Getting Started with Onshape (Second Edition)

Elise Moss

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Onshape is an exciting, new, completely cloud based CAD tool. Getting Started with Onshape is a quick paced guide geared towards users who have no experience in 3D modeling and very little or no experience with AutoCAD. Some experience with a computer and using the Internet is assumed.

Because Onshape can be used for FREE it opens up CAD to anybody who is interested in creating their own models, including members of the bourgeoning Maker community and students who want to learn how to use 3D design tools. Because Onshape is 100% cloud based, there is no software to install and it is always up to date. New features are available to use as soon as they are ready. The good news is that the tools, as outlined in this book, will continue to work the same way even as Onshape evolves.

This book guides you through the very basics of how to create models all the way to exporting to a stl file, which can be used to create a 3D print. Then you can send your stl file to one of many local or online shops that can print out a stl file. When you have completed this book you will have taken the first step to the Maker Faire journey.

In the first chapter of Getting Started with Onshape you will learn how to create an account, explore the workspace and learn how to share your documents with other people. Chapter two features a project where you are guided, step by step, to design your own singlet ring. Throughout this chapter you will learn many of the basic tools you need to use in nearly every project you create. The third chapter features a new project where you create all the parts of a scooter. This project builds on what you learned previously to create more complex designs while new features of Onshape are introduced. In the remaining chapters you will learn how to import parts from other CAD systems, assemble the parts of your scooter and finally create a set of engineering drawings for your scooter.

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