

# Bolt Chart ASME B16.5

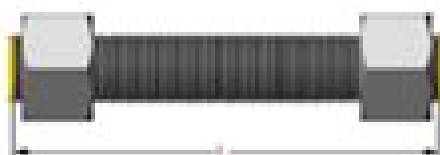
## ASME B16.5 Flanges 150 - 600 lb

Nominal Pipe Size			1/2	3/4	1	1 1/4	1 1/2	2	2 1/2	3	3 1/2	4	5	6	8	10	12	14	16	18	20	24
150 LB. Flanges	Stud Diameter Stud Length # of Studs	1/16 RF	1/2	1/2	1/2	1/2	1/2	5/8	5/8	5/8	5/8	5/8	3/4	3/4	3/4	7/8	7/8	1	1	1 1/8	1 1/8	1 1/4
			2 1/4	2 1/2	2 1/2	2 3/4	2 3/4	3 1/4	3 1/2	3 1/2	3 1/2	3 1/2	3 3/4	4	4 1/4	4 1/2	4 3/4	5 1/4	5 1/4	5 3/4	6 1/4	6 3/4
			4	4	4	4	4	4	4	4	8	8	8	8	8	12	12	12	16	16	20	20
300 LB. Flanges	Stud Diameter Stud Length # of Studs	1/16 RF	1/2	5/8	5/8	5/8	3/4	5/8	3/4	3/4	3/4	3/4	3/4	3/4	7/8	1	1 1/8	1 1/8	1 1/4	1 1/4	1 1/4	1 1/2
			2 1/2	3	3	3 1/4	3 1/2	3 1/2	4	4 1/4	4 1/4	4 1/2	4 3/4	4 3/4	5 1/2	6 1/4	6 3/4	7	7 1/2	7 3/4	8	9
			4	4	4	4	4	8	8	8	8	8	8	8	12	12	16	16	20	20	24	24
600 LB. Flanges	Stud Diameter Stud Length # of Studs	1/4 RF	1/2	5/8	5/8	5/8	3/4	5/8	3/4	3/4	7/8	7/8	1	1	1 1/8	1 1/4	1 1/4	1 3/8	1 1/2	1 5/8	1 5/8	1 7/8
			3	3 1/2	3 1/2	3 3/4	4 1/4	4 1/4	4 3/4	5	5 1/2	5 3/4	6 1/2	6 3/4	7 1/2	8 1/2	8 3/4	9 1/4	10	10 3/4	11 1/4	13
			4	4	4	4	4	8	8	8	8	8	8	8	12	12	16	20	20	20	24	24

## ASME B16.5 Flanges 900 - 2500 lb

Nominal Pipe Size			1/2	3/4	1	1 1/4	1 1/2	2	2 1/2	3	-	4	5	6	8	10	12	14	16	18	20	24
900 LB. Flanges	Stud Diameter Stud Length # of Studs	1/4 RF	3/4	3/4	7/8	7/8	1	7/8	1	7/8	-	1 1/8	1 1/4	1 1/8	1 3/8	1 3/8	1 3/8	1 1/2	1 5/8	1 7/8	2	2 1/2
			4 1/4	4 1/2	5	5	5 1/2	5 3/4	6 1/4	5 3/4	-	6 3/4	7 1/2	7 1/2	8 3/4	9 1/4	10	10 3/4	11 1/4	12 3/4	13 3/4	17 1/4
			4	4	4	4	4	8	8	8	-	8	8	12	12	16	20	20	20	20	20	20
1500 LB. Flanges	Stud Diameter Stud Length # of Studs	1/4 RF	3/4	3/4	7/8	7/8	1	7/8	1	1 1/8	-	1 1/4	1 1/2	1 3/8	1 5/8	1 7/8	2	2 1/4	2 1/2	2 3/4	3	3 1/2
			4 1/4	4 1/2	5	5	5 1/2	5 3/4	6 1/4	7	-	7 3/4	9 3/4	10 1/4	11 1/2	13 1/4	14 3/4	16	17 1/2	19 1/2	21 1/4	24 1/4
			4	4	4	4	4	8	8	8	-	8	8	12	12	12	16	16	16	16	16	16
2500 LB. Flanges	Stud Diameter Stud Length # of Studs	1/4 RF	3/4	3/4	7/8	1	1 1/8	1	1 1/8	1 1/4	-	1 1/2	1 3/4	2	2	2 1/2	2 3/4	-	-	-	-	-
			4 3/4	5	5 1/2	6	6 3/4	7	7 3/4	8 3/4	-	10	11 3/4	13 1/2	15	19 1/4	21 1/4	-	-	-	-	-
			4	4	4	4	4	8	8	8	-	8	8	8	12	12	12	-	-	-	-	-

The length L does not include the height of end points or chamfers. An end point is defined as an unthreaded length, chamfer or bevel, which extends beyond the thread. The bolt lengths are reference dimensions. Users may select other bolting lengths.



**Pressure Control Bolting**  
**ASTM A193 Grade B7 All Thread Studs**  
**ASTM A194 Grade 2H Heavy Hex Nuts**

*An Allied Group Company*

**Sigma Fasteners**  
 16723 Aldine Westfield Road  
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# Flange Bolt Reference Guide

**David C. Planchard, Marie P. Planchard**

A red circular graphic with a gradient, appearing as a stylized arrow or a partial circle, located to the right of the authors' names.

## **Flange Bolt Reference Guide:**

**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 propertiesFeatureManagersPropertyManagersConfigurationManagersRenderManagers2D and 3D Sketch toolsSketch entities3D Feature toolsMotion StudySheet MetalMotion StudySolidWorks SimulationPhotoView 360Pack and Go3D PDFsIntelligent Modeling techniques3D 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 PropertyManager 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

**SolidWorks 2016 Reference Guide** David Planchard,2015-12-16 The SOLIDWORKS 2016 Reference Guide is a comprehensive reference book written to assist the beginner to intermediate user of SOLIDWORKS 2016 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 2016 This book covers the following System and Document propertiesFeatureManagersPropertyManagersConfigurationManagersRenderManagers2D and 3D Sketch toolsSketch entities3D Feature toolsMotion StudySheet MetalMotion StudySolidWorks SimulationPhotoView 360Pack and Go3D PDFsIntelligent Modeling techniques3D printing terminology and more Chapter 1 provides a basic overview of the concepts and terminology used throughout this book using SOLIDWORKS 2016 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 PropertyManager 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 2016 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

*SOLIDWORKS 2018 Reference Guide* David Planchard, 2018-01-29 The SOLIDWORKS 2018 Reference Guide is a comprehensive reference book written to assist the beginner to intermediate user of SOLIDWORKS 2018 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 2018 This book covers the following System and Document properties FeatureManagers PropertyManagers ConfigurationManagers RenderManagers 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 2018 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 PropertyManager 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 250 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 complement the Online Tutorials and Online Help contained in SOLIDWORKS 2018 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      SOLIDWORKS 2017 Reference Guide David Planchard, 2017 The SOLIDWORKS 2017 Reference Guide is a comprehensive reference book written to assist the beginner to intermediate user of SOLIDWORKS 2017 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 2017 This book covers the following System and Document properties FeatureManagers PropertyManagers ConfigurationManagers RenderManagers 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 2017 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 PropertyManager 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 250 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 2017 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      *Mechanical Engineer's Reference Book* A. Parrish, 2014-05-19 Mechanical Engineer's Reference Book 11th Edition presents a comprehensive examination of the use of System International Units SI metrication It discusses the effectiveness of such a system when used in the field of engineering It addresses the basic concepts involved in thermodynamics and heat transfer Some of the topics covered in the book are the metallurgy of iron and steel screw threads and fasteners hole basis and shaft basis fits an introduction to geometrical tolerancing mechanical working of steel high strength alloy steels advantages of making components as castings and basic theories of material properties The definitions and classifications of refractories are fully covered An in depth account of the mechanical properties of non ferrous materials is provided Different fabrication techniques are completely presented A chapter is devoted to description of tubes for water gas sanitation and heating services Another section focuses on the accountant's measure of productivity The book can provide useful information to

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by step instructions that are perfect for both beginners and intermediate users Each project begins with desired outcomes and usage competencies so you ll know exactly what you ll learn and how to apply it Projects build your skills incrementally Throughout the book you ll learn to create machined plastic and sheet metal components explore the SOLIDWORKS user interface CommandManager and document and system properties You ll discover how to design simple and complex parts and assemblies with proper design intent You ll also explore how to use the SOLIDWORKS Toolbox and symmetry patterns and configurations to edit and reuse features and parts like the pros do And that s just the first six projects Next you ll investigate top down assembly modeling develop components in context with InPlace Mates convert a solid part into sheet metal and insert and apply sheet metal features With projects 8 and 9 you ll learn how to apply intelligent modeling techniques to a sketch feature or any SOLIDWORKS creation Prepare for the SOLIDWORKS Simulation Associate Finite Element Analysis CSWSA FEA exam with an overview of SOLIDWORKS Simulation important concepts and practice exam questions Plus bonus material in projects 10 and 11 describes the differences between additive and subtractive manufacturing and everything you need to know about 3D printing and the Certified SOLIDWORKS Associate Exam CSWA You will be delighted to find this is not just a dry technical manual The realistic project scenarios were created with the author s industry expertise and input of engineers department managers vendors and manufacturers who use SOLIDWORKS every day Whether you re looking to enhance your career or simply want to expand your knowledge of SOLIDWORKS Engineering Design with SOLIDWORKS 2025 is the ideal resource for you Includes a Bonus eBook Covering SOLIDWORKS and 3DEXPERIENCE Platform Included with your purchase of this book is a bonus eBook titled SOLIDWORKS and the 3DEXPERIENCE Platform This eBook is an insightful guide that introduces you to the 3DEXPERIENCE Platform and its integration with SOLIDWORKS This resource simplifies complex concepts allowing users to collaborate efficiently in a single modeling environment accessible through the SOLIDWORKS Task Pane The book features nine detailed step by step tutorials complete with models to practice and understand the tools and advantages of using SOLIDWORKS with the 3DEXPERIENCE platform This guide will help you understand the 3DEXPERIENCE Platform s capabilities demonstrating practical real world applications in educational and professional settings It s an essential resource for anyone looking to leverage the full potential of SOLIDWORKS in conjunction with the 3DEXPERIENCE platform Table of Contents Introduction 1 Overview of SOLIDWORKS and the User Interface 2 Fundamentals of Part Modeling 3 Fundamentals of Assembly Modeling 4 Fundamentals of Drawing 5 Extrude and Revolve Features 6 Swept Lofted and Additional Features 7 Top Down Assembly Modeling and Sheet Metal Parts 8 SOLIDWORKS Simulation 9 SOLIDWORKS and the 3DEXPERIENCE platform Appendix Glossary Index Bonus Chapters 10 Additive Manufacturing 3D Printing 11 Introduction to the Certified Associate Mechanical Design CSWA Exam

**Engineering Design with SOLIDWORKS 2024** David Planchard,2024-03-25 A comprehensive introduction to SOLIDWORKS using tutorial style step by step instructions Designed for beginning or intermediate

SOLIDWORKS users Learn to create parts and assemblies using machined plastic and sheet metal components Also covers Simulation Sustainability and Intelligent Modeling techniques Includes bonus chapters on the CSWA exam and 3D printing This edition features a new chapter and a bonus eBook on SOLIDWORKS and the 3DEXPERIENCE platform Are you looking to learn SOLIDWORKS As luck would have it you have found the perfect SOLIDWORKS resource for students designers engineers and professionals alike Engineering Design with SOLIDWORKS 2024 provides a solid foundation in SOLIDWORKS by using projects with step by step instructions that are perfect for both beginners and intermediate users Each project begins with desired outcomes and usage competencies so you ll know exactly what you ll learn and how to apply it Projects build your skills incrementally Throughout the book you ll learn to create machined plastic and sheet metal components explore the SOLIDWORKS user interface CommandManager and document and system properties You ll discover how to design simple and complex parts and assemblies with proper design intent You ll also explore how to use the SOLIDWORKS Toolbox and symmetry patterns and configurations to edit and reuse features and parts like the pros do And that s just the first six projects Next you ll investigate top down assembly modeling develop components in context with InPlace Mates convert a solid part into sheet metal and insert and apply sheet metal features With projects 8 and 9 you ll learn how to apply intelligent modeling techniques to a sketch feature or any SOLIDWORKS creation Prepare for the SOLIDWORKS Simulation Associate Finite Element Analysis CSWSA FEA exam with an overview of SOLIDWORKS Simulation important concepts and practice exam questions Plus bonus material in projects 10 and 11 describes the differences between additive and subtractive manufacturing and everything you need to know about 3D printing and the Certified SOLIDWORKS Associate Exam CSWA You will be delighted to find this is not just a dry technical manual The realistic project scenarios were created with the author s industry expertise and input of engineers department managers vendors and manufacturers who use SOLIDWORKS every day Whether you re looking to enhance your career or simply want to expand your knowledge of SOLIDWORKS Engineering Design with SOLIDWORKS 2024 is the ideal resource for you Includes a Bonus eBook Covering SOLIDWORKS and 3DEXPERIENCE Platform Included with your purchase of this book is a bonus eBook titled SOLIDWORKS and the 3DEXPERIENCE Platform This eBook is an insightful guide that introduces you to the 3DEXPERIENCE Platform and its integration with SOLIDWORKS This resource simplifies complex concepts allowing users to collaborate efficiently in a single modeling environment accessible through the SOLIDWORKS Task Pane The book features nine detailed step by step tutorials complete with models to practice and understand the tools and advantages of using SOLIDWORKS with the 3DEXPERIENCE platform This guide will help you understand the 3DEXPERIENCE Platform s capabilities demonstrating practical real world applications in educational and professional settings It s an essential resource for anyone looking to leverage the full potential of SOLIDWORKS in conjunction with the 3DEXPERIENCE platform Engineering Design with SOLIDWORKS 2022 David Planchard,2022-02 A comprehensive introduction to SOLIDWORKS using tutorial style step by



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*Engineering Design with SOLIDWORKS 2021* David Planchard, 2021 *Engineering Design with SOLIDWORKS 2021* 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

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Engineering Design with SolidWorks 2010 and Multimedia CD David C. Planchard, Marie P. Planchard, 2010-01-11

Provides an introduction to engineering design using SolidWorks 2010 through step by step tutorials that cover such topics as part modeling assembly modeling drawing extrude and revolve features and top down assembly modeling *Cruising World*, 1980-01 **Engineering Design with SolidWorks 2014 and Video Instruction** David Planchard, 2014

Engineering Design with SolidWorks 2014 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 SimulationXpress Sustainability SustainabilityXpress and DFMXpress and Intelligent Modeling techniques 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 Project 1 8 to achieve the design goals 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 design to compliment the SolidWorks Tutorials contained in SolidWorks 2014

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