SteamCAD User Guide

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Chapter 1

Introduction to SteamCAD

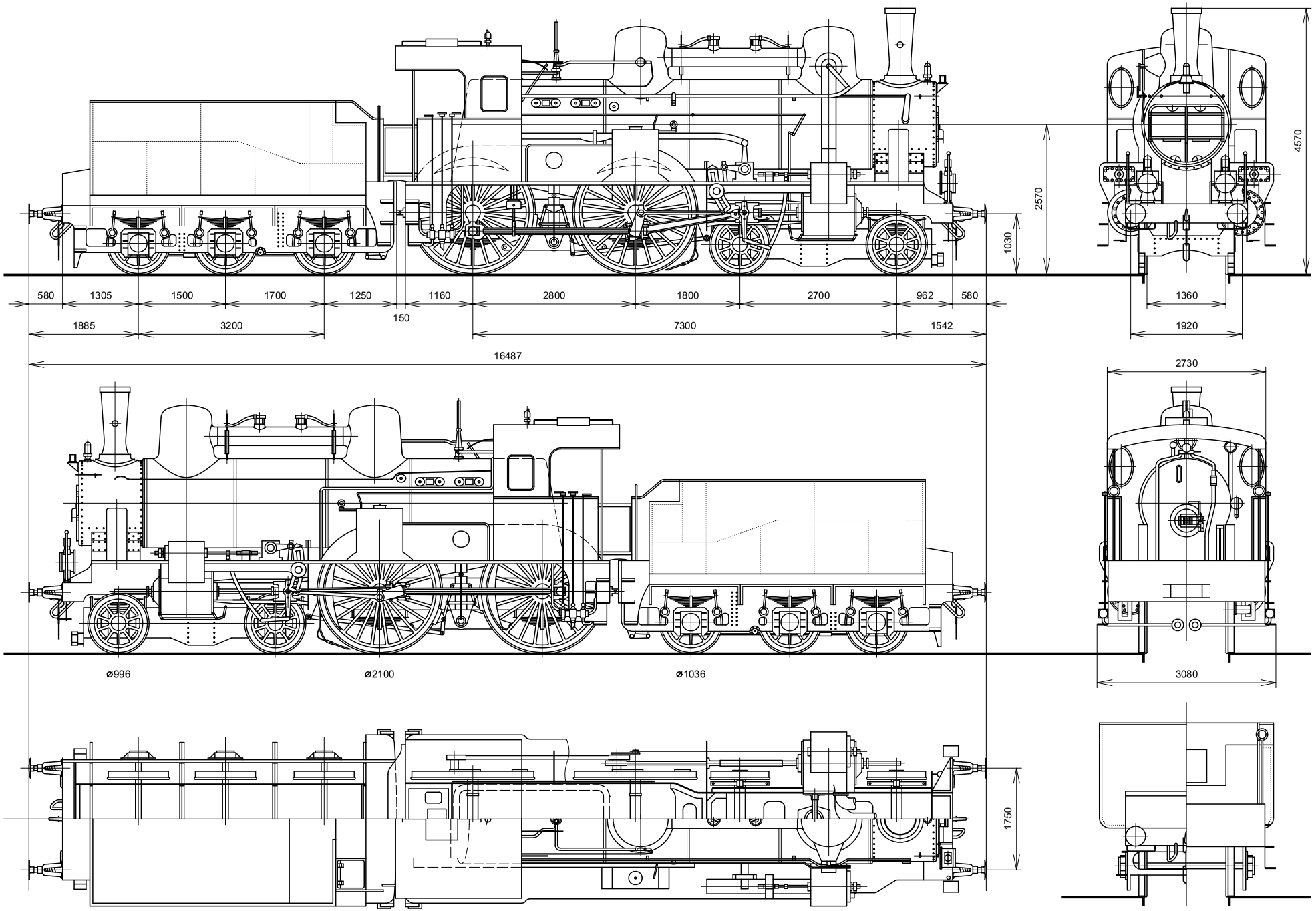
SteamCAD is a 2D drawing tool especially designed to create drawings of steam locomotives as you can see on the next page[1]. First of all, why is it called SteamCAD? There are basically two reasons for this name. One was already mentioned - the tool is specially designed for drawing steam engines. But there is another meaning, it should emphasize that the software is very lightweight. The user interface is minimalistic, the application window has a menu bar and a status bar only. There are no toolbars with thousands of buttons and functions. The main aim when designing SteamCAD was that the application contains as few tools, as few controls, as possible. All the functionality is accessible from the menu, and each command has a shotcut, so that it is possible to work with the application just using mouse and keyboard. For experienced user, almost no interaction with the application GUI is required.

As a result, the drawing area is not obscured by controls and can occupy almost the whole screen. This is what makes the SteamCAD different from any other CAD. SteamCAD is focused on the productivity, not on the fancy look.

1.1 History

SteamCAD is in many ways inspired by LibreCAD/QCAD. But it shares no code with it. I liked some of the LibreCAD/QCAD ways how it does things, and I have also contributed small piece of code to LibreCAD. But in the end I realized that DXF format, which LibreCAD adopted as the native storage format, is the main blocker. So in the end I decided to create a completely new software, taking the good things from LibreCAD and ommiting the bad things.

[1] This drawing presents the Austrian steam locomotive class 6 kkStB. This type was build in several Austrian factories from 1894 to 1898. The image is based on the factory's drawing and is at exact scale 1:87 (known as H0 scale).


580
1305
1500
1700
1250
150
1160
2800
1800
2700
962
580
1885
3200
7300
1542
16487
2570
1030
4570
1360
1920
2730
3080
ø996
ø2100
ø1036
1750

1.2 What is SteamCAD

SteamCAD is an open source software for creating 2D technical drawings, especially designed to produce vector drawings suitable for further publishing process. The software itself does not provide any printing capabilities. Its aim is solely to produce drawings in a format consumed by other technologies, either desktop or web based.

The software features several advanced technologies, namely length and angular unit handling, both for entering and for printing, and linear pattern conflict resolution. Both will be explained later in more details.

The software is available in Linux and Windows version. Both versions use native Windows management, and different technology for drawing backend. It uses Cairo on Linux and GDI on Windows. Cairo as a drawing backend is very slow on Windows, so we were forced to use other technology, thus the Windows experience is a little bit worse than the Linux's one. But the software is primarily intended for Linux users.

The philosophy of SteamCAD is to follow the manual drawing and adding some advanced features not available for hand drawing, such as mouse snapping tool.

1.3 What is NOT SteamCAD

SteamCAD only allows drawing of black lines. It does not utilize colors at all. However, you can use SteamCAD to quickly create exact 2D drawing and export the result into SVG format. Then continue with coloring in another aplication such as Inkscape.

SteamCAD also does not define areas. It only draws line and curve segments. You can create a hatch patter fill the same way you would do it in a hand drawing.

Finally SteamCAD only creates 2D drawings. It is not a 3D CAD.

1.4 Installing SteamCAD

To install SteamCAD on a Linux system, you should install the follwoing dependencies: gcc, cairo-dev and gtk+-dev. Then download the source code and run "make main" from the console. Before you run the application, copy the files DPapers.ini and DUnits.ini from the Papers fodler into the /home/<me>/.SteamCAD folder. The meaning of the ini files will be explained later.

The compilation steps on Debian based systems might look as the following:

```
$ sudo apt-get install gcc libcairo2-dev gtk+-dev
$ git clone https://github.com/oskardolch/SteamCAD.git
$ cd SteamCAD
$ mkdir Build
$ make libdxflib.a
$ make main
```

Then you can run SteamCAD from the Build folder or copy the executable file anywhere.

Building of the software on MS Windows is not straightforward, since it involves building of the cairo.dll. So the best is to download the zip with binaries for either 32bit or for 64bit platform. Note that Windows XP and older systems are not supported by the provided build. If you want to run it on an unsuported Windows system, you should compile it yourself.

There is no installation utility provided for MS Windows, just unzip the package into a directory where you have write access to. So the best choise is somewhere under your documents folder.

1.5 Prerequisities

SteamCAD has minimal hardware requirements, you should be able to run it on almost every computer. However, what you desperately need is a 3-button mouse with wheel. There is no way to work with SteamCAD without such a device.

Chapter 2

Drawing Basics

There are 8 line types available in SteamCAD - line, circle, ellipse, “arc ellipse”, which is an ellipse-like shape composed from circular arcs, parabola, hyperbola, spline and evolventa.

Before we discuss each of the basic elements, we should mention one aspect also (probably) unique to SteamCAD. From geometry point of view, the elements can be either bound or unbound. To draw a bound element, like circle or ellipse, it is quite easy. However, the question is what to do with unbound elements, like line, parabola or hyperbola? In SteamCAD, when you draw an unbound element, it exists in the drawing as a “whole”. What does it mean? It means that if you draw for example a hyperbola, and then zoom out, the hyberbola ends at the drawing view. You can zoom out as much as you want and the whole hyperbola is still there. And it remains like that until you cut it from two sides. Then it becomes a bounded object.

Each drawing primitive has one or more modes how to be drawn. The modes comprise of mouse button and key combinations. However, in most situation, holding the Ctrl key while placing a point will snap it to the invisible grid. The grid size can be set in document properties. Some curved primitives also require one or two line primitive to be selected prior placing the curve. We will explain all the modes in this chapter.

2.1 Line

Line can be started by pressing “L” shortcut. Once you are in line mode, you can place the first point by clicking the left mouse button. When the first point is set, the behavior of the Ctrl control changes. For placing the second point of a line with Ctrl, the point is snapped so that the line angle fits certain angles. The “angular grid” is set to 15 degrees by default, but can be set in the document properties.

Line has a second placing mode, which is if you start the first point by right mouse button click. In this case, the line is drawn so that it mirrors the second point into the first point. This is useful if you need to find a middle point of two points.

When entering the line mode, there is also an edit box on the status bar. You can specify the angle the line should incline from the horizontal line.

2.2 Circle

The circle can be started by pressing “C” shortcut and can operate in four modes.

1. Plcaing the centre by right mouse button and a point on the circle by left mouse button.
2. Placing three points on the circle by left mouse button clicks.
3. Placing two poins defining the diameter by first left mouse button click and left mouse button double click.
4. Placing two points on the circle by left mouse button clicks while keeping the circle cetner on a selected line.

When placing a circle, there is also an edit box on the status bar, wehere you can specify the circle radius.

2.3 Ellipse

The ellipse can be started by pressing “E” shortcut and can operate in three modes.

1. Placing two focus points by right mouse button click and one point on the ellipse by left mouse button.
2. When two non-parallel lines are selected, left mouse double click places the ellipse so that the ellipse center lies on the lines intersection and the major and minor ellipse semiaxes mirros the selected lines.
3. When two non-parallel lines are selected, two left mouse clicks define two points on the ellipse while the ellipse center lies at the two selected lines intersection.

When placing with selected intersecting “construction” lines, the ellipse is constructed so that the tanget on the ellipse and a construction line intersection is parallel to the second construction line. This can be useful when drawing in axonometric projection.

2.4 Arc ellipse

Arc ellipse is a special primitive, which in fact could be replaced by four circular arcs. However, it is there to help mimic the old style of hand drawing, when creating an exact ellipse was difficult, and the ellipse was often approximated by a shape consisting of circular arcs.

It has two modes of operation. Two non-parallel line primitives must always be selected, and the arc ellipse can either be placed by one left mouse button double click, or by two consecutive left mouse button clicks. The shortcut for arc ellipse is “A”.

2.5 Hyperbola

To place a hyperbola, two non-parallel lines must be selected. The hyperbola is then placed by left mouse button click and this is the only mode. The shortcut for hyperbola is “H”.

2.6 Parabola

Parabola always requires one line primitive to be selected. It is then placed by single left mouse button click and the clicked point represents the parabola’s focus. Parabola can be invoked by “P” shortcut.

2.7 Spline

Spline places a second order B-spline curve. The curve is designed so that it goes through the clicked points. The points are placed by left mouse button click and the number of points is not limited. Clicking the right mouse whenever during the curve creation toggles the closeness. Initially, the spline is open. When clicking the right mouse button, the spline becomes a closed curve.

Double click the left mouse button to finish the curve. The double clicked point becomes part of the curve definition. Spline can be invoked by pressing “S” shortcut.

2.8 Evolventa

This is a curve created by rolling out a line along a circle. This is included in SteamCAD as a gear creation helper. To start evolventa, one circle must be selected. Start evolventa by pressing “V” shortcut and place two points by left mouse button click to define two points on the evolventa.

2.9 Placing a line tangent to two curves

Line is a special curve and has more placing options than the two mentioned in the section 2.1. We have already mentioned that holding Ctrl key while positioning the second point keeps the point restricted at certain angles. Moreover, if one primitive (of arbitrary type) is selected while dwawing a line, the angle restricted for Ctrl key is derived from the selected curve.

This can be used to draw a line perpendicular to another one or a line tangent to a curve. If a primitive is selected and you put the first line point on that curve, holding the Ctrl key will restrict to angle of the line with respect to the tangent of the selected curve.

A special and quite frequent task in mechanical engineering is to draw a line which is tangent to two selected curves, usually circles. SteamCAD can do this, even not restricting the curve type to circle. The procedure can be as folows:

1. Select one curve you want to place the line as a tangent.

2. Switch to line mode, and put the first point somewhere on the selected curve.

3. Hold down Ctrl key and move the line so that it gets the required angle. In this case we want the line position to be tangent, but can be any angle available through the angular grid, so the line can also be perpendicular, for example.

4. Once you have the desired angle, press Shift key (while still holding the Ctrl key) and move the line towards the second curve. The line will start following the first curve while keeping the initial angle.

5. When your mouse pointer gets close to the second curve, use the snapping mechanism to find either the tangent or the perpendicular point.

6. Click left mouse button to finish the line.

2.10 Snapping

When designing SteamCAD, the first idea was to put various options for snapping as we can see in other CAD systems, like snap on lines, intersections, line ends etc. In the end I decided that SteamCAD has a fixed snap logic. It snappes to curves, curve ends, intersections, tangent points and perpendicular points. The snap to pure curve element has approximatelly half gravity than the other elements.

Considering that an arbitrary zoom is available in SteamCAD, it gives you an oportunity to snap at the intended point in most situations. When it happens that the snap is impossible - usually due to too many elements close each other, one can right click the unwanted element and select “Disable snap” from the pop-up menu. It is also possible to disable the snap for the whole selection, using the “Alt+N” shortcut.

The disabled snapping can be later restored either by right click or by “Alt+N” command. And the snapping property is not persistent in the model, so saving the document and reopen it will clear all the dissabled snaps.

2.11 View manipulation

As a final topic of this chapter, let’s look at the view manipulation. There are two commands for manipulating the view available in the menu - it is the **Best Fit** and **Normal Size**. The best fit in most drawing applications calculates the minimal bounding box of all objects in the view and zooms to this. Since SteamCAD can contain unbounded objects, there is no meaning for such a command. So the best fit in SteamCAD simple zooms to the whole drawing area (representing the defined paper) with some margin.

The normal size command tries to zoom the view so that it’s physical units correspond to the dimensions on the screen, using the available monitor properties. It works with resonable accuracy on most of the physical monitors, and with less accuraci in virtualized environments.

The other two common view opeartions - zoom and pan - can only be managed by the mouse. This is why a 3-button mouse with wheel is necessary. Zoom is simply performed by rotating the while, while panning is performed by clicking the wheel and dragging the view.

These operations are used so heavily in technical drawing so that it does not make sense to create a menu item for them. Moreover, these zoom and pan actions are available always, it means regardless the drawing mode you are currently in, you can still use the wheel to zoom and pan. The wheel is not used by any other command so it is not in a conflict with anything else, thus always ready to manipulate the view.

Chapter 3

Drawing Tools

Placing the primitives mentioned in the chapter 2 is usually not sufficient to make a meaningfull drawing. So let's discuss other tools avialbale in SteamCAD.

3.1 Selection

Once you finish drawing a primitive, you can escape the drawing mode pressing the "Esc" key and you get into selection mode. The selection mode operates in four ways:

1. Simple selection using mouse click. In this case all selected objects are unselected and the newly clicked object is selected.

2. Multiple selection when holding down the Ctrl key. In this case the clicked object is added to the selected set if it was previsously not selected. Otherwise the object is deselected.

3. Multiple selection with dragging the left mouse button. In this case all the objects fully covered by selection rectangle are selected.

4. Multiple selection with dragging the right mouse button. In this case only the objects partially covered by selection rectangle are selected.

You can combine the dragging rectangle selection with the Ctrl key to add objects to the current selection set.

3.2 Copying Objects

SteamCAD does not utilize clipboard as it is usual for similar applications. The reason for not using clipboard is that SteamCAD simply draw lines, nothing more. It does not define areas, polylines or groups of objects. So there are no Copy and Paste commands. To create a copy of objects, SteamCAD provides four methods more convenient for technical drawing:

1. **Copy Parallel** - this command is available with the shortcat “Ctrl + C”. The selected object is coppied and placed upon left mouse click at a given distance from the original object. The command works for all the primitive types, so it actually creates a geometry equidistant from the original geometry. So for example an ellipse coppied as “parallel” creates a curve which is not an ellipse anymore, except when the distance from the original ellipse is zero. You can also copy parallel already copied objects.

   There is also an edit box in the status bar, where the exact distance can be specified. If it contains valid number, the distance is locked and the mouse only controls on which side from the origin the new object should be copied. Moreover, this only holds for unbound curves. For bound or closed shapes, positive values copy the object to ‘ouside’ while negative values copy the object to ‘inside’. If the edit box is cleared or it contains invalid distance value, the mouse controls both the position and the distance. If the edit box already contains a value from previous commands, you can reuse this value pressing the Enter key.

   Also for Ellipse, Arc Ellipse, Hyperbola and Parabola, it is possible to move the copied object further than is the smallest curve radius. In this case, while holding the Shift key, you can move beyond the curve center. This is especially usefull for Ellipse and Arc Ellipse.

2. **Move** - this command is available with the shortcut “Alt + M”. Once activated, two edit boxes appear in the status bar. You can enter the distance and the number of copies. If you don’t specify the number of copies, or set it to zero, no copy is created and the object is simply moved. Otherwise the number of copies specified is created and the objects are distributed uniformly along the move path.

   If you specify the distance, you would be prompted to select a line to copy the objects along. The positive direction is considered from left to right and from bottom to top. You can also clear the distance edit box, in this case you will be prompted to click two points - one for move “from” and the second for move “to”.

3. **Rotate** - this command is available with the shortcut “Ctrl + R”. Similarly like with move command, you can provide the angle and number of copies. If you don’t specify the angle, you will be later prompted to click two points to rotate “from” and to rotate “to”. The command itself is activated by clicking a point, which is the rotation origin.

   If the angle to rotate is less than 360 degrees, the number of copies is the number specified in the edit box. However, if the angle is equal to 360 degrees, the number of copies is the number in the edit box minus 1. This is because the last copy coincidents with the original. So for example, if you are designing a wheel with 10 spokes, you will draw 1 spoke and then rotate it by 360 degrees with 10 copies. And the results will be exactly what you expect - 10 spokes ingluding the original one, evenly distributed in the wheel rim.

   This is for your convenience and it basically does the same, like rotating one spoke for 360 - 36 degrees with 9 copies.

4. **Mirror** - this command is available with the shortcut "Alt + I". This command is activated by clicking a line object to mirror the selected objects around.

All the copy commands keep the original select set selected.

3.3 Deleting and Undoing Changes

Similarly as with clipboard, SteamCAD implements very little from usual Undo/Redo commands. Here Undo and Redo only applies to deleted objects. You can delete the selected objects simply by pressing the "Delete" key. All other SteamCAD operations can be undone quite easily in a natural way, how we will see later.

3.4 Cutting and Extending Objects

A line spanned accros all the paper is usually not what we would like to see in technical drawings. So the next step is to trim the line to desired length. This is done by activating the **Knife** command ("Alt + K") and clicking the selected primitive at the point you want to cut it. You can cut any number of selected primitives at once, and you can also cut a single primitive as many times as you want. After you finish with cutting, you can enter the selection mode and delete the unwanted parts.

A command complementary to Knife is called **Extend** ("Ctrl + E") and it is actually something like undo for knife. If you realize that a primitive is trimmed at a wrong place, and the line is actually "shorter" than it should be, you can simply activate the Extend command and click the end of the line, which should be extended. The clicked half of the line is then restored to the original, unbounded shape.

3.5 Break apart

This command is available with the shortcut "Alt + B". When you copy one of the curves mention in the section 3.2, item 1, and you will move the copy beyond the smallest curve radius, the curve is not smooth anymore. Invoking this command while such a curve is selected, the curve will be decomposed into smooth parts only. This can be usefull when trying to snap to the curve itself, since SteamCAD does not support anything like self crossing curves.

3.6 Rounding Objects

The last command for creating shapes is called **Round** ("Ctrl + B" - should have been "Bevel" originaly, but later I realized that such a command can easily be replaced by other tools). Selecting any two primitives, the command attempts to place a circular arc tangent to both curves. You can also specify the radius of the arc. Unlike round command in similar CAD system, the

SteamCAD's version does not modify the original selected shapes. It's up to the user to trim them as appropriate.

3.7 Line Shaping

Finally, when you finish drawing your shapes, you may want to specify the line thickness and pattern. You can do this for the whole selected set at once, invoking the **Line Style** dialog ("Alt + S"). Here you can specify the line width, excentricity and pattern. The pattern should contain even number of values, every pair represents the length of the line segment and the length of the hole.

The line excentricity means how much is the line set off its mathematical origin. A typical usage of this parameter is whan drawing rails. Look at the introductory image of the class 6 locomotive - the excentricity for rails is set to 100%. It means that the wheels and dimensions are properly aligned to the rails, while the line is significantly thick to emphasize that this is actually the machine base.

The line thickness can be an arbitrary decimal value. A positive value sets the real line thickness in paper units. Lines with positive thickness are scaled accordingly to the current zoom. A line with thickness zero is always drawn as 1 pixel width, regardless the zoom. Lines with negative thicknes are drawn scaled as positive lines, but only in SteamCAD. Lines with negative thicknes are not exported to the target format. Exports will be discussed later in the section 4.6.

Chapter 4

Advanced SteamCAD Functions

SteanCAD is especially designed to create high quality drawings with precise dimensions at exact scale. Its main purpose is to prepare output for printing or other publishing. To achieve this aim requires few more functions and utilites. Let's look at the rest of the SteamCAD functionality in this chapter.

4.1 Setting Page Dimension, Drawing Scale and Defaults

As was said, SteamCAD output is primarily intended for presentations, that's why each SteamCAD drawing must have a page defined. To do this, open the **File** → **Properties** dialog. This dialog serves to set up all important values for the drawing and should be invoked for each new file.

First you should select the page size from the combo box with predefined values. Initially, A4 - A0 plus US Letter and US Legal paper sizes are available. But SteamCAD is not restricted to those predefined paper sizes. You can add any paper size you like into the DPapers.ini file. Let us remind you that this file should be copied to your home folder, under hidden .SteamCAD subfolder on Linux systems and should be in the folder with SteamCAD.exe on Windows systems. Open the file and examine its structure. It is very simple, it contains list of semicolon separated values. Each line of this file represents one paper size. The first value is the paper name, as it appears in the combo box. The second value is the abbreviation of units used to specify the page dimension. The last two values are actually the width and height of the paper. The width should be less than the height, thus the paper is considered to be a portrait by default. Don't exchange the page dimensions in the DPapers.ini file. The paper orientation should also be set in the File Properties dialog.

You can see in the ini file that two units are used - millimeters and inches. SteamCAD is not restricted to those two units, as we will see later in the section 4.2. For now just remember that the unit used to specify the page size should appear in the DUnits.ini file.

Further you should specify the drawing scale. This is very important, SteamCAD simplifies the drawing creation significantly by not forcing you to recalculate the values from the real world dimensions to the actual scale used for the drawing. You can enter the real machine dimensions

and SteamCAD automatically recalculates them to the drawing scale.

But what to do if you want to enter the value in the paper dimension? For example, your drawing is at the scale 1:87. It is fine that you can enter the machine wheel diameter as real world dimension, but then you want to shift a part of the drawing by 5 mm on the paper? You can switch between paper and real world coordinates by "Ctrl + P", and you can actually check what coordinates are currently used from the **Edit → Paper Units** menu item.

The next thing you need to control are the three unit defaults - world length unit, angular unit and paper length unit. As you guess, these units are used when you enter the precise values in the copy parallel, move, rotate and when creating a line or circle primitive.

Next you need to specify the vertical, horozontal and angular grid. The vertical and horizontal grids are always specified in paper units as they don't represent any real world entity. The grid values are used when placing a primitive while holding the Ctrl key.

Then you need to specify the dimension for graphic elements, such a line thickness, pattern parts, font sizes and dimension arrows. It will most likely be millimeters but other units such as points are common in typography. Finally, specify some default values. Those are values, which can be changed for each element individually.

Regarding the line thicknes, setting it to zero will allow to draw a schema with virtually unlimited complexity. However, since the output is intended for real printing at given scale, it is reasonable to set it to some meaningful positive value. It will later help you to indicate how much details you should put into your drawing.

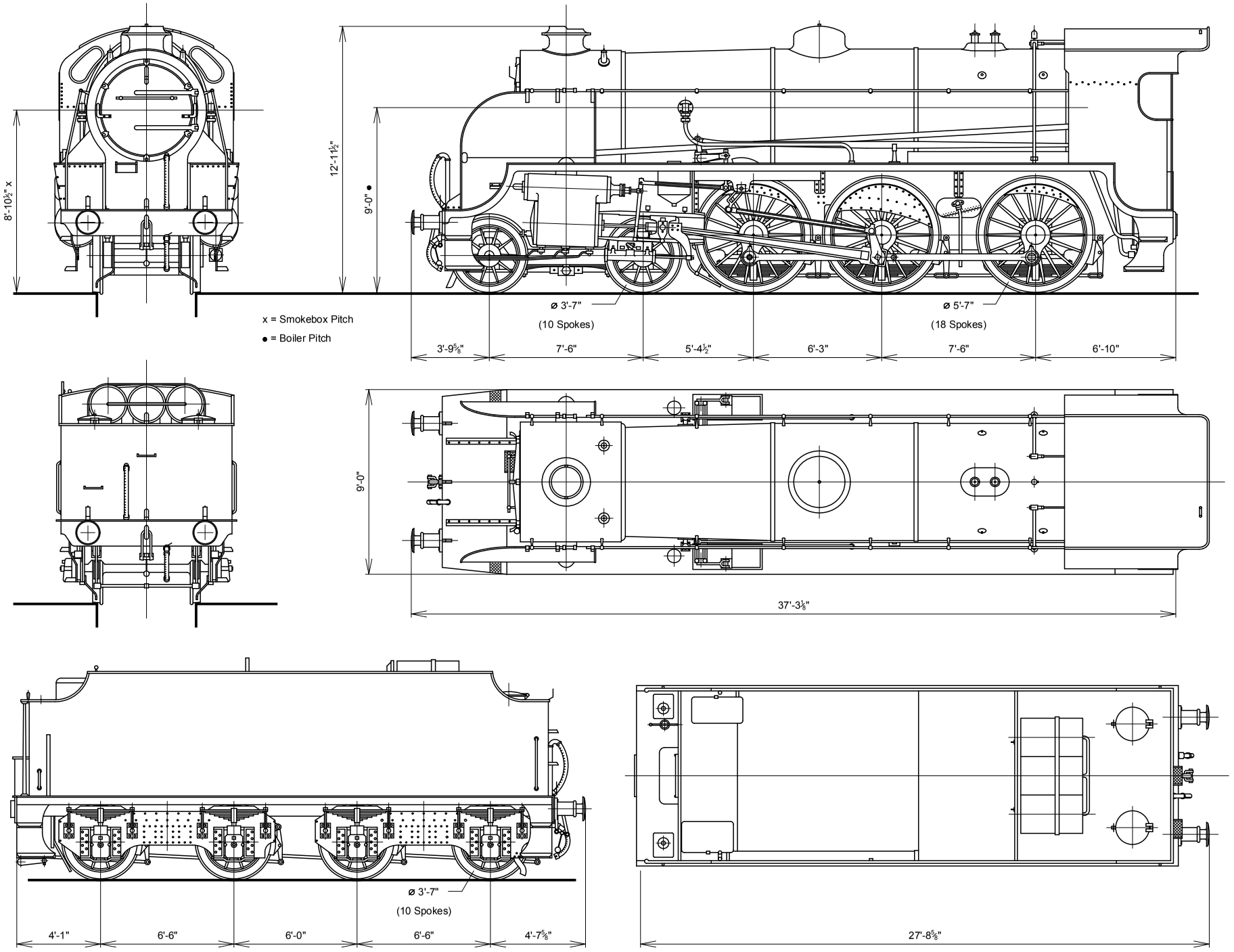
4.2 Units, units and units

Units are the absolute key value of SteamCAD. Units are used both to enter a value and to print a value in form of dimension. Unfortunately, the world is still not unified in terms of using one unit system for all technical drawings. Even if something like that happens in the future, there still would exists drawings of old time machines designed in various unit systems. Look at the drawing[1] on the next page.

This is an example of using imperial unit system in technical drawing. But SteamCAD is not arogant in such a way that it would only support metric and imperial systems. You can use any length and angular unit system you like, you can even design your brand new one.

To do this, open the file DUnits.ini. Let us remind you that this file should be copied to your home folder, under hidden .SteamCAD subfolder on Linux systems and should be in the folder with SteamCAD.exe on Windows systems. The structure of this file is more complex than the structure of DPapers.ini, and it even contains a description explaining the content of this file.

[1]This steam locomotive is actually Southern Railway class S15. It was drawn in SteamCAD at the scale 1:76 following an excellent drawing by Ian Beattie published in Railway Modeller from August 1986.


8'-10½" x
12'-11½"
9'-0" •
x = Smokebox Pitch
• = Boiler Pitch
ø 3'-7"
(10 Spokes)
ø 5'-7"
(18 Spokes)
3'-9⅝"
7'-6"
5'-4½"
6'-3"
7'-6"
6'-10"
9'-0"
37'-3⅛"
ø 3'-7"
(10 Spokes)
4'-1"
6'-6"
6'-0"
6'-6"
4'-7⅝"
27'-8⅝"

To fully understand the unit buzz and its usage, we must go a little bit into technical details. Although we support an arbitrary length and angular unit, there is one length and one angular unit, from which all other units are derived. By no means preffering any unit system, the base length unit was chosen as millimeter and the base angular unit is degree (90 degrees to the right angle). And actually all dimensions stored in the SteamCAD files are internally stored in milimeters. (No angles are stored in SteamCAD files.)

So, if you want to add a new unit, just enter it into DUnits.ini, specify the unit type (length or angle) and specify the unit ratio to the base unit. Only note one irregularity - while when specifying the ratio for a length unit as 1 in = 25.4 mm, the ratio for angular unit is reversed: 1 min = 1/60 deg. This is chosen for convenience - not to be forced to enter numbers with unlimited decimal fraction.

Each unit has a name, an abbreviation, and also an alternate abbreviation. The reason for two abbreviations, or two forms, or two unit symbols is simple. Some unit have a symbol, which we would like to see on the printed output, but this symbol is not common and does not appear on most of the keyboards. So it would be nice if we need to enter the unit name or abbreviation, to enter it in another form, available from most of the keyboards easily. An example of such a unit is degree. While we would like to see the "°" in the output, it is much more convenient to enter "deg" when specifying the unit while drawing.

4.3 Entering Precise Values

As we've seen in the section 4.1, there are three default units which can be set for the drawing. The default unit means that if you enter a value into the edit box without any unit specification, the default unit is used for the given context. However, you are not restricted on entering the values using the default unit. If you want to enter a value in another unit than the default, just specify any unit abbreviation with the value.

Example, you have millimeters as default unit, but you want to enter an imperial dimension - just type 5in, or 5" in the edit box.

And that's not all. The SteamCAD edit box also works as a primitive calculator. So if you know the wheel diameter is 1856 mm, you don't need to divide it by two to enter the circle with the correct radius. Just type 1856/2 in the edit box.

You also don't need to recalculate values to one unit. Just type "10cm 8mm" in the edit box to get the length 18 mm exactly. This is quite simple, but putting 5'-3$^2/_5$" would be more difficult. However, in SteamCAD, you can just enter "5'3+2/5"". But please don't enter it as "5' - 3+2/5"". SteamCAD would interpret the hyphen symbol as minus in this case and would subtract 3$^2/_5$" from 5'.

Also if you need to put 0.33333333333333333, just enter 1/3. If you need 0.66666..., you can enter 2/3 or 1 - 1/3. And you get the fractional value with maximum accuracy. SteamCAD also understands a predefined constant "pi", which is being interpreted as π.

4.4 Dimensioning and Labeling

A technical drawing has very little or no meaning without dimensions. So of course, SteamCAD support dimensioning, however, SteamCAD does it in a way different from other CAD systems. Most of CAD systems let you click three points and generate the whole dimension including the label for you. Not so SteamCAD.

If you want to put a dimension in SteamCAD, you have to draw all the dimension lines and supporting lines yourself, using standard primitives. As I spoke to several engineers, this would be a nightmare for them. But there exists a reason for doing it in this way - SteamCAD is focused on the presentation output, not for creating an asset for manufacturing. So each SteamCAD product should be more artistic work than engineering drawing. It means that the user needs full control on how all the lines are placed.

So to enter a dimension, select any primitive, enter dimension mode by the "D" shortcut, and click two points on the line to place the dimension. As was mentioned, you can select ANY primitive to place the dimension along, not only a line. If you place a dimension on the line, SteamCAD will guess that the dimension will be of length type. If the selected primitive is a circle, SteamCAD will guess that the dimension will be angular. If the primitive is of any other type, SteamCAD will have no guess and place three question marks as the label.

Once you insert a dimension, it is created with default arrows and if it is distance or angle, also with default mask for that dimension. Double clicking the dimension label, you can edit the dimension properties - the arrow types, sizes, label font, label size, and the label content.

The label content is so called dimension mask. The mask can mix both plain text and placeholders for substituting values. The placeholders can be marked either by square brackets [] or by curly brackets {}. If the placeholder is marked by square brackets, it means that the real world value will be substituted. If the placeholder is marked bu curly brackets, the scaled values will be substituted.

The default mask for length label is [D:2], which means that real value in millimeters with two decimal places will be substituted, and no dimension symbol will be placed to the label. The defaul mask for angular dimension is [r:2]° which means that angle in degrees with two decimal places will be substituted, followed by the degree symbol.

If you want the label to represent paper length in milimeters without the unit symbol, change it to {D:2}. If you don't specify the precision, it will be printed with 6 decimal digits. If you specify the precision as zero (:0), it will be printed without decimal places and separator.

You can also specify the precision as "f". In this case, the non integer part will be printed as a fraction with denominator up to 64. So if you want to print labels as on the SR class S15 example, specify the dimension mask as [ft:0]'-[in:f]".

If it happens that SteamCAD does not want to format the label how you wish, you can omit the placeholder and only put a plain text in the mask. In this case it will be printed as it is, with two exceptions - if you want to print 5'-3$^2/_5$" put it as 5'-3_2/5". And if you want to print the symbol for diameter, put asterisk (*) at the begining of the mask (outside all possible placeholders). If the mask starts with asterisk, it will be interpreted as the diameter symbol (similar to Ø).

Finally, note that it is possible to specify the default length and angular mask in the document properties dialog.

You can also move and rotate the dimension labels using the standard Move and Rotate commands. You can also use the dimensions for creating a standalone labels. In this case, create a dummy line, put a dummy dimension on it. Set the line thickness to a negative value, set the dimension arrows to none and set the label content to a plain text.

4.5 Saving and Opening Files

SteamCAD stores the files in its own binary format with extension .sdr. The files should be transferable between Linux and Windows. There are fairly known commands **New**, **Open**, **Save** and **Save as** which create, read and write sdr files. Besides those, there are also **Save Selection...** and **Include...** commands. The first one only saves the objects selected in the current drawing to a new file. This is if you draw a steam engine with a tender and you find that the tender migth be re-used for another locomotive class. The **Include...** command will import an sdr file while keeping the drawing opened. Those two commands somehow replace the missing clipboard.

4.6 Exporting Files

The sdr files only hold the drawing definition, they should not serve as the final SteamCAD output. SteamCAD drawings are supposed to undergo some further processing to get published or presented. That's why SteamCAD implements variety of export formats.

The first one is **PDF** (portable drawing format) format. This is good if you want to quickly review your drawing or if you want to print it. All PDF viewers should implement printing.

Other two export formats are **PS** (postscript) and **EPS** (encapsulated postscript). The first one can be sent directly to a postscript enabled printer. The second one is intended to be included in larger documents. This manual is an example of using EPS files. It is written in LaTeX and the SteamCAD images were included as EPS figures.

The fourth export format is **SVG** (scalable vector graphics), which is an xml based text format natively supported by most of the web browsers. So such a file is suitable for presenting its content on the web. Lot's of drawing programs, such as Inkscape, also allow importing SVG files, so this format can be used for further processing of the image.

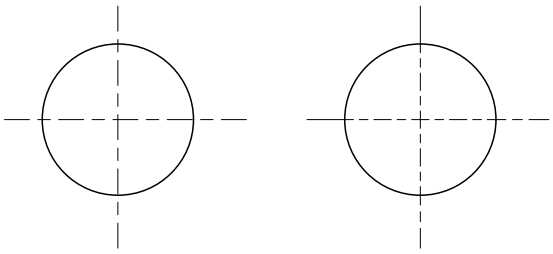
The fifth format is **PNG** (portable network graphics), which is loselessly compressed raster file. The SteamCAD graphics is exported with fixed resolution 600 dpi. The raster file can be further processed in an image manipulation program, such a GIMP.

The last format is **DXF** (drawing exchange format). The implementation of this format is quick and dirty, and it is not intended for serious work. However, the format can be read by most of the CAD systems, so it might be possible to import a SteamCAD drawing into another CAD, but it would certainly require lots of manual adjustments, since DXF format philosophy is far away from the SteamCAD's one.

4.7 Patterned Lines Conflict Resolution

Drawing patterned lines used to have strong rules when the drawing were created by hands. These rules were mostly relaxed with dawn of CADs, since it was too difficult to implement them and too easy to live without them. Aestetics is simply not something to bother about.

However, the printing quality is important for SteamCAD, so some of the rules for placing patterned lines were implemented. Namely, when drawing patterned lines, the author should avoid crossing lines at the voids. Since it is not always possible, SteamCAD does not attempt to detect all such places automatically, but it provides a tool called **Mark Conflicts** (with a shortcut "Ctrl + F") to manually enter places which should be covered by the first element of the line pattern. Please note the FIRST, it means that not the longest element, but the first one in the definition will be attempted to cross the conflict point. So it is advantageous to define the first element of a pattern as the longest one.



The previous figure shows the difference when not specifying conflicts (left image) and with conflicts at the center and circle boundary (right image). As you can see, the line pattern is considered to be a "rubber band" and can be stretched to fulfill the requirements.

You can unmark the conflict by clicking at the conflict position again while in mark conflicts mode. Also when you mark as conflicted an end of the line, only the half of the first element will be drawn at that position. This can be used when creating a petterned line of several smoothly joined curves. Such a join can be observed on the first class 6 locomotive, inside the driver's cab.

4.8 Rescale Images and Change Dimensions

There are few more commands left, which we haven't discussed yet.

1. **Mesure distance** ("Alt + D") - you can measure any distance by clicking two consequent points. The distance and angle is reported on the status bar in the default units and depending on the Paper units switch, they are reported either in world default units, or paper default units.

2. **Tools** → **Rescale and Units** - this command can be used to both change the default length and angular mask for the whole drawing and to change the scale of the whole drawing. To change the mask can be useful if you created a drawing at a given scale, say 1:87, all the dimensions are reported in world distances, and you will find that for modelling purpose, it will be better to know the scaled dimensions. After running the command, all dimensions will be changed to paper units.

To rescale the whole drawing may be handy if you create the drawing say at scale H0 (1:87) and later you find that a drawing at the scale TT (1:120) would also be nice.

3. **Tools** → **Statistics** - this command displays a simple dialog box showing how many elements and what primitive types are used in the drawing.

Chapter 5

Conclusion

5.1 What Next?

Some people may be interested in what are the future plans with SteamCAD. The answer is there are none. SteamCAD is finished software (the only one in the whole computer world?), there are no plans to extend it. It does everything it was supposed to do, if there are bugs in the software, they are now the features of the software.

Well, not quite so. Of course, if the software need some adjustment in the future to work on new operating systems, it will be updated.

I have some plans to use the SteamCAD engine as a base for a 3D CAD, but it would be a different software with different name. And there is definitively no timeframe when this might occurr.

5.2 SteamCAD Update 1

Despite of what was written in the section 5.1, we have published a bunch of patches in November 2018. The patch includes several bug fixes, improved precision when snapping to objects and improved handling of paralel curve copies when the distance goes beyond the smalles curve radius.

It also includes an update of this guide.

5.3 Troubleshooting

5.3.1 What to do if snap does not seem to work?

It may happen that the cursor snaps to one line but strictly ignores another line intersecting the first one. You checked all the lines involved and all have snap enabled. This may happen if for some reason a line is doubled. Try to delete the line, where the snap works to see, if there is not another line at the same place.