

NOVEDADES SOLIDWORKS 2015



Solid Works Manual 2015

David Planchard



Solid Works Manual 2015:

SOLIDWORKS 2015 in 5 Hours with Video Instruction David Planchard, 2015-02 SOLIDWORKS 2015 in 5 Hours with video instruction introduces the new user to the basics of using SOLIDWORKS 3D CAD software in five easy lessons. This book is intended for the student or designer that needs to learn SOLIDWORKS quickly and effectively for senior capstone machine design kinematics dynamics and other engineering and technology projects that use SOLIDWORKS as a tool. Engineers in industry are expected to have SOLIDWORKS skills for their company's next project. Students need to learn SOLIDWORKS without taking a formal CAD course. Based on years of teaching SOLIDWORKS to engineering students, SOLIDWORKS 2015 in 5 Hours concentrates on the areas where the new user improves efficiency in the design modeling process. By learning the correct SOLIDWORKS skills and file management techniques, you gain the most knowledge in the shortest period of time. You develop a mini Stirling Engine and investigate the proper design intent and constraints. The mini Stirling Engine is based on the external combustion closed cycle engine of Scottish inventor Robert Stirling. In addition to 3D modeling, the engine can be used to teach and connect many engineering and physics principles. You begin with an overview of SolidWorks and the User Interface (UI), its menus, toolbars, and commands. With a quick pace, you learn the essentials of 2D sketching, part and assembly creation, preform motion study, develop detailed part and assembly drawings, and much more. View the provided videos for each section of the book to enhance your experience. SOLIDWORKS Interface, 2D Sketching, Sketch Planes, and Sketch tools, 3D Features, and Design Intent. Creating an Assembly, Fundamentals in Drawings, Part 1, Fundamentals in Drawings, Part 2.

Engineering Design with SolidWorks 2015 and Video Instruction David Planchard, 2014-11-28 Engineering Design with SolidWorks 2015 and video instruction is written to assist students, designers, engineers, and professionals. The book provides a solid foundation in SolidWorks by utilizing projects with step-by-step instructions for the beginner to intermediate SolidWorks user. Explore the user interface, CommandManager, menus, toolbars, and modeling techniques to create parts, assemblies, and drawings in an engineering environment. Follow the step-by-step instructions and develop multiple parts and assemblies that combine machined plastic and sheet metal components. Formulate the skills to create, modify, and edit sketches and solid features. Learn the techniques to reuse features, parts, and assemblies through symmetry, patterns, copied components, Design Tables, Bills of Materials, Custom Properties, and Configurations. Address various SolidWorks analysis tools and Intelligent Modeling techniques along with Additive Manufacturing, 3D printing. Learn by doing, not just by reading. Desired outcomes and usage competencies are listed for each project. Know your objective up front. Follow the steps in Projects 1-9 to achieve the design goals. Review Project 10 on Additive Manufacturing, 3D printing, and its benefits and features. Understand the terms and technology used in low-cost 3D printers. Work between multiple documents, features, commands, and custom properties that represent how engineers and designers utilize SolidWorks in industry. Review individual features, commands, and tools with the Video Instruction. The

projects contain exercises The exercises analyze and examine usage competencies Collaborate with leading industry suppliers such as SMC Corporation of America Boston Gear and 80 20 Inc Collaborative information translates into numerous formats such as paper drawings electronic files rendered images and animations On line intelligent catalogs guide designers to the product that meets both their geometric requirements and performance functionality The author developed the industry scenarios by combining his own industry experience with the knowledge of engineers department managers vendors and manufacturers These professionals are directly involved with SolidWorks every day Their responsibilities go far beyond the creation of just a 3D model The book is designed to compliment the SolidWorks Tutorials contained in SolidWorks 2015 View the provided videos in the book to enhance the user experience SolidWorks Interface 2D Sketching Sketch Planes and Sketch tools 3D Features and Design Intent Creating an Assembly Fundamentals in Drawings Part 1 Part 2 **SOLIDWORKS 2019 Tutorial** David Planchard, 2018-12-21 SOLIDWORKS 2019 Tutorial is written to assist students designers engineers and professionals who are new to SOLIDWORKS The text provides a step by step project based learning approach It also contains information and examples on the five categories in the CSWA exam The book is divided into four sections Chapters 1 5 explore the SOLIDWORKS User Interface and CommandManager Document and System properties simple and complex parts and assemblies proper design intent design tables configurations multi sheet multi view drawings BOMs and Revision tables using basic and advanced features In chapter 6 you will create the final robot assembly The physical components and corresponding Science Technology Engineering and Math STEM curriculum are available from Gears Educational Systems All assemblies and components for the final robot assembly are provided Chapters 7 10 prepare you for the Certified Associate Mechanical Design CSWA exam The certification indicates a foundation in and apprentice knowledge of 3D CAD and engineering practices and principles Chapter 11 covers the benefits of additive manufacturing 3D printing how it differs from subtractive manufacturing and its features You will also learn the terms and technology used in low cost 3D printers Follow the step by step instructions and develop multiple assemblies that combine over 100 extruded machined parts and components Formulate the skills to create modify and edit sketches and solid features Learn the techniques to reuse features parts and assemblies through symmetry patterns copied components apply proper design intent design tables and configurations Learn by doing not just by reading Desired outcomes and usage competencies are listed for each chapter Know your objective up front Follow the steps in each chapter to achieve your design goals Work between multiple documents features commands custom properties and document properties that represent how engineers and designers utilize SOLIDWORKS in industry *Engineering Design with SOLIDWORKS 2018 and Video Instruction* David Planchard, 2018 Engineering Design with SOLIDWORKS 2018 and video instruction is written to assist students designers engineers and professionals The book provides a solid foundation in SOLIDWORKS by utilizing projects with step by step instructions for the beginner to intermediate SOLIDWORKS user featuring machined plastic and sheet metal components

Desired outcomes and usage competencies are listed for each project The book is divided into five sections with 11 projects

Project 1 Project 6 Explore the SOLIDWORKS User Interface and CommandManager Document and System properties simple and complex parts and assemblies proper design intent design tables configurations multi sheet multi view drawings BOMs and Revision tables using basic and advanced features Additional techniques include the edit and reuse of features parts and assemblies through symmetry patterns configurations SOLIDWORKS 3D ContentCentral and the SOLIDWORKS Toolbox Project 7 Understand Top Down assembly modeling and Sheet Metal parts Develop components In Context with InPlace Mates along with the ability to import parts using the Top Down assembly method Convert a solid part into a Sheet Metal part and insert and apply various Sheet Metal features Project 8 Project 9 Recognize SOLIDWORKS Simulation and Intelligent Modeling techniques Understand a general overview of SOLIDWORKS Simulation and the type of questions that are on the SOLIDWORKS Simulation Associate Finite Element Analysis CSWSA FEA exam Apply design intent and intelligent modeling techniques in a sketch feature part plane assembly and drawing Project 10 Comprehend the differences between additive and subtractive manufacturing Understand 3D printer terminology along with a working knowledge of preparing saving and printing CAD models on a low cost printer Project 11 Review the Certified Associate Mechanical Design CSWA program Understand the curriculum and categories of the CSWA exam and the required model knowledge needed to successfully take the exam The author developed the industry scenarios by combining his own industry experience with the knowledge of engineers department managers vendors and manufacturers These professionals are directly involved with SOLIDWORKS every day Their responsibilities go far beyond the creation of just a 3D model

SOLIDWORKS 2018 Tutorial with Video Instruction David Planchard, 2018 SOLIDWORKS 2018 Tutorial with video instruction is written to assist students designers engineers and professionals who are new to SOLIDWORKS The text provides a step by step project based learning approach It also contains information and examples on the five categories to take and understand the Certified Associate Mechanical Design CSWA exam The book is divided into four sections Chapters 1 5 explore the SOLIDWORKS User Interface and CommandManager Document and System properties simple and complex parts and assemblies proper design intent design tables configurations multi sheet multi view drawings BOMs and Revision tables using basic and advanced features In chapter 6 you will create the final robot assembly The physical components and corresponding Science Technology Engineering and Math STEM curriculum are available from Gears Educational Systems All assemblies and components for the final robot assembly are provided Chapters 7 10 prepare you for the Certified Associate Mechanical Design CSWA exam The certification indicates a foundation in and apprentice knowledge of 3D CAD and engineering practices and principles Chapter 11 covers the benefits of additive manufacturing 3D printing how it differs from subtractive manufacturing and its features You will also learn the terms and technology used in low cost 3D printers Follow the step by step instructions and develop multiple assemblies that combine over 100 extruded machined parts and components Formulate the skills to create

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SOLIDWORKS 2020 Tutorial David Planchard, 2019-12 Uses step by step project based tutorials designed for beginning or intermediate users Will prepare you for the Certified SOLIDWORKS Associate Exam Includes a chapter introducing you to 3D printing SOLIDWORKS 2020 Tutorial is written to assist students designers engineers and professionals who are new to SOLIDWORKS The text provides a step by step project based learning approach It also contains information and examples on the five categories in the CSWA exam The book is divided into four sections Chapters 1 5 explore the SOLIDWORKS User Interface and CommandManager Document and System properties simple and complex parts and assemblies proper design intent design tables configurations multi sheet multi view drawings BOMs and Revision tables using basic and advanced features In chapter 6 you will create the final robot assembly The physical components and corresponding Science Technology Engineering and Math STEM curriculum are available from Gears Educational Systems All assemblies and

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SOLIDWORKS 2022 Tutorial David Planchard,2022-04 Uses step by step project based tutorials designed for beginning or intermediate users Will prepare you for the Certified SOLIDWORKS Associate Exam Includes a chapter introducing you to 3D printing SOLIDWORKS 2022 Tutorial is written to assist students designers engineers and professionals who are new to SOLIDWORKS The text provides a step by step project based learning approach It also contains information and examples on the five categories in the CSWA exam The book is divided into four sections Chapters 1 5 explore the SOLIDWORKS User Interface and CommandManager Document and System properties simple and complex parts and assemblies proper design intent design tables configurations multi sheet multi view drawings BOMs and Revision tables using basic and advanced features In chapter 6 you will create the final robot assembly The physical components and corresponding Science Technology Engineering and Math STEM curriculum are available from Gears Educational Systems All assemblies and components for the final robot assembly are provided Chapters 7 10 prepare you for the Certified Associate Mechanical Design CSWA exam The certification indicates a foundation in and apprentice knowledge of 3D CAD and engineering practices and principles Chapter 11 covers the benefits of additive manufacturing 3D printing how it differs from subtractive manufacturing and its features You will also learn the terms and technology used in low cost 3D printers Follow the step by step instructions and develop multiple assemblies that combine over 100 extruded machined parts and components Formulate the skills to create modify and edit sketches and solid features Learn the techniques to reuse features parts and assemblies through symmetry patterns copied components apply proper design intent design tables and configurations Learn by doing not just by reading Desired outcomes and usage competencies are listed for each chapter Know your objective up front Follow the steps in each chapter to achieve your design goals Work between multiple documents features commands custom properties and document properties that represent how engineers and designers utilize SOLIDWORKS in industry

SOLIDWORKS 2016 in 5 Hours with Video Instruction David Planchard,2016

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Engineering Dynamics Labs with SOLIDWORKS Motion 2015 Huei-Huang Lee, 2015-04 This book is designed as a software based lab book to complement a standard textbook in an engineering dynamics course which is usually taught at the undergraduate level This book can also be used as an auxiliary workbook in a CAE or Finite Element Analysis course for undergraduate students Each book comes with a disc containing video demonstrations a quick introduction to SOLIDWORKS eBook and all the part files used in the book This textbook has been carefully developed with the understanding that CAE software has developed to a point that it can be used as a tool to aid students in learning engineering ideas concepts and even formulas These concepts are demonstrated in each section of this book Using the graphics based tools of SOLIDWORKS Motion can help reduce the dependency on mathematics to teach these concepts substantially The contents of this book have been written to match the contents of most mechanics of materials textbooks There are 11 chapters in this book Each chapter contains two sections Each section is designed for a student to follow the exact steps in that section and learn a concept or topic of Engineering Dynamics Typically each section takes 20 40 minutes to complete the exercises Each copy of this book comes with a disc containing videos that demonstrate the steps used in each section of the book a 123 page introduction to Part and Assembly Modeling with SOLIDWORKS in PDF format and all the files readers may need if they have any trouble The concise introduction to SOLIDWORKS PDF is designed for those students who have no experience with SOLIDWORKS and want to feel more comfortable working on the exercises in this book All of the same content is available

for download on the book's companion website [SOLIDWORKS 2018 Quick Start with Video Instruction](#) David Planchard, 2018-02 SOLIDWORKS 2018 Quick Start with video instruction introduces the new user to the basics of using SOLIDWORKS 3D CAD software in five easy lessons. This book is intended for the student or designer that needs to learn SOLIDWORKS quickly and effectively for senior capstone machine design kinematics dynamics and other engineering and technology projects that use SOLIDWORKS as a tool. Engineers in industry are expected to have SOLIDWORKS skills for their company's next project. Students need to learn SOLIDWORKS without taking a formal CAD course. Based on years of teaching SOLIDWORKS to engineering students, SOLIDWORKS 2018 in 5 Hours concentrates on the areas where the new user improves efficiency in the design modeling process. By learning the correct SOLIDWORKS skills and file management techniques, you gain the most knowledge in the shortest period of time. You develop a mini Stirling Engine and investigate the proper design intent and constraints. The mini Stirling Engine is based on the external combustion closed cycle engine of Scottish inventor Robert Stirling. In addition to 3D modeling, the engine can be used to teach and connect many engineering and physics principles. You begin with an overview of SOLIDWORKS and the User Interface (UI), its menus, toolbars, and commands. With a quick pace, you learn the essentials of 2D sketching, part and assembly creation, perform motion study, develop detailed part and assembly drawings, and much more.

Engineering Graphics with SOLIDWORKS 2016 and Video Instruction David Planchard, 2016. Engineering Graphics with SOLIDWORKS 2016 and video instruction is written to assist the technical school, two-year college, four-year university, instructor, student, or industry professional that is a beginner or intermediate SOLIDWORKS user. The book combines the fundamentals of engineering graphics and dimensioning practices with a step-by-step project-based approach to learning SOLIDWORKS with video instructions. Learn by doing, not just by reading. The book is divided into four sections. Chapters 1-3 explore the history of engineering graphics, manual sketching techniques, orthographic projection, Third vs First angle projection, multi-view drawings, dimensioning practices, ASME Y14.5 2009 standard, line type, fit type, tolerance fasteners in general, general thread notes, and the history of CAD leading to the development of SOLIDWORKS. Chapters 4-9 explore the SOLIDWORKS User Interface and CommandManager, Document and System properties, simple machine parts, simple and complex assemblies, proper design intent, design tables, configurations, multi-sheet, multi-view drawings, BOMs, and Revision tables using basic and advanced features. Follow the step-by-step instructions in over 80 activities to develop eight parts, four sub-assemblies, three drawings, and six document templates. Chapter 10 provides a section on the Certified Associate Mechanical Design (CSWA) program with sample exam questions and initial and final SOLIDWORKS models. Chapter 11 provides a section on Additive Manufacturing, 3D printing, and its benefits and features. Understand the terms and technology used in low-cost 3D printers. Review individual features, commands, and tools using the video instruction and SOLIDWORKS Help. The chapter exercises analyze and examine usage competencies based on the chapter objectives. The book is designed to complement the SOLIDWORKS Tutorials located in the

SOLIDWORKS Help menu Desired outcomes and usage competencies are listed for each project Know your objectives up front Follow the step by step procedures to achieve your design goals Work between multiple documents features commands and properties that represent how engineers and designers utilize SOLIDWORKS in industry The author developed the industry scenarios by combining his own industry experience with the knowledge of engineers department managers vendors and manufacturers These professionals are directly involved with SOLIDWORKS every day Their responsibilities go far beyond the creation of just a 3D model

SOLIDWORKS 2021 Tutorial David Planchard, 2020-12 SOLIDWORKS 2021 Tutorial is written to assist students designers engineers and professionals who are new to SOLIDWORKS The text provides a step by step project based learning approach It also contains information and examples on the five categories in the CSWA exam The book is divided into four sections Chapters 1 5 explore the SOLIDWORKS User Interface and CommandManager Document and System properties simple and complex parts and assemblies proper design intent design tables configurations multi sheet multi view drawings BOMs and Revision tables using basic and advanced features In chapter 6 you will create the final robot assembly The physical components and corresponding Science Technology Engineering and Math STEM curriculum are available from Gears Educational Systems All assemblies and components for the final robot assembly are provided Chapters 7 10 prepare you for the Certified Associate Mechanical Design CSWA exam The certification indicates a foundation in and apprentice knowledge of 3D CAD and engineering practices and principles Chapter 11 covers the benefits of additive manufacturing 3D printing how it differs from subtractive manufacturing and its features You will also learn the terms and technology used in low cost 3D printers Follow the step by step instructions and develop multiple assemblies that combine over 100 extruded machined parts and components Formulate the skills to create modify and edit sketches and solid features Learn the techniques to reuse features parts and assemblies through symmetry patterns copied components apply proper design intent design tables and configurations Learn by doing not just by reading Desired outcomes and usage competencies are listed for each chapter Know your objective up front Follow the steps in each chapter to achieve your design goals Work between multiple documents features commands custom properties and document properties that represent how engineers and designers utilize SOLIDWORKS in industry

Engineering Graphics with SOLIDWORKS 2018 and Video Instruction David Planchard, 2017-12-28 Engineering Graphics with SOLIDWORKS 2018 and Video Instruction is written to assist students designers engineers and professionals who are new to SOLIDWORKS The book is divided into four sections Chapters 1 3 explore the history of engineering graphics manual sketching techniques orthographic projection Third vs First angle projection multi view drawings dimensioning practices ASME Y14.5 2009 standard line type fit type tolerance fasteners in general general thread notes and the history of CAD leading to the development of SOLIDWORKS Chapters 4 9 explore the SOLIDWORKS User Interface and CommandManager Document and System properties simple machine parts simple and complex assemblies proper design intent design tables configurations multi sheet multi view drawings BOMs and

Revision tables using basic and advanced features Follow the step by step instructions in over 80 activities to develop eight parts four sub assemblies three drawings and six document templates Chapter 10 provides a section on the Certified Associate Mechanical Design CSWA program with sample exam questions and initial and final SOLIDWORKS models Chapter 11 helps you understand the differences between additive and subtractive manufacturing Comprehend 3D printer terminology along with a working knowledge of preparing saving and printing a 3D CAD model on a low cost printer Review individual features commands and tools using the video instruction and SOLIDWORKS Help The chapter exercises analyze and examine usage competencies based on the chapter objectives The book is designed to complement the SOLIDWORKS Tutorials located in the SOLIDWORKS Help menu Desired outcomes and usage competencies are listed for each project Know your objectives up front Follow the step by step procedures to achieve your design goals Work between multiple documents features commands and properties that represent how engineers and designers utilize SOLIDWORKS in industry The author developed the industry scenarios by combining his own industry experience with the knowledge of engineers department managers vendors and manufacturers These professionals are directly involved with SOLIDWORKS every day Their responsibilities go far beyond the creation of just a 3D model

SolidWorks 2015 Reference Guide David Planchard, 2014-11-02 The SolidWorks 2015 Reference Guide is a comprehensive reference book written to assist the beginner to intermediate user of SolidWorks 2015 SolidWorks is an immense software package and no one book can cover all topics for all users This book provides a centralized reference location to address many of the tools features and techniques of SolidWorks 2015 This book covers the following System and Document properties Feature Managers Property Managers Configuration Managers Render Managers 2D and 3D Sketch tools Sketch entities 3D Feature tools Motion Study Sheet Metal Motion Study SolidWorks Simulation PhotoView 360 Pack and Go 3D PDFs Intelligent Modeling techniques 3D printing terminology and more Chapter 1 provides a basic overview of the concepts and terminology used throughout this book using SolidWorks 2015 software If you are completely new to SolidWorks you should read Chapter 1 in detail and complete Lesson 1 Lesson 2 and Lesson 3 in the SolidWorks Tutorials If you are familiar with an earlier release of SolidWorks you still might want to skim Chapter 1 to become acquainted with some of the commands menus and features that you have not used or you can simply jump to any section in any chapter Each chapter provides detailed Property Manager information on key topics with individual stand alone short tutorials to reinforce and demonstrate the functionality and ease of the SolidWorks tool or feature The book provides access to over 240 models their solutions and additional support materials Learn by doing not just by reading Formulate the skills to create modify and edit sketches and solid features Learn the techniques to reuse features parts and assemblies through symmetry patterns copied components design tables configurations and more The book is designed to compliment the Online Tutorials and Online Help contained in SolidWorks 2015 The goal is to illustrate how multiple design situations and systematic steps combine to

produce successful designs The author developed the tutorials by combining his own industry experience with the knowledge of engineers department managers professors vendors and manufacturers He is directly involved with SolidWorks every day and his responsibilities go far beyond the creation of just a 3D model

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Engineering Statics Labs with SOLIDWORKS Motion 2015 Huei-Huang Lee, 2015 This book is designed as a software based lab book to complement a standard textbook in an engineering statics course which is usually taught at the undergraduate level This book can also be used as an auxiliary workbook in a CAE or Finite Element Analysis course for undergraduate students Each book comes with a disc containing video demonstrations a quick introduction to SOLIDWORKS and all the part files used in the book This textbook has been carefully developed with the understanding that CAE software has developed to a point that it can be used as a tool to aid students in learning engineering ideas concepts and even formulas These concepts are demonstrated in each section of this book Using the graphics based tools of SOLIDWORKS

Motion can help reduce the dependency on mathematics to teach these concepts substantially The contents of this book have been written to match the contents of most statics textbooks There are 8 chapters in this book Each chapter is designed as one week s workload consisting of 2 to 3 sections Each section is designed for a student to follow the exact steps in that section and learn a concept or topic of statics Typically each section takes 15 40 minutes to complete the exercises Each copy of this book comes with a disc containing videos that demonstrate the steps used in each section of the book a 123 page introduction to Part and Assembly Modeling with SOLIDWORKS in PDF format and all the files readers may need if they have any trouble The concise introduction to SOLIDWORKS PDF is designed for those students who have no experience with SOLIDWORKS and want to feel more comfortable working on the exercises in this book All of the same content is available for download on the book s companion website

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Mechanics of Materials Labs with SOLIDWORKS Simulation 2015 Huei-Huang Lee, 2015-03 This book is designed as a software based lab book to complement a standard textbook in a mechanics of material course which is usually taught at the undergraduate level This book can also be used as an auxiliary workbook in a CAE or Finite Element Analysis course for undergraduate students Each book comes with a disc containing video demonstrations a quick introduction to SOLIDWORKS and all the part files used in the book This textbook has been carefully developed with the understanding that CAE software has developed to a point that it can be used as a tool to aid students in learning engineering ideas concepts and even formulas These concepts are demonstrated in each section of this book Using the graphics based tools of SOLIDWORKS Simulation can help reduce the dependency on mathematics to teach these concepts substantially The contents of this book have been written to match the contents of most mechanics of materials textbooks There are 14 chapters in this book Each chapter is designed as one week s workload consisting of 2 to 3 sections Each section is designed for a student to follow the exact steps in that section and learn a concept or topic of mechanics of materials Typically each section takes 15 40 minutes to complete the exercises Each copy of this book comes with a disc containing videos that demonstrate the steps used in each section of the book a 123 page introduction to Part and Assembly Modeling with SOLIDWORKS in PDF format and all the files readers may need if they have any trouble The concise introduction to SOLIDWORKS pdf is designed for those students who have no experience with SOLIDWORKS and want to feel more comfortable working on the exercises in this book All of the same content is available for download on the book s companion website

Reviewing **Solid Works Manual 2015**: Unlocking the Spellbinding Force of Linguistics

In a fast-paced world fueled by information and interconnectivity, the spellbinding force of linguistics has acquired newfound prominence. Its capacity to evoke emotions, stimulate contemplation, and stimulate metamorphosis is actually astonishing. Within the pages of "**Solid Works Manual 2015**," an enthralling opus penned by a very acclaimed wordsmith, readers attempt an immersive expedition to unravel the intricate significance of language and its indelible imprint on our lives. Throughout this assessment, we shall delve into the book's central motifs, appraise its distinctive narrative style, and gauge its overarching influence on the minds of its readers.

<http://www.frostbox.com/results/detail/default.aspx/Study%20Guide%20For%20Ged%20Math%20Test.pdf>

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