- Draft WB Dimensions: Parametric dimension objects
- TechDraw WB Dimensions: Parametric dimension objects

Measurement Interface

With FreeCAD's modular architecture it's important to have extensibility in mind. Currently there's one measurement command in the base application

“std_measureDistance”. This will likely be reused as a base command which can be extended by workbenches.

Therefore the base command has to work with abstract geometry. Workbenches will be responsible for translating the specific geometry to something abstract.

Measurement Types

Implemented in the base of FreeCAD. These types would handle all the drawing and user interaction. Such types could be the following:

- Distance
- Angle
- Diameter
- Center of Mass

Geometry Targets

Implemented by workbenches. These would be required by the measurement types to get the necessary properties from the unknown geometry. Such targets could be the following:

- Location
- Axis/Line
- Plane
- Orientation

Simplification

Some of the mentioned commands seem to have overlapping functionality, it would be great to remove such redundancy which makes the code more consistent and maintainable. One simplification approach could be to reuse the persistent annotation objects for modal measurements.

Project Schedule

Before the official approval of the project, I will be reading the codebase, investigating possible implementations and acquiring feedback from the community.

Schedule Conflicts

I'm currently 80% employed, besides that I will have to reserve some time for the Cad Sketcher project for reviewing contributions and guiding newcomers. Therefore I would prefer to expand the standard timeline of 12 weeks.

Community Bonding period (May 4 to 28)

During this period, I will further investigate how the existing tools are working and will communicate with mentors and other developers to finalize implementation details and adopt community feedback.

Week 1-4

- Investigating the codebase and possible solutions
- Implementation of a basic prototype
- Gathering feedback for prototype from mentor
- Iterate over alternatives and decide end solution
- Implementation of Deliverable #1

Week 5-7

- Implementation of Deliverable #2
- Testing and debugging

Week 8-10

- Implementation of Deliverable #3
- Final testing and debugging

Week 11-12

- Cleanup and simplification
- Add documentation of implemented features
- Add remaining unit tests
- Work on optional deliverables if time allows
- Buffer period. Address any remaining issues

Bio

I'm David Friedli, an open source enthusiast interested in end-user applications in the area of graphics and engineering. I've studied construction technology and am therefore experienced with commercial CAD systems. I started using FreeCAD back in 2013 for my diploma work and have used it occasionally since then.

In the past years I've gotten into computer science and started contributing to some open source projects. Most notably i've made some patches to blender¹ written in C and authored the blender addon CAD Sketcher² written in Python. With that I learned a lot about software development, workflows and tools it requires. While this task will definitely be a challenge for me, I'm motivated to appropriate myself the required skills and take the project further even after the GSoC event.

I am really looking forward to getting involved in FreeCAD development.

Thanks for your effort!

¹

<https://projects.blender.org/blender/blender/commit/f417787ee116f25889f853a98319d4e351edb3b5>

<https://projects.blender.org/blender/blender/commit/b1bdfb6e32b9371048677196ac256290cfd35d0b>

² https://github.com/hlorus/CAD_Sketcher