新世纪高校机械工程规划教材

GAOXIAO GUIHUA JIAOCAO

机械工程英语

(下)

主编　张宇

机械工业出版社
CHINA MACHINE PRESS

新世纪高校机械工程规划教材

机械工程英语（下）

主　编　张　宇

副主编　王优强　高　军

参　编　邢希东　刘敏杰

机 械 工 业 出 版 社

本书分上下两册出版。上册主要涉及机械工程材料及热处理、各种热加工工艺、机械设计、金属切削机床、液压系统与元件、金属切削原理、工装设计、特种加工技术、英语应用文；下册主要涉及公差配合与计量测控、质量控制与质量体系、计算机辅助设计与制造、自动化、工业机器人、计算机集成制造和柔性制造系统、先进制造技术及其支持技术，并对科技文献的三大检索工具和有影响的主要机械工程英文期刊做了介绍。本书引用资料新颖，内容涵盖面较广，既对学生学过的内容进行必要的覆盖，也有所扩展和延伸，使学生不仅提高专业英语阅读能力，也调动学生通过英文载体了解机械工程和制造技术的现状与发展趋势。上册有阅读和口语会话练习；下册重在阅读能力的培养。全书图文并茂，适合于机械设计制造及其自动化、机电一体化、材料成形及控制、金属材料专业的本科及高职高专学生使用，对机械工程技术人员提高专业阅读能力亦不失为一本有价值的学习教材或参考书。

图书在版编目（CIP）数据

机械工程英语/刘镇昌，张宇主编. 一北京：机械工业出版社，2003.2

ISBN 7-111-11278-4

Ⅰ.机… Ⅱ.①刘… ②张… Ⅲ.机械工程－英语－高等学校－教材 Ⅳ.H31

中国版本图书馆 CIP 数据核字（2003）第 001777 号

机械工业出版社（北京市百万庄大街 22 号　邮政编码 100037）

策 划 人：高文龙　王世刚

责任编辑：茹雪飞

封面设计：姚　毅　　　责任印制：路　琳

北京蓝海印刷有限公司印刷·新华书店北京发行所发行

2003 年 1 月第 1 版第 1 次印刷

1000mm×1400mm B5·11.5 印张 ·437 千字

0 001—3 000 册

定价：29.00 元（上下册）

新世纪高校机械工程规划教材
编审委员会

前　言

本教材是按照山东高校机械工程类规划教材编审会 2001 年 12 月会议提出的系列教材出版计划组织编写的。

本教材定名为《机械工程英语》，分上、下两册出版。每册含 6 个单元，每个单元含 4 课，每课供 2 学时使用。全书上、下册全部讲完需要 96 学时。各专业可以适当节选内容，以满足各自不同的教学要求。上册主要涉及机械工程材料及热处理、各种热加工工艺、机械设计、金属切削机床、液压系统与元件、金属切削原理、工装设计、特种加工技术、英语应用文（履历书、协议书、研究报告、讲演稿）；下册主要涉及公差配合与计量测控、质量控制与质量体系、计算机辅助设计与制造、自动化、工业机器人、先进制造技术及其支持技术，并对科技文献的三大检索工具和有影响的主要机械工程英文期刊做了介绍。

本书引用资料新颖，内容涵盖面广。上册有阅读和会话练习；下册重在阅读能力的培养；每课均配有练习题。可供机械设计制造及其自动化、机电一体化、材料成形及控制、金属材料专业的本科及高职高专学生在学完基础英语以后使用；对从事上述专业的工程技术人员和管理者，亦不失为一本有价值的机械工程英语学习教材或参考书。

本书课文全部节选自欧美文献原著。为保持原著的语言风格，编者对选材一般只作删节，不作改写。本书的图不符合机械制图国家标准，但为使学生熟悉美、英等国目前仍然使用的英制单位和工程图习惯，对原著中采用的各种计量单位及图样标均不作改动。

参加本书编写的共有山东大学、山东理工大学、山东轻工业学院、青岛建筑工程学院、山东建筑工程学院、山东科技大学共六所院校的十一位老师。上册编写人员为山东轻工业学院王仁人（Unit 1）、山东建筑工程学院荆海鸥（Unit 2）、山东大学刘战强（Unit 3）、山东科技大学王吉岱（Unit 4、5）、山东建筑工程学院刘喜俊（Unit 4、5）、山东大学刘镇昌（Unit 6）。下册各单元参编人员为山东理工大学张宇（Unit 1 Lesson1～3、Unit 2）、山东理工大学高军（Unit 1 Lesson 4、Unit 3）、青岛建筑工程学院刘敏杰（Unit 4）、山东理工大学邢希东（Unit 5）和青岛建筑工程学院王优强（Unit 6）。

安徽技术师范学院项宏萍教授参加了上册 Unit 6 的大部分撰稿工作；加

拿大来华的外籍教师 Keith Klockow 先生和美国的 Paul Erik Jacobson 工程师对上册会话和练习部分进行了审校，特别是 Paul Erik Jacobson 工程师非常尽心。他们的付出对本教材质量的提高起了很大作用，特此表示衷心感谢。

受编者水平所限，本书恐难免谬误，敬请读者批评指正。

刘镇昌　张宇

2002 年 10 月

目　录

前言

Unit 1　Geometric Tolerancing and Metrology ········ *1*

Lesson 1　Geometric Dimensioning and Tolerancing ········ *1*

Lesson 2　Fundamentals of Metrology ········ *10*

Lesson 3　Electronic Gaging ········ *16*

Lesson 4　Applications of Electronic Instrument Systems ········ *24*

Unit 2　Quality Control and Quality Systems ········ *32*

Lesson 5　Quality Control Fundamentals ········ *32*

Lesson 6　Basic Quality Problem Solving ········ *41*

Lesson 7　Process Capability Analysis ········ *48*

Lesson 8　ISO 9000 and ISO 14000 ········ *57*

Unit 3　Computerized Manufacturing ········ *66*

Lesson 9　The Evolution of CAD/CAM ········ *66*

Lesson 10　Computer Aided Design ········ *76*

Lesson 11　Computer Aided Process Planning ········ *85*

Lesson 12　Computer Aided Manufacturing ········ *94*

Unit 4　Automation ········ *103*

Lesson 13　Automation of Manufacturing Processes ········ *103*

Lesson 14　Industrial Robots ········ *110*

Lesson 15　Computer-integrated Manufacturing ········ *117*

Lesson 16　Flexible Manufacturing Systems ········ *123*

Unit 5　Advanced Manufacturing Technology ········ *128*

Lesson 17　Agile Manufacturing ········ *128*

Lesson 18　Rapid Prototyping and Manufacturing ········ *135*

Lesson 19　Nanotechnology ········ *140*

Lesson 20　Introductions to Artificial Intelligence ········ *146*

Unit 6 Supporting Techniques ········ *153*

Lesson 21 Group Technology ········ *153*

Lesson 22 Concurrent Engineering ········ *159*

Lesson 23 Cost Analysis ········ *166*

Lesson 24 Introduction to Mechanical Engineering Literature ········ *174*

参考文献 ········ *182*

Unit 1 Geometric Tolerancing and Metrology

Lesson 1 Geometric Dimensioning and Tolerancing

Introduction

What is Geometric Dimensioning and Tolerancing (GD&T)?

GD&T is a three dimensional international engineering language used on mechanical drawings. This language consists mainly of symbols that are clearly defined in ASME Y14.5M-1994 put out by the American Society of Mechanical Engineers. This is the drawing standard used in North America and recognized throughout the world. It replaces the earlier ANSI Y14.5M1982 standard and has been expanded to be almost identical to its ISO counterpart. The standards are complete in identifying ways of using the various geometric symbols and other methods to clearly show the designer's intent.

Why Do We Need Geometric Dimensioning and Tolerancing?

Proper use of GD&T guarantees the form, fit and function that engineering intended without assumptions in the shop or elaborate notes that everyone interprets differently. GD&T will save your company money by providing consistent interpretations, increasing manufacturing tolerances and promoting efficiency and quality throughout the engineering, manufacturing and quality functions. Our experience has revealed that many designers, shop and quality control personnel, although working with GD&T over several years, do not fully understand the requirements and/or do not utilize all the benefits of GD&T.

Design and production systems, complexity, computerization, and global manufacturing have made exacting engineering drawing requirement mandatory. Functional gaging, tools, components dimensions, and manufacturing benefit with GD &T. The study of GD&T is important, because it is the communication glue among design, manufacturing processes, and quality.

Manufacturing and engineering systems require a language that is understandable; otherwise, it is not coherent and usable. A technical language is defined by a "standard", and one that is widely used is ASME Y14.5M1994. Our purpose is to harmonize GD & T with manufacturing process. You may have been exposed to GD&T in a CAD or drafting class. This unit gives manufacturing understanding to these symbols.

Dimension and Tolerance

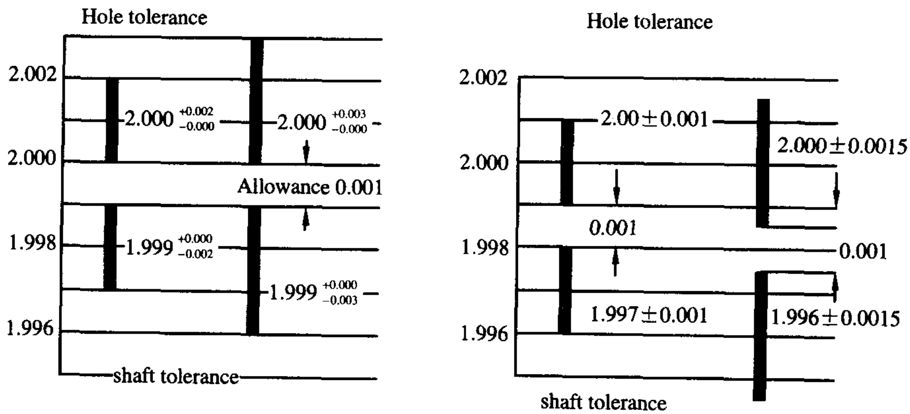
Size and location of a feature are determined by a dimension. Dimensions are seen in engineering drawings, which represent a number that is approximated in actual manufacturing. A shaft may have a nominal size of $2\frac{1}{2}$ in. (63.5 mm), but the exact decimal equivalent measurement is unobtainable in manufacturing. Although fractional dimensions are found in older drawings, decimal equivalents are preferred practice. Typically, it is unlikely that manufacturing is able to produce parts to 2.500 in., because there is natural in manufacturing.

Tolerance is the amount of variation permitted in the dimension. All dimensions have tolerance, either specific or general (which is stated in the title box). A dimension is a joint number, which includes a basic dimension value with its tolerance. A tolerance considers functional engineering and manufacturing requirements. For example, a dimension could be 2.500 ± 0.005, or 2.5 ± 0.0005. The dimension given in thousandths or tenththousandths implies lesser or greater accuracy.

Tolerance is the range of allowable dimensions, knowing that a dimension is never exact, such as 2.50000 in. A tolerance is similar to the term "significant digits". It provides the machinist with a cutoff number that is achievable. Tolerances may be "tight" or have too many significant digits. With a tight tolerance, cost rapidly increases.

Dimensions are given as limit, unilateral, or bilateral. An example of limit dimensioning is $\begin{matrix}3.505\\3.495\end{matrix}$. A limit dimension has the larger value on top, and the fractional line is omitted between the two values. Limit methods of dimensioning are preferred practice. Bilateral dimensioning form is 2.500 ± 0.005, where both tolerance values are equal in this case. Bilateral practice does not require both tolerance values to be the same. A special case is unilateral, such as $3.495^{+0.010}_{-0.000}$, where the lower value is zero. Unilateral tolerance means that any variation is made in only one direction from the nominal dimension.

Manufacturing deals with fitting and mating parts, a shaft in a hole, for example. Allowance, which is sometimes confused with tolerance, is defined differently. It is the minimum clearance space intended between mating parts and represents the condition of tightest possible fit. Notice Figure 1.1a, where unilateral tolerances are used. For the two cases in Figure 1.1a, the allowance is 0.001. Similarly in Figure 1.1b, the use of bilateral tolerances shows an allowance of 0.001. An advantage of the unilateral tolerance permits changing the tolerance while still retaining the same allowance. A force fit or an interference fit between mating parts requires the tolerance to have a zero or negative allowance.



a)　　　　b)

Figure 1.1 Tolerance and allowance

a) unilateral　　b) bilateral

Symbols

Geometric dimensioning and tolerancing symbols given by Figure 1.2. These symbols replace many of the notes on engineering drawings. Advantages of symbols over notes are given as:

- Uniform and defined meaning. A note may be stated inconsistently.
- Compactness.
- An international language. Notes may require translation between languages.
- Manufacturing clarity.

Some of these symbols are seen in Figure 1.3. Study this machined part, where dimensions are in millimeters.

Maximum material condition (MMC) or Ⓜ deals with hole and shaft diameters and similar features.

Symbol	Characteristic
⏥	Flatness
—	Straightness
○	Circularity (roundness)
⌭	Cylindricity
⌒	Profile of a line
⌓	Profile of a surface
⊥	Perpendicularity (squareness)
∠	Angularity
//	Parallelism
↗	Circular runout
⌰	Total runout
⌖	Position
◎	Concentricity
⌯	Symmetry
Ⓜ	Maximum material condition MMC
Ⓛ	Least material condition LMC
ϕ	Diametrical (cylindrical) tol zone or feature
[40.5]	Basic, or exact, dimension
-[A]-	Datum feature symbol
[⌖ | ϕ0.2 Ⓜ | A]	Feature control frame

Figure 1.2 Geometric characteristics and terms

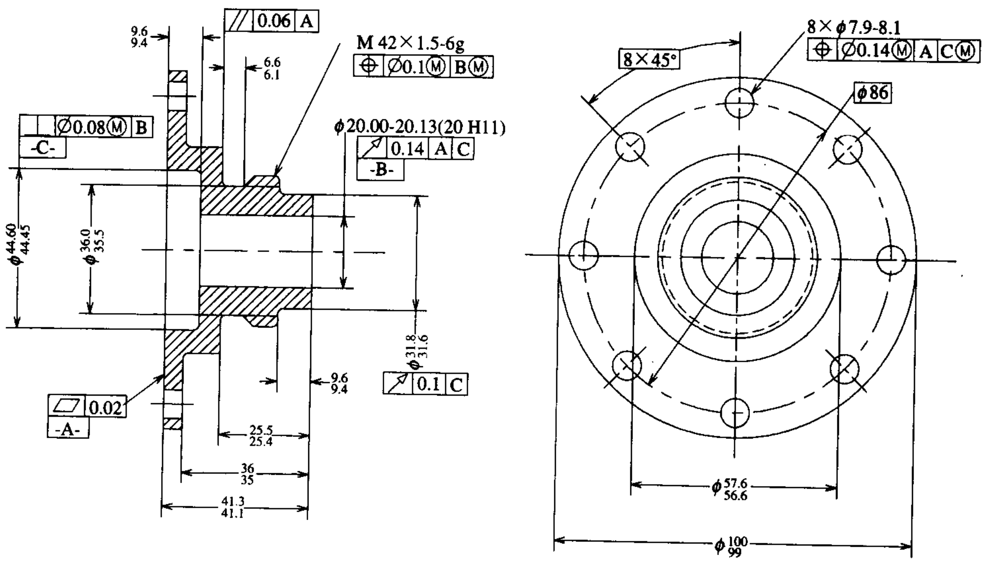


Figure 1.3 Sample GRT drawing

Ⓜ enclosed in the circle is the drawing symbol, although MMC is used in discussion. A hole that is given as 2.500±0.005 in. ranges from a maximum hole size of 2.505 to a minimum diameter of 2.495 in. The hole at its low limit contains more material than its high limit.

Thinking for MMC is opposite for shafts. For example, if the shaft diameter is 0.750±0.004 in., the high limit of the tolerance, when applied to the shaft, given 0.754 in., which is the MMC for the shaft, that having more material than the low limit. Generally, MMC allows greater possible tolerances. It aids interchangeability and permits functional gaging techniques. The MMC implies worst condition or critical size, and relates mating part. The MMC principle applies to features of size, such as a hole, slot, or pin with an axis or center plane.

Least material condition (LMC) or the drawing symbol Ⓛ is the condition in which a feature of size contains the least amount of material within the stated limits of size. It is the maximum hole diameter or minimum shaft diameter.

Regardless of feature size (RFS) indicates that geometric tolerance or datum reference applies at any increment of size of the feature within its size tolerance. It indicates tolerance apply to a geometric feature regardless of size, from MMC to LMC. RFS permits no additional positional, form, or orientation tolerance. The RFS applies only to features of size, such as a hole, slot, and pin with an axis or center plane. The RFS is implied, unless otherwise specified under MMC rules.

The term **basic** is a numerical value and dimension that describe the exact size, profile, orientation, or location of a feature or datum. It is the dimensional value that requires dimension be enclosed in a feature control frame (Figure 1.2 and 1.3). It is emphasized that not all dimensions are enclosed in frames. Only the important ones are framed and that depends on the engineering design requirements.

A datum is the theoretically exact point, plane, or axis that is the origin from which the location or geometric characteristics of features of a component are established. Each datum is featured in a frame and is identified by capital alphabetic letter.

For a tolerance of position, the datum reference letter is followed by the appropriate modifying symbol in the feature control frame. For all other geometrical symbols, RFS is implied, unless otherwise specified. Where a datum feature of size is applied on an RFS basis, the datum is established by physical contact between the feature surface or surfaces and surfaces of the processing equipment. Machine elements, which are variable in size (chuck, mandrel, or centers), are used to simulate a true geometric counterpart of the feature to establish the datum.

CAD and CAM

Computeraided design (CAD) and computeraided manufacturing (CAM) are the systems that describe the part as a geometric model, and interactively adding manufacturing data, deliver this information to a manufacturing system or machine tool to complete the part. Geometrical dimensioning and tolerancing is the practice that allows the dimensioning of the part into the manufacturing of the object.

The coordinate system is the same for the geometric model created by CAD and the CAM system that produces the part. The rectangular or Cartesian coordinates locate a point by its distance from each two or three mutually perpendicular intersecting planes. Twodimensional coordinate (in X and Y direction) locate a point on a plane. Three pointing the coordinate XYZ direction locate a point in space. Once a point is defined, there is opportunity for commands to the machine tool, which execute the required relative motion between the cutting tool and the part. Linear, rotary and composite machine tool motions are possible.

Appropriate Tolerances

When an engineer specifies dimensions for a part that is to be produced, it is necessary to include the tolerance along with the dimension. Tolerances should be specified from actual design requirements but with manufacturing capabilities clearly in mind. They should not be specified more accurately, even though the CAM equipment is capable of the greater accuracy. By knowing appropriate tolerances, an engineer is able to design a costeffective part.

Tolerances may be expressed by general notes printed on the drawing immediately with the dimension such as 2.500 ± 0.005, or in the title box, which is applied to those dimensions. For example, if the dimension is 2.500 in., and there is no specified tolerance with the dimension, then the tolerance that is applied is based on the 0.xxx significant thirdplace digit, as given in the title box. An illustrative example of general title block tolerances for customary dimensions given in inch units is given as

2 places, 0.xx	±0.010 in.
3 places, 0.xxx	±0.005 in.
4 places, 0.xxxx	±0.0005 in.

For a 2.240in. dimension, the range of allowable dimensions is 2.235 to 2.245 in. Most dimensions on a drawing use the general title block tolerances.

The tolerance may be applied directly to the dimension and appears between the dimensional arrows on the drawing. In this case the tolerance is not removed from the title block, but is given with the dimension.

Companies will determine appropriate tolerances that are suitable for their manufacturing equipment and product. An example of effective tolerances for a college or school shop, where the equipment is maintained, is given by Table1.1.

Table 1.1 Sample tolerance for traditional machine tools

Machine	Condition	Tolerance
Mill	Minimum geometric	0.005 in.
	Surface roughness (side cut)	32 in.
	Surface roughness (end cut)	63 μin.
	Straightness	0.005 in.
	Flatness	0.005 in.
	Parallelism (same side)	0.005 in.
	Parallelism (opposite side)	0.015 in.
	Angularity	0.1
	Perpendicularity	0.015 in.
Lathe	Minimum geometric	0.002 in.
	Surface roughness	16 μin.
	Concentricitys	0.005 in.
Grinder	Minimum geometric	0.001 in.
	Surface roughness	4 μin.
	Flatness (plate>1 in. long)	0.002 in.
	Flatness (plate <1 in. long)	0.001 in.

For example, the milling machine used for metal removal of parts that are not a surface of revolution will have a standard of accuracy of 0.001 in. Rigid machines may be capable of 0.0005 in. If deflection, heat expansion, nonrigid part, tool wear, inadequate tooling, and the like, are significant, the rule of thumb considers the appropriate tolerance as 0.005 in.

The lathe is the standard machine tool for surface of revolution. It has a standard accuracy of 0.001 in. Rigid or new CNC lathes can achieve an accuracy of 0.0005 in. Tighter tolerances are achieved more easily on the lathe than on the milling machine. But slender parts, interrupted cuts, hard metals, and other practical conditions do influence this tolerance. A minimum tolerance of 0.005 in. is frequently acceptable. Lathe surface roughness capability is superior to the milling machine, ranging from 250 to 16 μin. Concentricity tolerances greater than 0.005 in. are practical.

Grinders, as a family, may be rotational, planer, internal, or external. These machines are able to give tighter tolerances than either the milling machine or the lathe. Generally, accuracies of 0.0005in. are achievable, although 0.001 in. is preferred. A surface roughness range of 63 to 4 μin. is possible. Generally, grinders are a more expensive way to remove metal as compared to turning or milling.

New Words and Expressions

1. geometric dimensioning and tolerancing 几何精度设计
2. bilateral a. 双向的 bilateral tolerance 双向公差
3. unilateral a. 单向的 unilateral tolerance 单向公差
4. interference fit 过盈配合
5. counterpart n. 副本，极相似的人或物，配对物
6. mandatory [ˊmændətəri] l a. 命令的，强制的，必须遵循的；n. 代理人
7. glue [glu:] n. 胶，胶水，vt. 胶合，粘贴，粘合
8. harmonize[ˊha:mənaiz] v. 协调，谐和，调和，谐调，vt. 使...调和，使...一致
9. cutoff n. 1）切去（开，断，边），2）断开 3）截止（点）a.界限的，分界的
10. allowance [əˊlauəns] n. 1）允许（量），许可（限度），（配合）容差，（配合）公差，（加工）余量，间隙，紧度；2）津贴，宽容，允许；vt.定量供应
11. clarity [kˊlæriti] n. 清楚，透明，明了
12. MMC 最大实体条件
13. LMC 最小实体条件
14. RFS 独立原则
15. mutually [ˋmju:tʃuəli] ad. 互相地，互助
16. mandrel [ˊmændrəl] n.（圆形）心轴
17. the rule of thumb 经验法则（方法），凭感觉的方法，计算中的近似法
18. slender [ˊslendə] a. 1）细长的，狭的，窄的；2）薄弱的，单薄的
19. title block 工程图明细表
20. deflection [diflekʃən] n. 偏斜，偏转，偏差，挠曲

Exercises

1. Give an explanation of the following terms:

Feature	Datum	Straightness
Dimension	Concentricity	flatness

2. Discuss the advantage of GD&T symbols over drawing notes.
3. List the types of symbols for form.
4. List orientation symbols.
5. What is the difference between profile of a surface and profile of a line?