This is a digital copy of a book that was preserved for generations on library shelves before it was carefully scanned by Google as part of a project to make the world’s books discoverable online.

It has survived long enough for the copyright to expire and the book to enter the public domain. A public domain book is one that was never subject to copyright or whose legal copyright term has expired. Whether a book is in the public domain may vary country to country. Public domain books are our gateways to the past, representing a wealth of history, culture and knowledge that’s often difficult to discover.

Marks, notations and other marginalia present in the original volume will appear in this file - a reminder of this book’s long journey from the publisher to a library and finally to you.

Usage guidelines

Google is proud to partner with libraries to digitize public domain materials and make them widely accessible. Public domain books belong to the public and we are merely their custodians. Nevertheless, this work is expensive, so in order to keep providing this resource, we have taken steps to prevent abuse by commercial parties, including placing technical restrictions on automated querying.

We also ask that you:

+ **Make non-commercial use of the files** We designed Google Book Search for use by individuals, and we request that you use these files for personal, non-commercial purposes.

+ **Refrain from automated querying** Do not send automated queries of any sort to Google’s system: If you are conducting research on machine translation, optical character recognition or other areas where access to a large amount of text is helpful, please contact us. We encourage the use of public domain materials for these purposes and may be able to help.

+ **Maintain attribution** The Google “watermark” you see on each file is essential for informing people about this project and helping them find additional materials through Google Book Search. Please do not remove it.

+ **Keep it legal** Whatever your use, remember that you are responsible for ensuring that what you are doing is legal. Do not assume that just because we believe a book is in the public domain for users in the United States, that the work is also in the public domain for users in other countries. Whether a book is still in copyright varies from country to country, and we can’t offer guidance on whether any specific use of any specific book is allowed. Please do not assume that a book’s appearance in Google Book Search means it can be used in any manner anywhere in the world. Copyright infringement liability can be quite severe.

About Google Book Search

Google’s mission is to organize the world’s information and to make it universally accessible and useful. Google Book Search helps readers discover the world’s books while helping authors and publishers reach new audiences. You can search through the full text of this book on the web at [http://books.google.com/](http://books.google.com/)
MACHINE TOOL OPERATION
PREFACE

Given a class of twenty boys in day school, or twenty men in night school, a subject full of interest, and a teacher full of enthusiasm, can the teacher give information regarding the subject in a "talk" or "lecture," so that the majority of the members of the class will retain it twenty-four hours without some definite reference material?

Certain members of the class may be absent; how are they to get this information except by individual instruction unless a suitable text is provided?

If brief notes concerning the construction of the machine and the various elementary operations are available will they not serve to shorten the time taken for talks and thus give more time to practical application of principles?

Will not a student have more confidence and hence gain skill more quickly if he has, at hand, the necessary information (and certain information is necessary) while performing the operation or studying the mechanism?

The following text is the outgrowth of notes prepared by the author in an effort to answer intelligently the above and many similar questions.

The presentation of this material in book form is the result of requests from many teachers and students, that it be given in a more useful and permanent form than that of mimeographed notes. How best to arrange such material is a most difficult problem and any arrangement is open to criticism. This text is planned to permit all possible flexibility in its use. It is not a course of study arranged in any particular pedagogical sequence; the advisability of such a course is doubtful except for the one shop and then only for a given time. Certain chapters comprise a study of operations; other chapters may be used for reference and studied in connection with
these operations. The particular job on which the operation is to be made and the proper time to refer to the construction or use of the machine or tool must be decided according to conditions which obtain at a given time in a given shop.

The purpose of this text is to assist those who desire to get a knowledge of the principles and elementary operations of machine shop work. It is in no sense a treatise, nor is it a production manual. It is primarily designed to be used in connection with class talks or demonstrations in the school shop, although it may be used to supplement the information which a boy in the commercial shop may acquire by observation, practice, or other means.

The author's aim has been to adapt this text for use in Vocational, Industrial, Technical and Trade Schools, and in Apprenticeship courses where training in machine shop practice is given. To this end he has selected what he regards as the necessary elementary information concerning machine tool operation, and has endeavored to set it forth as simply and clearly as possible.

The chief function of the school shop is to teach boys to operate machines intelligently. It must be conceded that a knowledge of the construction of the machine tools and of the principles underlying their operation makes the essential difference between a machine operator and a machinist. Therefore, the mechanisms of typical standard machine tools have been described more or less in detail. Such descriptions should increase the interest that the boy will have in his shop work.

Many of the questions which are incorporated in various chapters of this book were originally printed in the Manual for Machinists, which was prepared by the author of this text for the War Department, Committee on Education and Special Training.

The author is sincerely grateful to the many manufacturers who have assisted him in the preparation of this book. Further, as a teacher, he desires at this time to acknowledge the help which the manufacturers render to teachers and students.
by generously sending catalogues, pamphlets and instruction books for school use. In conclusion, he wishes to express his appreciation and thanks to Mr. Frank E. Mathewson, Director, Technical and Industrial Department, Wm. L. Dickinson High School, for constant encouragement, and to his fellow teachers, Mr. Carlos H. Handforth, Mr. Henry Ouram, Mr. Paul F. Weld, and Mr. George C. Witt for honest criticism and valuable suggestions.

H. D. Burghardt.

Jersey City, N. J.,

September, 1919.
TO THE STUDENT

This is written for the young man in the shop who hopes to be a machinist. It is written for the ambitious young man (and most young men are ambitious), who expects to be, some day, a first-class machinist, a foreman, a superintendent. It is written for the alert young man, he who combines with his ambition a determination to work, to study, to think; a determination to make enthusiasm in his work overcome laziness, to make preparedness bring the opportunity. This shop you are in is one of a hundred, or of a thousand, or of ten thousand, where certain boys are being trained. Some boys are lazy and shiftless; they will be drudges all their lives. Others are the future foremen, superintendents, managers, owners. Are you “on the job”? Who is happier, the wide-awake fellow with his feeling of satisfaction in accomplishment, or the lazy fellow with his feeling of envy? Both have to work! What is work? Webster says, “Work is a physical or intellectual effort directed to some end.” Either one, physical effort or intellectual effort taken alone is drudgery. Properly combined they produce enthusiasm. Football is work, it is sport too; but beef alone or brains alone never made a football player. Machine shop practice is work. Often and especially at the beginning, it may be hard work and dirty work, but combined with the proper amount of “intellectual effort” it is interesting and increasingly interesting as one progresses.

What is meant by this training—this doing and this thinking—in machine shop work? Where does it begin and where does it end? It means a development of the hands, ears, eyes and mind in the power to do, to listen, to observe, to remember and to reason.
Machine shop practice consists of certain mechanical principles that are a part of all machine shop work everywhere—the principles of cutting tools, cutting speeds and feeds, actions of gears, screws, cams, etc.—applied in the construction of certain machines and tools and in the various machine operations: that is, in the methods of holding and doing the work. There are only a few principles, comparatively, but there is no end of methods. Machine shop training begins with the elementary principles, the easiest mathematical problems, and the simplest methods of applying these principles and problems on some kind of machine. It advances step by step, to other principles and to the application of all these principles in the doing of work, by various methods, on the other machines.

This text has been prepared with the single purpose of helping the boys in machine shops to gain quickly a working knowledge of the principles of machine work and their application in shop practice. After all, no text, no teacher, no foreman or no friend can help the boy who is not willing to help himself.

The Author.
CONTENTS

Preface .................................................. v
Word to the Student ................................. ix

THE LATHE

CHAPTER I

The Machinist’s Trade

1. What is a Machine Shop? ......................... 4
2. What may Constitute the Equipment of a Machine Shop? .. 5
4. What is Meant by “Bench Work” and “Floor Work?” .. 14
5. Are There Many Specified Divisions in the Machinist’s Trade? 14
6. How are the Employees of a Commercial Manufacturing Shop Classified? ..................................... 14
7. What is the Knowledge One Must Have to be an Expert Machinist? ........................................... 17
8. What Chance has a Machinist for Promotion? ........ 18
9. What are the Essential Characteristics of a Machinist? 18
10 How are These Characteristics Acquired? .................. 19

CHAPTER II

Lathe Construction and Manipulation

11. The Engine Lathe ...................................... 22
12. Running a Lathe ........................................ 24
13. Parts of the Lathe ....................................... 25
14. Cleaning and Oiling .................................... 25
15. A Few Suggestions ..................................... 28
16. The Bed .................................................. 29
17. The Headstock .......................................... 29
18. The Tailstock ........................................... 30
19. The Carriage ............................................. 31
20. The Feeding Mechanism ............................... 31
21. The Thread Cutting Mechanism ..................... 32
## CONTENTS

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>22. A Very Important Precaution</td>
</tr>
<tr>
<td>23. How Different Speeds are Obtained</td>
</tr>
<tr>
<td>24. Countershaft</td>
</tr>
<tr>
<td>25. Driving and Driven Pulleys</td>
</tr>
<tr>
<td>26. Driving and Driven (Follower) Gears</td>
</tr>
<tr>
<td>27. Simple Gear Train</td>
</tr>
<tr>
<td>28. Compound Gear Train</td>
</tr>
<tr>
<td>29. Direct Spindle Speeds</td>
</tr>
<tr>
<td>30. Lock Pin</td>
</tr>
<tr>
<td>31. Back Gears</td>
</tr>
<tr>
<td>32. Indirect Spindle Speeds, Back Gears</td>
</tr>
<tr>
<td>33. Revolutions per Minute (r.p.m.)</td>
</tr>
<tr>
<td>34. Changing Belts</td>
</tr>
<tr>
<td>35. The Tumbler Gear Train and the Change Gear Train</td>
</tr>
<tr>
<td>36. The Tumbler Gears</td>
</tr>
<tr>
<td>37. The Intermediate Gear in the Change Gear Train</td>
</tr>
<tr>
<td>38. Quick Change Gears</td>
</tr>
<tr>
<td>39. Apron Mechanism (Flather Lathe)</td>
</tr>
<tr>
<td>40. Apron Mechanism (American Lathe)</td>
</tr>
<tr>
<td>41. Features of High-duty Lathes</td>
</tr>
<tr>
<td>42. Double Back Gears or Reducing Gears</td>
</tr>
<tr>
<td>43. The Geared Head</td>
</tr>
<tr>
<td>44. Hendey Lathe Geared Head</td>
</tr>
<tr>
<td>45. American Lathe Geared Head</td>
</tr>
</tbody>
</table>

## CHAPTER III

### CUTTING TOOLS AND CUTTING SPEEDS

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>46. Cutting Tool Efficiency</td>
</tr>
<tr>
<td>47. Cutting Tools Used in Lathe Work</td>
</tr>
<tr>
<td>48. Cutting Angle</td>
</tr>
<tr>
<td>49. Clearance Angles</td>
</tr>
<tr>
<td>50. Rake Angles</td>
</tr>
<tr>
<td>51. Patent Tool Holders</td>
</tr>
<tr>
<td>52. Keep Cutting Tools Sharp</td>
</tr>
<tr>
<td>53. Judgment in Tool Grinding</td>
</tr>
<tr>
<td>54. Grinding Cutting Tools</td>
</tr>
<tr>
<td>55. Definitions of Speed, Feed and Chip</td>
</tr>
<tr>
<td>56. The Time Element</td>
</tr>
<tr>
<td>57. Cutting Feeds and Speeds</td>
</tr>
<tr>
<td>58. Cutting Speed Calculations</td>
</tr>
</tbody>
</table>
# CONTENTS

## CHAPTER IV

**The Scale, Caliper, Snap Gauge and Micrometer**

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>59. Scales</td>
<td>74</td>
</tr>
<tr>
<td>60. Calipers</td>
<td>75</td>
</tr>
<tr>
<td>61. Measuring with a Caliper</td>
<td>76</td>
</tr>
<tr>
<td>62. Setting and Reading an Outside Caliper</td>
<td>77</td>
</tr>
<tr>
<td>63. The Micrometer Caliper</td>
<td>78</td>
</tr>
<tr>
<td>64. Holding a Micrometer</td>
<td>80</td>
</tr>
</tbody>
</table>

## CHAPTER V

**Centering**

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>65. Holding Work in the Lathe</td>
<td>82</td>
</tr>
<tr>
<td>66. Importance of Carefully Locating the Center</td>
<td>83</td>
</tr>
<tr>
<td>67. The Centering Machine</td>
<td>84</td>
</tr>
<tr>
<td>68. Methods of Locating Centers</td>
<td>84</td>
</tr>
<tr>
<td>69. Use of Center Punch</td>
<td>86</td>
</tr>
<tr>
<td>70. Drilling and Reaming the Center</td>
<td>87</td>
</tr>
<tr>
<td>71. Size of Centers</td>
<td>88</td>
</tr>
<tr>
<td>72. Removing a Broken Center Drill</td>
<td>89</td>
</tr>
</tbody>
</table>

## CHAPTER VI

**Facing**

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>73. Adjusting the Work on Centers</td>
<td>91</td>
</tr>
<tr>
<td>74. The Facing Operation</td>
<td>92</td>
</tr>
</tbody>
</table>

## CHAPTER VII

**Turning in a Lathe**

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>75. Position of the Dead Center</td>
<td>98</td>
</tr>
<tr>
<td>76. Accuracy of the Live Center</td>
<td>99</td>
</tr>
<tr>
<td>77. Cleaning and Truing the Lathe Centers</td>
<td>100</td>
</tr>
<tr>
<td>78. Setting the Tool</td>
<td>101</td>
</tr>
<tr>
<td>79. Direction of the Feed</td>
<td>103</td>
</tr>
<tr>
<td>80. Oiling the Centers</td>
<td>104</td>
</tr>
<tr>
<td>81. Lubricating the Tool</td>
<td>104</td>
</tr>
<tr>
<td>82. Graduations on Cross Feed Screw</td>
<td>104</td>
</tr>
<tr>
<td>83. Lost Motion in the Cross Feed</td>
<td>105</td>
</tr>
<tr>
<td>84. Alignment of Centers</td>
<td>105</td>
</tr>
<tr>
<td>85. Quicker Methods of Aligning Centers</td>
<td>106</td>
</tr>
<tr>
<td>86. Setting the Speed</td>
<td>107</td>
</tr>
<tr>
<td>87. The Roughing Cut</td>
<td>107</td>
</tr>
</tbody>
</table>
CHAPTER VIII

CHUCKING WORK

104. Kinds of Chucks .......................... 125
105. Mounting the Chuck on the Spindle .......................... 127
106. Adjusting the Work in a Chuck .......................... 127
107. Radial Facing .......................... 128
108. The Cutting-off Tool .......................... 128
109. The Cutting-off Operation .......................... 129
110. Chattering .......................... 130
111. The Flat Drill .......................... 131
112. The Twist Drill .......................... 131
113. Sharpening a Drill .......................... 132
114. Speeds and Feeds of Twist Drills—Cutting Compounds .......................... 13
115. The Reamer .......................... 13
116. Drilling in a Lathe—Spotting the Center .......................... 14
117. Operation of Drilling the Hole .......................... 14
118. Machine Reaming .......................... 14
119. Hand Reaming .......................... 14
120. The Boring Tool .......................... 14
121. The Boring Tool Holder .......................... 14
122. Reasons for Boring .......................... 14
123. The Operation of Boring a Hole .......................... 14
124. Measuring a Hole .......................... 14
## CONTENTS

### CHAPTER IX

**Tapers and Angles**

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>125. Tapers</td>
<td>152</td>
</tr>
<tr>
<td>126. Off-setting the Tail Stock Slide</td>
<td>154</td>
</tr>
<tr>
<td>127. Rule for Off-set when Turning Taper</td>
<td>155</td>
</tr>
<tr>
<td>128. Method of Gauging Off-set</td>
<td>155</td>
</tr>
<tr>
<td>129. Setting of Turning Tool</td>
<td>156</td>
</tr>
<tr>
<td>130. Methods of Measuring Tapers</td>
<td>156</td>
</tr>
<tr>
<td>131. Fitting a Taper to a Gauge</td>
<td>157</td>
</tr>
<tr>
<td>132. Gauging the Size of a Taper</td>
<td>158</td>
</tr>
<tr>
<td>133. Duplicating a Taper Piece</td>
<td>158</td>
</tr>
<tr>
<td>134. Turning a Taper with a Square Nose Tool</td>
<td>158</td>
</tr>
<tr>
<td>135. Filing a Taper</td>
<td>158</td>
</tr>
<tr>
<td>136. The Taper Attachment</td>
<td>158</td>
</tr>
<tr>
<td>137. Boring Tapers with Taper Attachment</td>
<td>160</td>
</tr>
<tr>
<td>138. Boring Tapers with Compound Rest</td>
<td>160</td>
</tr>
<tr>
<td>139. Fitting Taper Holes</td>
<td>161</td>
</tr>
<tr>
<td>140. Angles</td>
<td>163</td>
</tr>
<tr>
<td>141. Classification of Angles</td>
<td>164</td>
</tr>
<tr>
<td>142. The Use of the Compound Rest for Turning Angles</td>
<td>165</td>
</tr>
<tr>
<td>143. Setting the Compound Rest</td>
<td>165</td>
</tr>
<tr>
<td>144. The Bevel Protractor</td>
<td>167</td>
</tr>
</tbody>
</table>

### CHAPTER X

**Threads and Thread Cutting**

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>145. Threads</td>
<td>170</td>
</tr>
<tr>
<td>146. Definitions of Thread Parts</td>
<td>171</td>
</tr>
<tr>
<td>147. Thread Standards</td>
<td>172</td>
</tr>
<tr>
<td>148. The Dimensions of the U. S. Standard Thread</td>
<td>173</td>
</tr>
<tr>
<td>149. Formula for Finding Exact Root Diameter of U. S. S’t’d. Thread</td>
<td>176</td>
</tr>
<tr>
<td>150. Taps</td>
<td>177</td>
</tr>
<tr>
<td>151. Tap Sets</td>
<td>178</td>
</tr>
<tr>
<td>152. Relief of Taps</td>
<td>178</td>
</tr>
<tr>
<td>153. Tap Size Drills</td>
<td>178</td>
</tr>
<tr>
<td>154. Length of Tapped Hole</td>
<td>179</td>
</tr>
<tr>
<td>155. The Operation of Tapping</td>
<td>179</td>
</tr>
<tr>
<td>156. The Threading Die</td>
<td>182</td>
</tr>
<tr>
<td>157. Definitions and Explanations of Terms Used</td>
<td>183</td>
</tr>
<tr>
<td>158. Operation of the Gears</td>
<td>186</td>
</tr>
<tr>
<td>159. Calculating the Sizes of Gears to Cut a Given Thread</td>
<td>187</td>
</tr>
<tr>
<td>160. Compound Gearing</td>
<td>188</td>
</tr>
<tr>
<td>161. Operation of Cutting the Thread</td>
<td>190</td>
</tr>
<tr>
<td>162. Preliminary Hints on Thread Cutting</td>
<td>191</td>
</tr>
</tbody>
</table>
CONTENTS

163. The Thread Stop ........................................... 193
164. Three-wire Method of Measuring Threads .................. 194
165. The Use of the Compound Rest for Cutting Threads ....... 195
166. Four Ways of Catching the Thread after Resetting the Tool . 197
167. To Cut a Left Hand Thread .................................. 197
168. To Cut a Thread on a Taper ................................ 197
169. To Cut an Inside Thread ................................. 198
170. Cutting a Thread Without the Reverse Belt ............... 198
171. Cutting a Square Thread ................................... 199
172. Cutting an Acme Thread .................................... 202
173. Cutting Metric Screw Threads ............................ 202
174. Multiple Threads ........................................ 204
175. Cutting a Multiple Thread ................................... 205

CHAPTER XI

FACE PLATE WORK

176. Definitions of Accessories Used ......................... 208
177. Typical Face Plate Set-ups ................................ 210
178. Hints on Face Plate Work .................................. 212

BENCH WORK AND WORK AT THE FORGE

CHAPTER XII

HAMMERS, SCREW-DRIVERS, WRENCHES, HACK SAWS

179. The Use of Hand Tools .................................... 217
180. Hammers ................................................. 218
181. Screw-drivers ............................................ 219
182. Wrenches ................................................ 221
183. A Few Suggestions Regarding the Use of Wrenches .... 221
184. Action of Check Nut ...................................... 224
185. The Hack Saw ............................................ 225
186. Proper Number of Teeth .................................. 225
187. Hack-saw Frames ......................................... 226
188. Hints on Hack Sawing ...................................... 227

CHAPTER XIII

LAYING-OUT

189. Laying-out .............................................. 229
190. Tools Used for Laying-out Work .......................... 229
191. Scribing the Lines ....................................... 231
192. The Operation of Laying-out .............................. 232
CONTENTS

CHAPTER XIV

CHIPPING, FILING, SCRAPING

193. Chipping ........................................... 235
194. Cold Chisels ....................................... 235
195. Grinding a Cold Chisel .......................... 236
196. The Operation of Chipping ....................... 237
197. Hints on Chipping .................................. 238
198. Use of Files ........................................ 240
199. Machine Shop Files ............................... 240
200. The Safe Edge ...................................... 244
201. Convexity of Files ............................... 244
202. Taper of Files ..................................... 244
203. File Handles ....................................... 245
204. Care of Files—Pinning ......................... 245
205. Cross Filing ....................................... 246
206. Operation of Filing .............................. 248
207. Draw-filing ....................................... 250
208. Filing Soft Metals ............................... 250
209. Needle Handle Files ............................ 252
210. Don't's in Filing .................................. 252
211. Reasons for Scraping ......................... 254
212. Tools Used for Scraping a Flat Surface .... 255
213. Sharpening the Flat Scraper .................. 256
214. The Scraping Operation ....................... 257
215. Hints on Scraping ............................... 258
216. Scraping Curved Surfaces ...................... 259

CHAPTER XV

WORK AT THE FORGE

217. Soldering .......................................... 261
218. The Principle of Soldering ........................ 261
219. Cleaning the Surfaces. Fluxes .................. 262
220. Soldering Coppers ............................... 263
221. Tinning a Copper ............................... 264
222. Dipping Solution .................................. 264
223. Soldering Operation ............................. 264
224. Brazing ........................................... 265
225. Babbitt ............................................ 266
226. Babbitting a Bearing ............................ 267
227. Steel ................................................ 269
228. Machine Steel and Carbon Tool Steel ....... 270
229. Hardening ........................................... 271
<table>
<thead>
<tr>
<th>CONTENTS</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>230. Hints on Hardening</td>
<td>271</td>
</tr>
<tr>
<td>231. Tempering Experiment—Temper Colors</td>
<td>273</td>
</tr>
<tr>
<td>232. Hints on Tempering</td>
<td>274</td>
</tr>
<tr>
<td>233. High Speed Steel</td>
<td>275</td>
</tr>
<tr>
<td><strong>APPENDIX</strong></td>
<td></td>
</tr>
<tr>
<td>Rules for Speeds of Pulleys</td>
<td>277</td>
</tr>
<tr>
<td>Rules for Gear Velocities</td>
<td>279</td>
</tr>
<tr>
<td>Machine Fits</td>
<td>280</td>
</tr>
<tr>
<td>Fastening Belts</td>
<td>281</td>
</tr>
<tr>
<td>Geometrical Progression</td>
<td>283</td>
</tr>
<tr>
<td>Principle of the Vernier</td>
<td>284</td>
</tr>
<tr>
<td>Reading the Vernier Caliper, etc.</td>
<td>286</td>
</tr>
<tr>
<td>Bevel Protractor with Vernier</td>
<td>287</td>
</tr>
<tr>
<td>Micrometer with Vernier</td>
<td>288</td>
</tr>
<tr>
<td>Screw Thread Micrometer</td>
<td>289</td>
</tr>
<tr>
<td>List of Tables</td>
<td>291</td>
</tr>
<tr>
<td>Tables 1 to 23</td>
<td>292-317</td>
</tr>
<tr>
<td><strong>INDEX</strong></td>
<td>319</td>
</tr>
</tbody>
</table>
MACHINE TOOL OPERATION

THE LATHE

CHAPTER I

THE MACHINIST'S TRADE

This has been called the mechanical age. Machinery is everywhere. Practically all of the necessities and the luxuries of life are made by machinery.

In thousands of factories food-stuffs are refined and prepared, fabrics are woven, and wood and metal are fitted to make the furnishings of civilization. Everywhere factories are building labor-saving devices for the homes and workshops and building other devices for the education, recreation and prosperity of the people. But, without the machinist there could be no engines or dynamos to furnish the power; there could be no machines because the machinist is the producer of them all. As a matter of fact there would be no factories.

Lincoln had no telephone in the White House, or no electric light. Think of the development of the telephone and the machines that have made it possible. Think of the development of the electric light and of the huge engines and dynamos in the electric power stations. Think of the growth of the automobile industry, of the improvements in the product year by year, and the improvements in the methods of manufacture. Consider the motor alone—the absolute reliability of the materials used, the necessary perfection of fit of its component parts, the marvelous methods of manufacture to make these parts by the thousands. These examples could
be multiplied indefinitely—sewing machines, cameras, phonographs, moving picture machines, typewriters, cash registers—are a few that are universally known. Everybody knows too that all of these things are made by machinery. Who invents the machines? Who develops the machines? Who builds the machines for making all these things? The loafer? No. The worker and thinker? Yes.

A few years ago there were two kinds of steel, "machine steel" and "tool steel." Today there are dozens of special steels—steels for gears, for shafts, for screws, for springs, for tools, even a kind of steel for the particular gear or axle or tool. The new methods of heat treatment of steel have added strength, toughness and temper far beyond the dreams of the past generation.

These typical industrial developments together with thousands of others have been made possible by the co-operation of trained men—machinists, designers, electricians, chemists and metallurgists working in harmony with progressive business men.

Progress means the development of improvements and the machine is the instrument of progress.

The machinist's trade is a great trade—great in its vital necessity, great in its ever-increasing interest, and great in the opportunity it offers for advancement. John Fritz, who founded the Bethlehem Steel Co., was a machinist in his youth; George Westinghouse started his life work in a machine shop. Dr. John A. Brashear, one of the greatest telescopic instrument makers and one of the greatest scientists of today, was once a machinist. Henry Ford's knowledge of machine shop work has served to make happy tens of millions of people.

The operator of a special machine for doing a certain class of work, whether in a machine shop or in a factory, is not a machinist; he is only a "machine operator." Such work usually calls for very little knowledge and less judgment. It is deadly monotonous and offers no particular chance for advancement.
It is the young man who is not satisfied with being a machine operator or a "machine hand" who is determined to be something more than "cheap help;" it is the wide-awake, thinking young fellow who becomes the expert journeyman mechanic, then the foreman, and finally the superintendent. It is not all fun; it means work; it means study; it may mean sacrifice of money at the start, but it is interesting, vitally stimulating and not deadening—walking a tread mill is only existing; going ahead is living.

In the large industrial centers, more than half the workers are employed in the metal trades; that is, more mechanics and machine hands are employed in these trades than are at work in all the other trades taken together. It is safe to say that 90% of the foremen in the great manufacturing plants have been promoted from the ranks of machinists, and that 90% of the superintendents of these factories have previously been foremen.

It is admitted by all manufacturers, that the proportion of expert machinists is growing constantly smaller; why then, should it be difficult for the young man in the shop to foresee the opportunities that may be his, if he is prepared.

During the last few years factory conditions have greatly changed; scientific management, motion study, etc., have made the machine hand more than ever before a mere part of the machine, while the standing of the real machinist is on an increasingly higher level. The same operations—turning, boring, drilling, etc., are performed, but are made easier, more quickly, and with greater accuracy. The cutting tools have more than double the efficiency of those of a few years ago. The machines have been built stronger, more rigid, more adaptable, more accurate. While these improvements have lessened the manual labor incident to machine work, the truly marvelous strides made in the manufacturing of thousands of parts exactly alike has advanced the making of the special machines, tools and gauges—that is, real machine shop work—to a much higher plane as regards the mental effort necessary to do this work.
Conditions that obtain in the machine shops, obtain also, in more or less degree, in the other metal trades—in the steel mills, in the forge shops, and in the foundries. Doubtless these conditions may be found in any trade or in any business—the uninteresting, monotonous work—the drudgery, is done by cheap help, unskilled or semi-skilled "hands." The trained man has the interesting work, the most pay, and best of all, the chance for promotion.

Fig. 1.—Typical Machine Shop View (Dusenberg Motor Co., Elizabeth, N. J.).

A SHORT MACHINE SHOP CATECHISM

1. What Is a Machine Shop?—A machine shop is a place in which metal parts are cut to the size required and put together to form mechanical units or machines, the machines so made to be used directly or indirectly in the production of the necessities and luxuries of civilization. Machine shop work is the basis of all mechanical production.
2. What May Constitute the Equipment of a Machine Shop?—Machine shop equipment consists in part of certain standard machine tools, the kind, the size and the number of machines depending, of course, upon the product of the shop. Machine shop equipment includes, in addition, the tools used at the bench and on the floor as well as the measuring and adjusting tools, the work-holding and tool-holding accessories, and the small tools used in the machines.

3. What Are the Standard Machine Tools?—The lathe, the drill press, the shaper, the planer, the milling machine, the grinding machine and the boring mill are usually considered standard machine tools. The turret lathe, the slotter, the gear cutting machine and many others are usually referred to as manufacturing machines or special machines. The propriety of the term special machine lies in the fact that these are modifications of standard machine tools and have been developed to meet specialized production problems.

It is difficult, and perhaps unnecessary, to draw the line between standard and special. A few years ago the milling machine, and still more recently the grinding machine, were considered special machines. Now they are regarded as very essential machines in the well equipped shop. One who is able to operate intelligently the machines recognized as standard will be able, with a minimum amount of study and experience, to understand and operate any special machine.

It must be understood that all kinds of machine tools are made in a great variety of types and sizes. Fortunately for the machinist, certain basic principles of construction and operation which obtain in one size of machine, obtain in all machines of that class, and further, many of these same principles are found in other kinds of machines. The student or apprentice does not have to begin all over again to learn to operate each size and kind of machine. If he has mastered certain principles regarding one size or type of any kind of standard machine tool he has certain knowledge that will help considerably in understanding the construction and operation of other machines. The similarity of construction of the
larger and smaller sizes of machines is illustrated in Figs. 2 and 3 and again in Figs. 4 and 5. Fundamentally the two lathes are alike in construction and operation; likewise the two drilling machines. The underlying principles of the proper cutting speeds and feeds, of the grinding of the cutting tools, of the adjustments of bearings, etc., etc., apply to all of them.

The Lathe.—The lathe (Figs. 2 and 3) is a metal-turning machine tool in which the work, while revolving on a hori-

![Fig. 2.—Turning in a 36 inch by 24 foot lathe (Courtesy American Tool Works Co.).](image)

zontal axis, is acted upon by a cutting tool which is made to move slowly (feed) in a direction more or less parallel to the axis of the work, (longitudinal feed) or in a direction at right angles to the axis of the work (cross feed). Either feed may be operated by hand or by power (automatically) as desired. When the feeding is in a direction parallel to the axis of the work, cylindrical or “straight turning” is accomplished. When the cut is in a direction at a slight angle to the axis of the work a “taper” is the result; more of an angle results in “turning to an angle.” The cut at right angles to the axis
of the work (the cross feed operation) is known as "facing" or "squaring." Cutting inside of a hole is termed "boring."

![Image](image-url)

**Fig. 3.—Turning in a 14" by 6' lathe. Electrical Laboratory Wm. L. Dickinson High School.**

**The Drilling Machine.**—The drilling machine or “drill press” (Figs. 4 and 5) is a machine tool used for producing
holes in metal. In this machine the work is securely held while a revolving cutting tool is fed into it. The cutting tool most commonly used is called a "drill;" it has, in effect, an action similar to a wood "bit."

![Image](image_url)

**Fig. 4.—Six spindle Heavy Duty Drilling Machine (Courtesy Niles-Bement-Pond Co.).**

*The Shaper.*—The shaper (Fig. 6) is a machine tool used for finishing flat surfaces of metal pieces of the smaller sizes, that is, pieces not usually over a foot or two long. In the shaper, the cutting tool has a reciprocating (forward and return) motion, and cuts on the forward stroke only. The work is usually held in a vise bolted to the work table and the regular feed is accomplished by causing the work table to move automatically at right angles to the direction of the cutting tool. The construction of the tool head permits of down feed at right angles to the regular feed, or at any other angle if desired. The cutting tools used in the shaper are similar to the turning tools used in the lathe.

*The Planer.*—The planer (Fig. 7) is a machine-tool used in the production of flat surfaces on pieces too large or too heavy or perhaps too awkward to hold in a shaper. In this machine the table or "platen" on which the work is securely fastened
has a reciprocating (forward and return) motion. The tool head may be automatically fed horizontally in either direction along the heavily supported cross rail over the work, and

automatic down feed is also provided. Cutting tools used in planer work are the same as those used in the shaper.

*The Milling Machine.*—The milling machine (Fig. 8) is a machine tool in which metal is removed by means of a revolv-
Fig. 6.—Shapers (Gould and Eberhardt, and American Tool Works Co.) in Wm. L. Dickinson High School.

Fig. 7.—Two Head Planer (Courtesy Whitcomb-Blaisdell Tool Co.).
Fig. 8.—Milling special taper reamers in Universal Milling Machine (Brown and Sharpe Mfg. Co.) in Wm. L. Dickinson High School.

Fig. 9.—Grinding spindles in a Plain Grinding Machine. (Courtesy Norton Grinding Co.).
ing cutter with many "teeth," each tooth having a cutting edge which removes its share of the stock. The work is supported by various methods on the work table, and may be

![Vertical boring mill](image)

**Fig. 10.—**Vertical boring mill (*King Machine Co.*) in Wm. L. Dickinson High School.

fed to the cutter either longitudinally, transversely, or vertically. A great variety of work may be done on a milling machine, and next to the lathe it is perhaps the most adaptable and interesting machine in the shop.

*The Grinding Machine.*—The grinding machine (Fig. 9)
is a machine tool, in which an abrasive wheel is used as a cutting tool to obtain an accurate and beautiful finish on metal parts, including soft and hardened steel. A large variety of types, and a number of sizes of surface grinding machines and external and internal cylindrical grinding machines are manufactured for ordinary and special grinding operations.

![Horizontal boring mill](Universal Horizontal Boring Machine Co.) in Wm. L. Dickinson High School.

**Fig. 11.**—Horizontal boring mill (Universal Horizontal Boring Machine Co.) in Wm. L. Dickinson High School.

**The Boring Mill.**—There are two distinct types of boring mill design, the vertical (Fig. 10) and the horizontal (Fig. 11), both of which are modifications of the lathe. Boring is the operation of enlarging a hole, usually by means of a single cutting tool, and the boring mill is designed primarily for the purpose of finishing holes that are impracticable to finish in a lathe or other machine because of the size or shape of the casting.
In a vertical boring mill, the work table revolves on a vertical axis and the cutting tool (which may be a drill or a boring tool or a turning tool) is arranged above the table and may be fed laterally, (toward or away from the center of the table) and up or down in any position. Because of these feeding arrangements, turning and facing may be accomplished as easily as boring. In the smaller sizes the various tools are arranged in a turret head.

In a horizontal boring mill, the cutting tool revolves on a horizontal axis. The spindle which carries the cutting tool may be fed longitudinally through the spindle head and in the more recent designs the spindle head may be fed vertically. The work table may be fed longitudinally and transversely. The horizontal boring mill while designed primarily for boring holes may also be used for finishing horizontal and vertical flat surfaces by means of a suitable milling cutter fastened to the spindle.

4. What is Meant by "Bench Work" and "Floor Work"?—Bench work in a machine shop consists of laying out, assembling and the final fitting of parts. When the same operations are performed on heavy work, the term floor work applies.

5. Are There Many Specified Divisions in the Machinist's Trade?—There are probably more opportunities for specializing in machine shop work than in all of the other trades taken together. Consider the range of sizes of machines and consequently the work to be done; also the opportunity of specializing on certain types of machines, such as planer, milling machine, or grinding machine. One might prefer model work, experimental work, or tool-making. Tool making itself is divided into several branches such as die making, jig making, gauge making, etc. In any event, the machinist of whatever class, or the tool-maker of whatever specialty, must have a certain machine shop sense and a knowledge of principles and methods. These can be acquired only by experience and study.

6. How are the Employees of a Commercial Manufacturing Shop Classified?—The machine shop is but one of the essential
units in the typical production plant. In the machine shop the special tools and machines are developed and built, and repairs made. Elsewhere in the factory are rooms filled with special machines or "manufacturing" machines run by machine operators. These operators become very skillful in doing one thing, but they are not mechanically trained.

It has not seemed necessary or advisable for manufacturing purposes to train but a small proportion of the employees to be anything but operators, or assemblers, or machine hands, with the result that few, indeed, are what may correctly be called machinists.

The employees of commercial manufacturing shops may be classified according to grades of attainment about as follows:

_**Machine Operator.**—A machine operator is one who merely operates a "manufacturing" machine doing one class of work. He is able to start and stop the machine, fasten in place the piece to be machined, and remove it when the operation is complete. He makes no cutting tool adjustments and is in no sense a mechanic.

_Assembler._—The assembler takes the parts already made and inspected and puts them together. In general, this work calls for some skill and a reasonable amount of common sense, but requires no particular mechanical intelligence except where the final fitting and adjusting is done on the job.

_Machine Hand._—A specialized machine hand is one who has very little general machine-shop knowledge but who has operated a special machine long enough to be skillful in a variety of work on this machine, or on a machine of this class. He is able to do his own set-up work and make the necessary adjustments.

_Machinist's Helper._—A machinist's helper knows the names and uses of the various small tools (cutting tools, measuring tools and gauges, holding tools, etc.) used in machine-shop work. In addition he may be able to do elementary bench work or machine work.

_A Specialized Machinist._—A specialized machinist is one who has had some general machine shop experience and has
made a specialty of some one machine or some one class of work, such as lathe work and planer work. He has a broader background of experience and more versatility than the machine hand.

_Bench and Floor Hands._—Bench hands and floor hands possess information and skill regarding a number of so-called hand operations (as differentiated from machine operations) such as filing, scraping, assembling and adjusting. A skilled bench hand or floor hand, in general machine shop work, has the ability also to read blue prints readily and to do lay-out work. In addition, a first class bench hand or floor hand has had, usually, considerable experience in machine operation.

_The Machinist._—The general machinist has had enough experience, has acquired enough information, has developed enough judgment, and possesses "head" enough to be able to set up intelligently and operate any standard machine tool and perform any bench or floor operation. In addition, he is able to harden and temper machine shop cutting tools.

_The Tool-maker._—The expert tool-maker qualifies substantially as the general machinist. Tool making is usually a lighter or smaller class of work and, generally speaking, involves more delicate workmanship, more accurate measurement than does general machine work. It also involves more mathematical calculations on the part of the workman and a more extended use of the various machine tool attachments.

_Apprentice Machinist._—The grades of apprentice vary naturally from beginning apprentice to advanced apprentice. The beginner may have no previous machine-shop experience, while the advanced apprentice should have a thorough training in the fundamental knowledge of a machinist. The typical apprentice agreement is based upon an understanding that the apprentice shall be given a few months’ experience on bench and floor work and on each of the standard machine tools, and in addition, shall be given an opportunity to learn the essential principles of the operation of each. The degree of attainment of an apprentice at a given time, depends of course upon the individual, other things being equal. The ambitious
apprentice is keenly desirous of learning the trade. Where
the employer fully meets his obligation, the apprentice is
offered every advantage to learn the operations, methods,
calculations and principles involved in machine shop practice,
to the end that his development may be rapid and sure.

7. What is the Knowledge One Must Have to be an Expert
Machinist?—He must have an understanding of certain fixed
principles which obtain in all machine shop practice, for
example:
The action of metal-cutting tools.
Cutting speeds.
Feeds and feeding devices.
Strength of materials—strains and stresses—rigidity and
spring.
Gear trains.
Measurements.
Adjustments, etc., etc.
He must have a sufficient knowledge of arithmetic to read
measurements from the various instruments, and to make the
necessary calculations for cutting speeds, gear velocities,
angles, threads, etc., etc.
He should have a sufficient knowledge of the principles of
mechanical drawing to be able at least to read blue-prints of
machine details.
He should have a reasonable working acquaintance with the
construction and operation of the typical standard machine
tools. To be an expert machinist does not imply a highly
specialized knowledge of all or perhaps of any one of the
machine tools, but it is an established fact that the high class
specialist on a particular kind of machine or class of work is
also to a considerable extent familiar with general machine
shop practice.
He must be resourceful in methods. The most efficient way
of doing a certain job often depends on the accuracy required;
the number of pieces to be made; the available machines;
and the available tools.
He must have a considerable knowledge of the sequence of
operations; the knowledge of how to go at the job to assure accuracy of result in the shortest time. It is said that there are over one hundred operations on the receiver of the Springfield rifle, which, when finished, weighs about a pound. Think of the satisfaction of being able to arrange the sequence of operations and to design the special tools and fixtures for such a job. It requires the knowledge of a machinist. Every job of five operations or of ten operations thoughtfully worked out is a problem solved and every problem solved is a help in solving the next.

8. What Chance has a Machinist for Promotion?—The chief advantage of the machinist's trade is in the opportunities it offers for promotion. Every machine shop foreman, naturally, must have been promoted from the ranks. Further, practically every superintendent and every master-mechanic of any industry manufacturing metal goods of any description, is a machinist. These men may have gone through the drawing room, but they were machinists before they were draftsmen.

Thousands of successful manufacturers were once machinists. They had ideas suggested by their machine shop experience. They put these ideas into practice and developed them. To mention only a few: Joseph R. Brown and Lucien Sharpe, founders of the Brown and Sharpe Manufacturing Co., of Providence, Rhode Island; Francis A. Pratt and Amos Whitney, founders of the Pratt & Whitney Co. of Hartford, Connecticut; Worcester R. Warner and Ambrose Swasey, founders of Warner & Swasey of Cleveland, Ohio, were all apprentice boys, and then machinists, before they were counted among the foremost manufacturers of the world.

9. What Are the Essential Characteristics of a Machinist?—Carefulness, accuracy, speed, judgment, and confidence are five essential characteristics that a skilled machinist must have.

Care of Self.—A machine is a good servant but a cruel teacher. It is a dreadful thing to lose a finger in order to learn that revolving gears are dangerous things to handle. It is better to be over cautious until habits are formed, which, without conscious reasoning on the part of the operator, make
a dangerous move around a machine practically impossible. That is, a well-trained machinist is careful through habit.

Care of Machine.—A mechanic is always careful not only of the appearance but of the good condition of his machine.

Accuracy.—It is very often necessary for a machinist to work within $\frac{1}{1000}$ of an inch. This is easy enough with the machine tools and measuring tools found in modern shop equipment. It means, however, that the machine must be perfectly adjusted and otherwise in first-class condition, that the cutting tool must be properly sharpened and set, and that the measuring tool is dependable for accuracy.

Speed.—An expert mechanic studies the methods and means of doing a job; makes sure that the machine, cutting tools, and measuring tools are in good condition and then with care, and without undue haste, operates the machine to obtain the maximum production. Carefulness, orderliness, thoughtfulness and close attention to the little things make for speed.

Judgment.—A man is successful in any business in about the same proportion as he acquires judgment. This is as true of a machinist, foreman or superintendent, as it is of any other business or professional man.

Judgment is the ability to decide correctly after comparing ideas, methods or facts. The mechanic must cultivate ideas, study methods, and learn facts regarding his trade. He must know when to rough, when to finish, where accuracy is necessary, and when and where it is not essential. He must be resourceful in ideas and methods in order to adapt himself to various shop conditions. Judgment is intelligence, and every job, well thought out and well done, sharpens the intellect and paves the way toward success.

Confidence.—The man who through study, thought, and careful application has confidence in his own ability to accomplish results has in this confidence a factor which makes for success.

10. How Are These Characteristics Acquired?—A skilled artisan is one who has the power to think and execute with knowledge and ability. To think is to employ the mental
capacity of distinguishing ideas and methods. To execute with expert ability means to employ the senses with confidence and accuracy.

The man who aspires to leadership in any trade or profession must study and must work. Theory and practice walk hand in hand toward skill, knowledge and efficiency, and a man's value to himself and to his employer is always in proportion to his efficiency.

Skill in machine work may be acquired by studying how and why certain operations are done and in connection with this study, a considerable experience in performing these or similar operations is essential.

A lifetime of merely doing is not sufficient to acquire knowledge, except in a very limited degree. One must take advantage of what others have done and are doing. A fund of information is available in the special articles in the trade papers, and in the advertisements in these magazines; in the manufacturers' catalogues and instruction bulletins; and in reference books of which dozens have been prepared regarding each of the standard machine tools. Whole volumes have been written also about machine parts such as gears and cams; and about machine shop mechanics and machine shop mathematics; about steel and the heat treatment of steel for various purposes. It is unnecessary to own all of these books but it surely is advisable to know where certain kinds of information can be found when wanted. It is almost as necessary for a machinist to appreciate the value of a reliable handbook as it is for him to know how to use a micrometer. To keep up-to-date it is well worth while to read regularly at least one of the magazines relating to the work of the machinist. The progressive machinist is a student.

Efficiency can be approached only through the application of the best methods. The selection of the best method requires sound reasoning. The power of sound reasoning is founded in the knowledge of principles. One must know why before he can reasonably know how.
This sounds serious; it is serious, but certainly not discouraging. Study is easy and work is fun when one is interested. Master the first principles, and get interested, develop that interest into the right kind of enthusiasm and your knowledge and your power, your good influence and your income, will grow and grow fast.
CHAPTER II

LATHE CONSTRUCTION AND MANIPULATION

11. The Engine Lathe.—The lathe while being the most important machine shop tool is also one of the simplest in its construction. Its simplicity, together with the wide range of operations of which it is capable make it especially interesting to the young mechanic. This, in addition to the fact that so many basic elements of machine construction and machine shop practice are involved in the construction and operation of the lathe, makes lathe work without question the proper elementary machine shop work.

The function of a lathe is the removal of metal, by means of a suitably formed cutting tool of hardened and tempered steel, from a piece of work which is securely supported and made to revolve.

The engine lathe or “lathe” as it is usually called in machine shops is power driven, has automatic feeds, and is provided with a lead screw for cutting threads. These machines are classified as to their size by the maximum diameter of work which may be revolved over the ways, such as 10”, 14”, 16”, 24”, etc. For particular classifications the total length of the bed is also noted. The smaller lathes, 14” and 16” (Fig. 12) are far more numerous than the larger lathes such as are illustrated in Fig. 13. The big lathes are, however, very necessary in manufacturing heavy ordnance, huge pumps, engines, shafts, etc.

The general lathe operations are straight (cylindrical) turning, taper turning, boring (straight or taper), facing (which is a cut at right angles to the axis of the work) and thread cutting. Attachments of particular value are available for special operations, such as the relieving or backing-off attachment,
Fig. 12.—Turning in a 16" lathe in Wm. L. Dickinson High School.

Fig. 13.—Shows two 60" × 200' gun lathes in the Washington Navy Yard. The lathe at the left is "boring;" the lathe at the right is "turning" (Courtesy Niles Bement Pond Co.).
milling attachment, grinding attachment, taper attachment, etc.

Special lathes of a large variety of patterns and sizes are made for different kinds of work, the most notable example of which is the turret lathe, (Fig. 14). The turret lathe is a "manufacturing" machine. Considerable mechanical ability is required to make and adjust the several cutting tools in the turret head and cross slide, but when the tools are once made and adjusted and the stops set, it requires no particular mechanical intelligence to operate the machine. To run an engine lathe on a variety of work is much more interesting and calls for a higher degree of intelligence than the operation of a turret lathe.

12. Running a Lathe.—Almost every young man in the shop—errand boy, apprentice boy, machinist's helper—wants to "run a lathe." It is a worthy ambition, but before he can
hope to do much more than start or stop the machine, he must learn about the cutting tools—their shape, how they are sharpened, and how they are held to peel off the metal. He ought to know how to read the thirty-seconds and sixty-fourths of an inch on a rule or "scale" quickly and accurately, how to "feel" with a caliper to obtain a measurement within .002". He should learn as soon as possible the names and functions of the parts of the machine. He should appreciate, to a reasonable extent the value of the proper cutting speeds and feeds. He should know how to oil the lathe carefully and thoroughly. After he has studied these things "running the lathe" will be more interesting.

It is not to be expected that the beginner will learn all about the construction of a lathe in one or two lessons. This chapter covers information that should be acquired as rapidly as possible but may be acquired in connection with the doing of jobs in the lathe. The right kind of boy will not be satisfied with merely operating the lathe any more than he will be satisfied with sitting down to study names and functions of parts, and theory of why and how. Experience and knowledge go together and make for keener interest and faster progress.

The real mechanic understands the construction of his machine; he knows the names and uses of the parts and the principles underlying the operation of the mechanisms. The more one learns of these things the more interesting the work becomes.

13. Parts of the Lathe.—On the following pages is illustrated a standard engine lathe with the parts numbered and named. Do not be satisfied until you know the name and function of each part.

14. Cleaning and Oiling.—One of the best ways for the beginner to start his acquaintance with a machine is to clean it thoroughly and oil it. Cleaning should be done while the machine is idle; never when it is running. A small piece of waste moistened with kerosene will serve to cut the dirt and grease, after which wipe with dry waste. The ways and other
PARTS OF THE LATHE

A. Bed.
   1. Ways or Vs.
B. Headstock.
   2. Main spindle, made of crucible steel accurately ground to size. The axis of rotation of the main spindle determines the center line of the lathe.
   3. Live center, revolves with spindle, forms central support for work.
   4. Small face plate, for driving work.
   5. Back gear operating handle; pulled toward the operator, engages back gears; pushed away from the operator, disengages the back gears.
   6. Lock pin, must be out when using back gears; must be in when not using back gears.
   7. Two speed gear handle, to the right, faster back gear speeds, to the left, slower back gear speeds.
   8. Motor shaft hand wheel, for turning spindle by hand.
C. Tailstock.
   9. Tailstock clamping bolt.
   10. Dead center, hard and smooth, does not revolve, furnishes bearing for work.
   11. Tailstock spindle.
   12. Oil well, holds oil for lubricating dead center.
   13. Tailstock spindle locking lever.
   14. Tailstock spindle handwheel, operates tailstock spindle screw which transmits motion to tail spindle.
   15. Tailstock slide adjusting screw.
   16. Tailstock slide, has short transverse adjustment on tailstock base. Adjusted by (15).
D. Lathe Carriage.
   17. Tailstock base.
   18. Cross slide, moves at right angles to center line of lathe; operated either by hand or automatically to give the cross feed of the lathe.
   19. Cross feed screw handle, with graduated bushing. Each division of the graduations usually denotes a movement of .001" of the cross slide.
   20. Tool post.
   21. Tool post ring and rocker.
   22. Tool rest. This figure shows compound rest—with graduated base which may be swiveled on cross slide to any desired position.
   23. Compound-rest feed screw handle.
   24. Carriage clamp screw, for clamping the carriage rigidly on the ways. Used when facing large diameters.
   25. Hand wheel for regular (longitudinal) hand feed.
   26. Feed pinion handle.
   27. Longitudinal feed control.
   28. Cross feed control.
   29. Split-nut (or half-nut) handle.
   30. Hand wheel for motor control.
E. Feed Mechanism and Thread Cutting Mechanism.
   31. Reverse gear or tumbler gear handle, for reversing the direction of power feeds and also the rotation of lead screw; middle position is neutral.
   32. Feed pulley—(driver).
   33. Feed pulley—(driven).
   34. Gear on stud.
   35. Intermediate gear.
   36. Gear on screw.
   37. Three-speed feed box.
   38. Quick change gear handle (three positions for three different speeds of feed rod or lead screw).
   39. Feed gears, for transmitting motion to the feed rod.
   40. Positive clutch, for transmitting motion to lead screw.
   41. Feed gear and lead screw clutch handle, to the right, causes positive clutch (40) to operate lead screw; to left, causes feed gears (39) to operate feed rod; middle position, neither lead screw or feed rod revolves.
   42. Feed rod.
   43. Feed rack.
   44. Lead screw (used only for cutting threads).
   45. Feed stop (adjustable), for stopping the longitudinal feed at any desired point.
F. Accessories.
   46. Thread stop.
   47. Large face plate.
   48. Center rest or (steady rest) or (back rest).
   49. Follower rest.
   50. Change gears.
   51. Lathe chuck.
   52. Drill chuck.
exposed bearings should be especially clean before oiling. Use a stick to get in the corners, make a good job of it.

Oil the ways and other flat bearings (the dovetail bearing of the cross slide and over back of the bed where the carriage gib slides) by rubbing on the oil with the fingers.

Every piece that revolves has one or more bearings and every bearing has an oil hole. Find every revolving part by turning the lathe by hand. Find the particular oil holes, be sure they are not stopped up with dirt, and put in sufficient oil to lubricate the bearings thoroughly. Common sense will help one to judge how often a bearing should be oiled and how much oil to use. It is perhaps sufficient to say that it is a crowning disgrace to let a machine get "stuck," and also that a bearing flooded until the oil drips on the floor is an indication of ignorance or carelessness.

While the student is cleaning and oiling is an excellent time to learn more concerning the features of the machine. Short descriptions of the unit parts of the lathe are given immediately after the following hints.

15. A Few Suggestions:
1. Roll up your sleeves.
2. Do not wear a ring.
3. Keep the wrenches, measuring tools, etc., arranged, not thrown, on the lathe board. Never put work, files, tools, etc., on the ways of the lathe, but do put a little oil on them occasionally.
4. Then put the oil can where you can’t jab your face against the spout.
5. Keep your hands away from revolving gears.
6. If you prefer to clean with your finger, the hole in the spindle or a hole you are boring, stop the machine or you may leave your finger in the hole.
7. Ask questions after a reasonable amount of thought and study.
8. Do not move a handle to see what will happen, especially if the machine is running; reason out what the handle is for, or learn in some way, and then move it. Make sure you are right before starting the machine.
9. If you make a mistake do not make the second one of trying to cover it up. The first may sometimes be excusable, the second never. Remember every one respects an honest straight-forward chap.

10. Don't make the same mistake twice.

**DESCRIPTIONS OF LATHE UNITS**

The engine lathe may be said to comprise six essential features: the bed, the headstock, the tailstock, the carriage, the feeding mechanism, and the thread cutting mechanism.

16. The Bed.—The bed is of sufficient depth and width to give rigidity under heavy cuts and is braced inside by cross girths to give stability and strength. Ways or Vs are machined and scraped on top of the bed. The outside or carriage ways afford a perfectly aligned track for the travel of the carriage. The inside ways furnish a permanent seat for the headstock, and a perfectly aligned seat for any desired position of the tailstock. These ways are usually about 90 degrees included angle, and have the tops well rounded to prevent bruising.

17. The Headstock.—The headstock complete comprises the headstock casting; the main spindle; the necessary mechanism for obtaining the various spindle speeds; and also the "reverse gears" which are used to transmit motion from the spindle to the feed rod of the feeding mechanism, and to the lead screw of the thread cutting mechanism. The main spindle of the lathe revolves in two bearings, one at each end of the headstock. These bearings are very accurately machined and scraped to bring the axis of rotation of the spindle parallel to the ways, that is, "in alignment." A hole extends through the entire length of the main spindle and this hole is bored taper at the front end to receive the live center. The bearing surfaces of the spindle, the threaded front end, or "nose," and the taper hole for the live center, are made mechanically accurate and true and great care should be exercised to keep them so. The main spindle of a machine controls the speed of the work, when the work revolves, as in a lathe; or the speed of the cutting tool, if the tool revolves as in
a drilling machine. It must be mechanically true and perfectly aligned because a spindle out of true in any way, or in imperfect alignment, or with improperly adjusted bearings, will cause trouble.

18. The Tailstock.—The tailstock complete comprises the tailstock spindle; the tailstock slide in which is bored the housing for the spindle; the base which is fitted to the inside ways; the screw and handwheel which control the movement of the spindle; the device for clamping the spindle; and the tailstock clamping bolts. The tailstock is for the purpose primarily of giving an outer bearing and support for work being turned on centers. To accommodate different lengths of work, it may be moved along the inside ways and clamped in any position, and in addition, the tailstock spindle (11), (Fig. 16) which holds the dead center (10) is adjustable longitudinally by means of a handwheel, which operates the screw (S). The tail spindle is carefully fitted in its housing and is normally in exact alignment with the main spindle. There is no vertical adjustment whatever, but the tail stock slide may be adjusted transversely; that is, towards or away from the operator by means of the adjusting screws A (a corresponding screw is on the other side of tailstock.) A key way (or spline) is cut about two-thirds the length of the spindle from the inside and a key located on the back side of the housing serves to keep the spindle from turning while
allowing it to slide freely. The spindle is hollow. It is bored taper on one end to receive the center and counterbored on the inside end to receive and hold the bronze nut $N$. The shoulder screw $S$ may revolve freely in the cap $C$ by turning the handwheel (14) but has no end motion. Hence as it turns in the nut $N$ it causes the spindle to move towards or away from the handwheel depending upon which way the handwheel is turned.

To Remove the Dead Center.—Turn the handwheel “back” until the end of the screw hits the end of the center and forces it out.

Cautions.—If the spindle is turned out so far as to run off the screw, be very careful to see that the key way lines up with the key before turning it back.

Be careful not to turn back too far or the spindle will jam against the shoulder of the screw.

19. The Carriage.—The carriage consists of the saddle and the apron. The saddle is fitted to the outside ways and is gibbed to the bed. It is in the form of a letter H, being bridged across the lathe bed to carry the cross slide and tool-rest. The particular function of the carriage is to carry the cutting tool. It may be caused to move (“feed”) by hand or by power along the outside ways, lengthwise of the bed or “longitudinally,” and further, the cross piece of the saddle is machined to provide a way for the tool rest slide so that the cutting tool may be moved by hand or by power to give a “cross feed” at right angles to the long feed. The apron contains the gears and clutches for transmitting motion from the feed rod to the carriage, and also contains the split nut (or half nuts) which engages with the lead screw when cutting threads.

20. The Feeding Mechanism.—Two or more gear trains (series of gears in mesh) one of which is located within the headstock (tumbler or “reverse” gear train) and the other at the end of the lathe (change gear train) serve to transmit motion from the main spindle to the feed rod which extends along the entire length of the bed. From the revolving feed rod, motion is transmitted through various gears which are
located in the apron to finally cause the carriage to move on the ways; or when cross feed is desired, to cause the cross slide to move transversely. The motion of the feed gears in the apron is controlled by means of frictions and the control knobs for both longitudinal and cross feeds are located on the front of the apron within easy reach of the operator.

21. The Thread Cutting Mechanism.—The thread cutting mechanism includes the necessary gears to transmit motion from the main spindle to the lead screw. These may be, and usually are, the same gears used to transmit motion to the feed rod. The lead screw extends along the bed above the feed rod. It is of substantial diameter with a fairly coarse thread cut with great accuracy.

Motion of the lead screw may be transmitted to the carriage by closing the two halves of a split nut over the screw, the split nut being securely fastened to the apron. The thread cutting mechanism should not be used for "feeding" and the feeding mechanism cannot be used for cutting threads.

Note.—The feeding mechanism and the thread cutting mechanism are more fully explained, beginning page 43.

22. A Very Important Precaution.—A habit one should cultivate when learning to run a lathe is to make sure that the carriage moves freely on the ways before starting the machine.

The first thing an experienced machinist always does when going to work on a lathe is to move the carriage on the ways by the hand feed to make sure

1. That the split nut is not tightened.
2. That the feed control is not tightened.
3. That the carriage clamp screw is not tightened.
4. That the ways are oiled.

If the split nut and feed are both tight when the lathe is started the apron will be broken causing several dollars damage. If either is put in when the carriage clamp screw is tightened, the apron mechanism will be strained and injured. If the ways are dry they will become roughened and spoiled.
Questions on Lathe Construction 1

1. What are the ways used for? How are they shaped? How are they finished? Why?

2. Explain how the carriage is moved along the ways by hand. What is the feed rack? What is the feed rack pinion? Why is it called a pinion?

3. How are the ways cleaned and oiled properly? What will occur if they are allowed to become dry?

4. Where is the "live-center" located? Where is the "dead center" located?

5. Why are the centers called "live" and "dead"? Which is hard? Which is soft? Why?

6. Move the tailstock along the bed. What other lengthwise adjustments may be given the dead center?

7. If through carelessness the tail spindle is run off the screw, what caution must be taken regarding the keyway?

8. How is the dead center removed? What caution must be observed?

9. How is the tailstock adjusted sideways? Why is it necessary to first loosen the clamping bolts?

10. How is the tail spindle tightened? Will a quarter of a turn of the locking lever loosen it?

11. Where is the main spindle of the lathe? Why must it be substantial and accurate? Why must the bearings be substantial and accurate?

12. What establishes "the center line of a lathe"? What is it parallel to? When is the dead center "in line"?

13. How is the live center removed? What is the use of the "witness mark"?

14. What are two advantages of the hollow spindle?

15. What part of the carriage is called the saddle? The apron? The tool rest?

16. How is the top of the saddle finished? Why must it be kept clean and well oiled?

17. Can you move the carriage by hand when the split nut is closed? When the feed control knob is tightened? When the carriage clamping screw is tightened? Give reasons.

18. Why does a machinist, before starting to work on a lathe, always try the carriage to make sure it runs freely?

19. How is motion transmitted from the main spindle to the feed rod? To the lead screw?

20. Describe the action of the split nut. Why is it called a split nut?

21. How are the gears in the apron oiled?
SPINDLE SPEEDS

23. How Different Speeds are Obtained.—Because of the wide difference in diameters that may be turned in a lathe, it is necessary to have the spindle revolve at different speeds in order to obtain the proper cutting speed\(^1\) for any size of work, from a very small diameter to the largest that the lathe will swing.

In a direct motor driven machine several different spindle speeds are obtained by the variable speed motor and other changes are made by means of gearing in the headstock. Most machines, however, are belt driven and the speeds are determined by the sizes of pulleys, and also by gearing in the headstock of the machine. In a belt-driven machine motion is transmitted by belting from an engine or a motor to the "line shaft," from the line shaft to the "countershaft" and from a cone pulley on the countershaft to another cone pulley on the machine. (A cone pulley is a pulley with different diameters or "steps.")

24. Countershaft.—Fig. 17 shows one design (Reed) of countershaft with two friction pulleys (or loose pulleys) and a cone pulley. The countershaft runs in bearings which are supported by hangers which are fastened to the stringers above. It is essential that the countershaft is level and also that it is parallel to the line shaft, or the belts from the line shaft to the countershaft will not stay on when the machine is working.

The cone pulley is tightened to the shaft by means of a set-screw. Each friction pulley is normally free on the shaft while its friction ring is tightened to the shaft by two set screws. The friction ring is finished on the outside a trifle smaller than the diameter of the finished inside of the pulley rim. It will be observed that the ring is split between the two lugs. The friction levers are fastened to these lugs and adjusted so as to expand the ring sufficiently to hug the pulley when the wedge is forced between them. The position of the

\(^1\) Cutting speed, see page 70.
wedge is controlled by the fork fastened to a rod to which is attached the "shipper." One of the belts from the line shaft is a "crossed" belt while the other is an "open" belt, consequently, the loose pulleys run in opposite directions. The position of the wedge, therefore, determines whether the countershaft (and the machine) shall be (1) idle, (2) running forward, or (3) reverse. The reverse is seldom used except when cutting threads.

![Diagram of Countershaft](image)

**Fig. 17.—Countershaft. (F. E. Reed Co.)**

**SPEEDS OF PULLEYS AND GEARS**

The beginner should consider carefully the construction of the driving mechanism of the lathe, and understand thoroughly the method of obtaining the different speeds, because similar mechanical principles are involved in nearly every sort of machine tool. The first thing to understand when beginning a study of machine speeds is the driving and driven action of pulleys and gears.

**25. Driving and Driven Pulleys.**—When two pulleys are connected by a belt, motion is transmitted from the driving pulley to the driven pulley.

If the driven pulley and the driving pulley are the same
diameter the driven pulley will make as many revolutions as the driving pulley, because the distance that the belt is carried along by frictional contact with the driving pulley during one revolution is equal to the circumference of the driving pulley, which is equal to the circumference of the driven pulley. If the driven pulley is twice the diameter of the driving pulley (Fig. 18), the driven pulley goes half as fast as the driving pulley, because the distance that a point on the belt is carried along by frictional contact with the driving pulley during one revolution is equal to half the circumference of the driven pulley. If the driven pulley is one-third as large in diameter as the driving pulley, it will revolve three times as fast as the driving pulley because the circumference of the driving pulley is three times the circumference of the driven pulley. That is, the speeds of driving and driven pulleys are to each other inversely as their diameter.¹

26. Driving and Driven (Follower) Gears.—The same reasoning is true with gears with different numbers of teeth as with pulleys of different diameters. If a driving gear having 30 teeth (Fig. 19) is in mesh with a gear having 60 teeth, when the driving gear has made one full revolution it has engaged only 30 of the 60 teeth of the follower gear and turned it only half around. If a driving gear $D$ has 40 teeth, ($a$, Fig. 20),

¹ For formulas on speeds of pulleys see Appendix (page 277).
and the follower gear $F$ has 20 teeth one revolution of the driving gear will revolve the follower gear two revolutions. That is, the velocities of the driving and follower gears are to each other inversely as the numbers of their teeth.¹

27. Simple Gear Train.—Two gears in mesh are called a pair of gears. Three or more gears, the first meshing with the second, the second with the third and so on, constitutes a simple train of gears. The gear between the driving and follower gears in a simple train is known as an idler or intermediate. A small gear in a pair or a train or gears is often called a “pinion.”

Placing an intermediate gear (of any number of teeth) between two gears changes the direction of rotation of the follower gear but does not change the velocity ratio. This is illustrated in $b$, Fig. 20. One revolution of $D$ will engage 40 teeth in $I$ (no matter how many teeth $I$ has), and $I$ will engage 40 teeth in $F$ turning it around twice, the same as in $a$ without the intermediate. Note, however, that in $b$ the direction of rotation of the follower gear $F$ is changed.

Any number of intermediates will not effect the velocity ratio between the driving and the follower gears but the direction of rotation of the follower gear depends on the number of intermediates, thus with one intermediate the follower gear will revolve in the same direction as the driving gear ($b$, Fig. 20) and with two intermediates the direction will be reversed, $c$, Fig. 20.

¹ For formulas on velocities of gears see Appendix (page 279).
28. Compound Gear Train.—If, however, there are two gears fastened to the same shaft, or to a quill, or fastened together in any way so that when one revolves the other must revolve at the same speed, one engaged by the driving gear and the other engaging the follower, these two gears mounted on the same shaft are not intermediates. They are respectively the follower gear of the first pair and the driving gear of the second pair of four gears which form a compound gear train. In a compound gear train the sizes of the gears between the first driver and the final follower cannot be disregarded as in a simple train. This is illustrated in Fig. 21. Suppose $D_1$ revolves 12 times, $F_1$ will revolve four times being three times as large; $D_2$ will revolve 4 times because it is fastened to same shaft as $F_1$ and the final driven gear $F_2$ will make two revolutions.

Questions on Pulley Speeds and Gear Velocities

1. What is meant by the velocity of a gear?
2. The velocities of two gears in mesh (with teeth engaging) varies inversely as the numbers of teeth. What do you mean by "inversely"?
3. A driving gear $D$ has 60 teeth and meshes with a gear $F$ of 40 teeth. How many times will $F$ (the follower gear) revolve when $(D)$ makes 10 revolutions. Why?
4. Introduce an intermediate gear $(I)$ of 120 teeth between $D$ and $F$. How will this effect the result as to the relative speed of $F$? As to direction of $F$.
5. Introduce one more intermediate of any number of teeth in this train of gears. What will be the result as to relative speed of $(F)$? As to direction of $(F)$.
6. What is meant by a simple train of gears?
7. What does the introduction of one or more intermediates serve to do as regards the speed of the follower gear? As regards direction of follower gear? Is this always true in any simple train of gears?
8. The reverse of the lathe is often faster than the forward motion. What causes this?
9. When installing a lathe it is desired to have the slowest speed of the cone pulley 100 r.p.m. The largest step of the cone pulley on the lathe is 10" dia. and the smallest step on countershaft cone pulley is 6" dia., the loose pulley is 12" dia. What diameter pulley will be required on the line shaft which runs at 250 r.p.m.

10. A pulley 12" in diameter is running at 220 r.p.m. and is connected by a belt to a pulley 8" in diameter. How fast does the smaller pulley revolve?

11. What is meant by an inverse ratio?

29. Direct Spindle Speeds.—There are usually two or more series of speeds in all except the smallest sizes of machine tools. They are commonly known as the direct speeds or "back gears out" and indirect speeds or "back gears in."

Different direct speeds in a belt driven (cone pulley) lathe are obtained by changing the driving belt to the various steps on the cone pulley. The cone pulley of the lathe is the driven pulley and changing the belt from a larger to a smaller step on this pulley (and a corresponding larger step on the countershaft cone pulley, which is the driving pulley) increases the speed.

The cone pulley (Fig. 23) is not fastened to the spindle but may revolve freely upon it. The spindle driving gear $D$ called the "face gear" is keyed to the spindle, and to cause the spindle to revolve, it is necessary to transmit the motion from the belt driven cone pulley to the face gear $D$. This may be accomplished directly by locking together the cone pulley and the face gear $D$ by the lock pin $L$. As many different direct speeds may be thus obtained as there are steps on the cone pulley.

30. Lock Pin (Fig. 22).—The plunger $P$ is really the locking pin; when it enters the cone pulley it locks the face gear to the pulley.
The spring $S$ tends to push the plunger into the hole in the cone pulley but in the position shown in Fig. 22 is kept from doing so by the pin $C$. To "put in" the lock pin, turn the knurled knob $K$ until the pin enters the hole $H$ and pull the belt by hand until one of the holes in the pulley comes in front of $P$ and $P$ enters the hole. *Never* start the machine by power until the plunger is in the hole in the pulley. When "taking out" the lock pin pull the knob $K$ until the pin is out of the hole and turn the knob part way around.

31. **Back Gears.**—In order to get a larger number of different speeds, engine lathes, and nearly all other machine tools are equipped with back gears. The function of the back gears is to give a speed to the spindle which is slower than the speed of the cone pulley. This reduction in speed also gives a corresponding increase in power. Some lathes have two or more sets of back gears.

Referring to Fig. 23 the back gears $B$ and $C$ are both fastened to the quill\(^1\) which revolves on the shaft $E$ which is supported by brackets back of the spindle. It will be noted that the ends of this shaft, the bearings, are eccentric (out of center) with the part of the shaft on which the quill revolves.

This construction is for the purpose of changing the position of the back gears, putting them in to engage with the gears $A$ and $D$ or taking them out of mesh by partly rotating the shaft by means of the back gear handle $H$.

32. **Indirect Spindle Speeds, Back Gears In** (Application of Compound Gearing).—It will be observed (Fig. 23) that the back gears are in and the lock pin is out. Gear $A$ is fastened securely to the cone pulley and power is transmitted from the cone and gear $A$ to back gears $B$ and $C$ and from $C$ to face gear $D$ which is keyed to the spindle. If gear $B$ is three times as large as $A$ it will revolve $\frac{1}{3}$ as fast. $B$ and $C$ are both fastened to the same quill and revolve at equal speeds. If $D$ is three times as large as $C$ it will revolve $\frac{1}{3}$ as fast, with the result that $D$ will revolve $\frac{1}{2}$ of $\frac{1}{3}$ or $\frac{1}{6}$ as fast as $A$.

\(^1\) **Quill.**—Hollow sleeve which revolves on a shaft and carries pulleys, gears, clutches, etc.
There are, of course, as many back gear speeds as there are steps on the cone pulley. A lathe with three steps on the cone pulley and with back gears would thus have six spindle speeds, three direct and three indirect.

Fig. 23.—Lathe headstock. In the cut (b) shows a view looking down on the headstock with the back gears supported back of the cone pulley; (a) is an end view and shows the two positions of the back gears and back gear handle, heavy line “in” light line “out.”

33. Revolutions per Minute (R.P.M.).—In order that the beginner may realize the different revolutions per minute (r.p.m.) of the lathe spindle at each of the positions of the con-

Fig. 24.—Speed indicator. One hundred turns of the spindle A causes one revolution of dial B which is graduated in one hundred divisions and every ten divisions numbered. Speeds too fast to count mentally are easily obtained. Rubber tips may be applied to the indicator spindle.

troller handle (or of the belt on the different steps of the cone pulley if belt driven) he may ascertain the different speeds by counting the lower number of revolutions, and by using a speed-
indicator (Fig. 24) for the faster speeds. (Get the number of revolutions for \( \frac{1}{2} \) minute and multiply by 2 to get the r.p.m.)

34. Changing Belts.—When changing the belt from one cone to another remember that the belt “leads” from one pulley to another. As illustrated in Fig. 25 the belt at C is leading from pulley A to pulley B, and at D it is leading from B to A.

A belt is controlled in its leading direction, thus a pressure at C will lead the belt off the pulley B, and a pressure at D will lead the belt off the pulley A.

It is good practice always to throw a belt off with a wrench handle or a stick, but it is often necessary to throw it on by hand. Get fairly close to the pulley, use the flat of the hand, hold the fingers stiff and close and never curl them around the edge of the belt or they might get pinched.

Suppose it is required to “speed up” a lathe, the operator will run the belt down to the smallest step on the lathe pulley, and by putting his right hand inside the belt and pulling it steadily towards him will speed up the cone pulley; then with a slight pressure in the proper direction and with a tossing movement of his left hand, he will find it easy to place the belt on the larger step of the cone pulley on the countershaft.

Sometimes if the belt is fairly tight it is well to use a belt pole,¹ but with a little experience one can usually place the belt on any step in a few seconds. It is often advisable to have the

¹ Belt Pole.—A pole of sufficient length to reach the line shafts and countershafts, with a pin or stud 4” or 5” long fastened at one end at right angles to the pole. Used to push belts on or off.
back gears and the lock pin both out when changing the belt because this allows the driven pulley to run free and makes the change much easier.

Note.—For description of how belts are fastened see appendix (page 231).

THE OPERATION OF LATHE FEEDS

The regular or longitudinal feed of the lathe is the travel of the whole carriage along the ways of the lathe parallel to the axis of rotation of the main spindle, that is, parallel to the center line of the lathe. The cross feed of the lathe is the travel of the cross slide at right angles to the center line of the lathe.

Each of these feeds may be operated in either direction by hand (called hand feed) or automatically in either direction (called power feed). The amount of the hand feed is controlled absolutely by the hand of the operator: it may be slow or fast, and the lathe may or may not be running. The power feed, however, is an automatic movement of the tool carrier of a desired definite amount for each revolution of the spindle. Motion is transmitted from the revolving lathe spindle through gearing to the feed rod, and from the revolving feed rod through the apron gears to the carriage, or to the cross feed screw, as desired. This mechanism is interesting because it involves the use of gears, clutches, and certain other mechanical principles.

36. The Tumbler Gear Train and the Change Gear Train.—Fig. 26 illustrates the gearing from the spindle to the feed rod. The gear $Sp$ is keyed to the spindle and transmits motion through the tumbler gear $R_1$ to the inside stud gear or fixed stud gear $FS$ which is keyed to the stud shaft. (The use of the gear $R_2$ will be explained presently.) This train of gears is called the tumbler gear train or sometimes the reverse gear train and is usually within the headstock casting.

The stud gear $St$ is a change gear, that is, if desired it may be removed and a larger or smaller gear put on in its place.
Since it also is keyed to the stud shaft it revolves when the fixed stud gear F.S. revolves. The stud gear transmits motion through the intermediate gear I to the screw gear\(^1\) Sc which also a change gear. Sc is keyed to the lower feed box shaft and motion is transmitted from this shaft to the upper feed box shaft by the gears E and B. When the clutch C is in the position shown in the diagram motion is transmitted from the upper feed box shaft to the feed rod by the feed gears G1 and G2.

\(^1\)Screw Gear.—In the older lathes the screw gear was keyed direct to the lead screw, hence its name. It was used only for thread cutting. In such lathes feed motion was transmitted by a belt from a cone pulley on the stud shaft to a cone pulley on the feed rod.
When the sliding clutch member $C$ is moved to engage with the fixed clutch member $H$, the lead screw is caused to revolve and the feed rod stops because the feed gears $G_1$ and $G_2$ are no longer engaged.

It may be stated here that the tumbler gear train and the change gear train illustrate a practically standard principle of lathe construction, and that the gears $E$, $B$, $G_1$ and $G_2$ illustrate a principle of construction employed by several lathe manufacturerers.

36. The Tumbler Gears.—The operation of the tumbler gears $R_1$ and $R_2$ is illustrated in Fig. 27. These gears are carried on a bracket which is pivoted on the stud shaft and operated by means of the reverse gear handle (31 Fig. 15), to any one of the three positions shown. It will be observed that the two tumbler gears are intermediate gears between the spindle gear and the fixed stud gear, and are so mounted on the bracket as to make it possible for the operator to have either one intermediate or two intermediates in mesh between the driven gear and the driving gear, or to throw them both out of mesh with the driving gear. That is with the three positions of the reverse gear handle the stud shaft may have (1) forward movement, (2) reverse, (3) no motion. The function of the tumbler gears is to reverse the direction of revolution of the feed rod when turning or of the lead screw when cutting threads.

37. The Intermediate Gear in the Change Gear Train.—The bracket which carries the intermediate gear is arranged
to pivot on the stud shaft through a certain distance (see Fig. 28). The stud on which the intermediate gear revolves is adjustable to any position in a fairly long slot in the bracket. With these two adjustments it is possible to arrange the inter-

![Diagram of gear train](image)

**Fig. 28.—Change gear train showing intermediate gear bracket.** When changing gears, first loosen the nuts A and B, then the binding screw C, and finally the nut D, being careful not to let the intermediate fall too hard against the bottom of the bracket. When the gears on stud or screw or on both are changed as desired, first lift the intermediate to engage the stud gear, pinching a piece of paper between them to allow for clearance, and tighten D, then swing the bracket to tighten paper between the intermediate and the screw gear and tighten C.

mediate gear to engage both the stud gear and the screw gear no matter what size these gears may be.

**38. Quick Change Gears.**—Certain modern lathes are equipped with what are termed "quick change gears." In such a lathe it is not necessary to change gears on the stud and screw to give the different feeds desired or to cut the different pitches of threads. The change is made by merely shifting the position of one or more handles.
The Hendey lathe is perhaps the most noted example of the lathe with quick change gears. In this lathe the lead screw is splined and is used to operate the feed mechanism as well as for thread cutting. The diagram (Fig. 29) illustrates the gearing by means of which thirty-six different feeds and also thirty-six threads, may be obtained by changing the positions of two handles. One handle serves to slide the feathered gear $B$ along the shaft $M$ to a position in front of any one of the cone of gears $A$, and also to lift a tumbler gear (not shown) which meshes with $B$ and which when lifted to engage a gear in the cone of gears $A$ transmits motion from $B$ to that gear. (In the diagram the gear $B$ as shown with the tumbler gear engaged would transmit motion to the seventy (70) tooth gear in the cone.) All of the twelve gears in the cone are keyed to the lead screw. Thus with a given speed of the shaft $M$ any one of twelve different speeds of the lead screw may be obtained because each one of the cone of gears $A$ is, when engaged, a follower gear of a different size. Three different speeds of shaft $M$ may be obtained by changing the position of the handle which slides the gears $L, K$ and $H$.

$^1$Feather.—A sliding key fastened in a gear or clutch to keep it from turning while allowing it to slide lengthwise.
to the three positions shown. Therefore twelve speeds of the lead screw to each of three speeds of \( M \) gives thirty-six different feeds, or if the split nut is used, thirty-six different threads.

39. The Apron Mechanism. (Flather Lathe).—The way in which motion is transmitted from the revolving feed rod through the apron gears to cause either feed to operate is shown in Fig. 30. The feed pinion (7) engages the feed rack (see 43, Fig. 15) and when it revolves causes the table to move along the ways (longitudinal feed). The gear (10) engages a small pinion which is keyed to the cross feed screw and when it revolves causes the cross feed screw to turn. The other gears in the apron are used either to transmit motion from the feed rod to gear (7) to give longitudinal feed, or to gear (10) to give the cross feed, and are thus explained:

Motion is transmitted from the feed rod to the bevel
pinion (1). (The feed rod has a long spline (keyway) and the pinion is feathered on the feed rod. The carriage may be moved along the ways of the lathe and the pinion may slide over the feed rod but when the feed rod turns the bevel pinion turns.) The pinion (1) turns the bevel gear (2). The small spur gear (3), shown in dotted lines, is fastened to the bevel gear (2) and meshes with gear (4) and gear (4) meshes with gear (9). Therefore when the feed rod revolves the gears (1), (2), (3), (4) and (9) revolve, whether or not either feed is operating.

To obtain the longitudinal feed tighten the feed knob a which operates the friction between gear (4) and gear (5); (gear (5) is shown in dotted lines) and causes (5) to revolve with (4). Gear (5) engages gear (6) and when (6) turns it causes the feed pinion (7) to move and thus feeds the carriage.

To obtain the cross feed tighten the cross feed control knob b. This tightens the friction between gear (9) and gear (10) and as stated above, gear (10) meshes with a small pinion (not shown) fastened to the cross feed screw.

The hand wheel c operates the pinion (8) which meshes with (6). Therefore when the hand wheel is turned it causes a movement of (8) to (6) to (7) and moves the carriage.

The two halves of the split nut (11) and (12), Fig. 31 and also Fig. 30, are operated by the handle d through the cam slots (13) and (14) in the disc. The pins (15) and (16) which lie in the slots are caused to move toward each other when the handle is moved in the direction of the arrow and away from each other as the handle is moved back.
40. Apron (American Lathe).—A type of apron that is very popular in lathe construction is illustrated in Fig. 32. One feature is the bevel gear feed reversing mechanism. It will be noted that instead of one bevel pinion feathered on the feed rod this apron has two, one on each side of the large bevel gear. The engagement of the pinions with the bevel gear is controlled by a handle on the front of the apron. Each pinion may serve to revolve the gear but being on opposite sides revolves it in an opposite direction. The direction of rotation of the bevel gear determines the direction of all the other apron gears and consequently the direction of the feeds. When the handle is in the middle position (as in the illustration) neither pinion engages the bevel gear and there is no feed.

Note.—Special attention is called to the bevel gear reversing mechanism. It is probably the most widely used reversing mechanism in machine construction. It is used in almost every kind of machine tool. Be sure to understand the principle of the construction.

Another valuable feature is the non-interfering lever. This lever is arranged in such a way as to make it impossible to close the split nut when either bevel pinion is in mesh with
the gear. This "fool-proof" construction therefore makes it impossible to throw in the feed when the split nut is closed.

Still another feature of the apron illustrated is the chasing dial which is used when cutting threads without reversing the lathe. This is explained in the chapter on Threads and Thread Cutting. (See page 199.)

Questions on Lathe Construction II

1. What is the object of having several spindle speeds (revolutions per minute) in a lathe?
2. What do you understand by cone-pulley drive?
3. Explain the use of a countershaft. Why are there two "loose pulleys"?
4. Explain the action of the friction clutch in the loose pulley.
5. When are the "back gears" used? Explain the eccentric action of the back gear shaft.
6. Is the lock pin used when the back gears are "in"? When they are "out"? Give reasons.
7. Explain in detail the method of putting the lock pin in.
8. Explain in detail the principle of back gear action.
9. Make a sketch of the tumbler gears, illustrating how the stud shaft may be driven either forward or reverse.
10. What is the purpose of the intermediate gear in the change gear train of the lathe?
11. Make a sketch which will show the principle of the action of the cone of gears in the quick change gear mechanism of a lathe.
12. Make a sketch which will show the principle of the action of the friction clutch, as used in the apron mechanism.
13. Make a sketch which will show how motion may be transmitted from a bevelled gear on the feed rod through gearing to cause the cross feed screw to revolve.
14. Make a sketch which will illustrate the principle of the bevel gear reversing mechanism.

HIGH-DUTY LATHES

41. Features of High-duty Lathes.—The use of high-speed steel and the resulting increased power requirements have made necessary in machine design a much stronger construction of the various parts of the machine and a more efficient drive. Keen competition on the part of the manufacturers has developed many interesting types of direct gear driven
and clutch controlled mechanisms which give a greater number of spindle speeds than is possible in the cone pulley type, and also a considerable increase in the power delivered. These machines are differentiated from the cone pulley type by the terms "Geared Head" when applied to lathes, "Selective Gear Drive" or "Constant Speed Drive" when applied to shapers, milling machine, etc.

Other features of the geared head or constant-speed drive are the ease and flexibility of the speed changes (by means of levers instead of by changing belts) and its adaptability to direct motor drive, or to driving direct from the line shaft.

With the increase in driving power of the geared head lathe a corresponding increase in strength and rigidity in the machine as a whole is necessary, particularly the apron and other parts of the feeding mechanism. For this reason high-duty lathe aprons are usually provided with a back plate which affords a back support for the studs. The pinions and gears are made from specially selected steel and the studs are case-hardened and ground.

All modern high-duty lathes up to 24" are usually provided with the "quick change" gear mechanism for feeds and thread leads. This mechanism usually provides 32 or more (depending on the make) feeds and as many thread changes, any one of which is instantly available by operating one or possibly two levers. These levers, together with an index plate, are conveniently arranged on the gear box.

42. Double Back Gears or Reducing Gears.—For the purpose of obtaining a wider range of speeds than would be possible with one set of back gears, some lathes are provided with double back gears or "reducing gears."

In Fig. 33, a, b and c show the arrangement of the gears in the direct motor driven, double back geared, Flather 14" lathe. The view a is a vertical section through the headstock and does not show the back gears. The views b and c are horizontal sections showing the back gears but not the gearing from the motor.

Referring to a, Fig. 33, $G_1$ is a gear keyed to the motor
shaft and transmits motion through the compounding gears \(G_2\) and \(G_3\) to \(G_4\). Gears \(G_4, F,\) and \(A\) are fastened together and revolve freely on the main spindle. (In this head these gears may be said to replace the cone pulley and gear \(A\) in the cone pulley drive as shown in Fig. 23.)

**Fig. 33.**

By locking together the gear \(G_4\) and the face gear \(D\) which is keyed to the main spindle, as many different direct speeds may be obtained as there are speeds of the motor.

Referring to views \(b\) and \(c\), Fig. 33, the gears \(A, F, G_4\) and \(D\) are the same as shown in \(a\), with the power transmitted to \(G_4\). Gears \(B, E\) and \(C\) are the back gears. Gear \(C\) is fastened to the quill and \(B\) and \(E\) are feathered and may be moved to the right as shown in \(b\) or to the left as shown in \(c\).
Pull out the lock pin, move the back gears to the right and put them "in" as shown in b. Gear E will mesh with gear F and gear C will mesh with the face gear D. If F and E are of the same size, and C is one-third as large as D, then D will revolve one-third as fast as F, thus giving a series of speeds one-third slower than the direct speeds of G₄.

To obtain a still slower series of speeds put the back gears out and move B and E to the left and then put them in again as shown in c. Gear E no longer meshes with F but gear B meshes with A, and C meshes with D. If A is one-fourth as large as B, gears B and C will revolve one-fourth as fast as A. Gear D will go one-third as fast as C and therefore \( \frac{1}{3} \) of \( \frac{1}{4} \) or \( \frac{1}{12} \) as fast as A and G₄.

43. The Geared Head.—The gearing mechanism as applied to machine tools for the purpose of speed changes or feed changes may be divided into two types of gear combinations—the tumbler gear type in which a tumbler gear may be moved to mesh with any one of a cone of gears, and the sliding gear type in which certain gears mounted on a sleeve may be moved into or out of engagement with their respective mates.

Each may be used in connection with other gearing through positive or friction clutches or other means for the purpose of increasing the range of speeds.

Various designs of drives of either type are to be found in nearly all kinds of modern machine tools—lathes, shapers, milling machines, etc., and the following illustrations and descriptions of the geared heads of two lathes made by different manufacturers will serve to show an example of each. The Hendey lathe illustrates the tumbler gear principle and the American lathe illustrates the sliding gear principle.

44. Hendey Lathe Geared Head.—Fig. 34 shows the development of the gear trains and Fig. 35 shows the arrangement of these gears in the head stock. In the following description the letters refer to either figure.

The mechanical features consist of the wide faced Driving Pulley; the Power Shaft S on which is carried a Tumbler Bracket or Rocker R with Driving Pinion D and the Intermedi-
ate Tumbler Gears ($I_1$ and $I_2$); the Main Spindle with a Three Gear cone $G_1, G_2, G_3$, the Pinion $A$ and the Large Face Gear $F$, all running free on the spindle, (the pinion $A$ and the cone $G_1, G_2, G_3$ revolving as one); a Positive Tooth Clutch $C$ feathered to the spindle and working between the large gear of cone $G_3$ and face gear $F$; and the Back Gear Quill with Back Gears $B_1$ and $B_2$ cast integral and pinned to the back gear shaft.

**Fig. 34.**

**Gear Runs**

*Direct*, Clutch lever (L, Fig. 35) to the left; clutch member engaging $G_3$.
- $D-I_1-G_1$ (Fastest speed)
- $D-I_1-G_2$
- $D-I_1-G_3$
- $D-I_1-I_2-G_3$

*Indirect*, (through backgear). Clutch lever to the right; clutch member engaging gear $F$.
- $D-I_1-G_1-A-B_1-B_2-F$
- $D-I_1-G_2-A-B_1-B_2-F$
- $D-I_1-G_3-A-B_1-B_2-F$
- $D-I_1-I_2-G_3-A-B_1-B_2-F$ (Slowest speed)
There are eight spindle speeds in geometric progression, four direct and four through the back gears. The back gears remain in mesh with the spindle gearing and are engaged by the clutch. The clutch has immediate control over two spindle speeds one direct and one indirect (which two depending on the position of the intermediate gears \((I_1)\) and \((I_2)\)). The clutch may be used to stop and start the spindle which makes for convenience and speed in operation. The gear runs for the eight speeds are shown under Fig. 34.

45. American Lathe Geared Head.—Fig. 36 shows the development of the gear train and Fig. 37 shows the arrangement of the gears in the head with the cover raised. In the following description the letters and numbers refer to either figure.

The mechanical features of this head consist of the wide faced Driving Pulley; the first shaft or Driving Shaft \(S_1\) which carries the Driving Gears \((3)\) and \((1)\) with the Friction clutch member \(K\) working between them; the Second Shaft \(S_2\) with the keyed gear \((9)\), the feathered Pinion \((10)\) and the freely
revolving Quill or Sleeve $Q_1$ on which are mounted the gears (4), (6), and (2); and the Main Spindle with the large Face Gear (11) (which is keyed to the spindle and carries the feathered member of the Positive Clutch $J$), and the loose Sleeve $M$ to which is keyed the gear (8) and the other member of positive clutch $J$ and on which is feathered the Quill $Q_2$ carrying the sliding gears (5) and (7). There are eight spindle speeds in geometric progression any one of which is immediately available.

![Diagram of lathe construction](image)

**Fig. 36.**

<table>
<thead>
<tr>
<th>Gear Runs</th>
<th>Levers</th>
</tr>
</thead>
<tbody>
<tr>
<td>1–2–4–5</td>
<td>A–C–E</td>
</tr>
<tr>
<td>3–4–5</td>
<td>A–C–F</td>
</tr>
<tr>
<td>1–2–6–7</td>
<td>B–C–E</td>
</tr>
<tr>
<td>3–4–6–7</td>
<td>B–C–F</td>
</tr>
<tr>
<td>3–4–5–8–9–10–11</td>
<td>A–D–F</td>
</tr>
<tr>
<td>1–2–6–7–8–9–10–11</td>
<td>B–D–E</td>
</tr>
<tr>
<td>3–4–6–7–8–9–10–11</td>
<td>B–D–F</td>
</tr>
</tbody>
</table>

The control levers are arranged at the front of the head and are charted $A–B$, $F–E$, and $C–D$ respectively.

The $A–B$ lever moves the sliding gear quill $Q_2$; from middle or neutral position to $A$ slips (5) into mesh with (4); to $B$ slips (7) into mesh with (6).

The $F–E$ lever operates the driving gear friction clutch mem-
ber $K$; from neutral position to $F$ engages gear (3), to $E$
engages gear (1). The friction clutch $K$ has immediate con-
trol over two spindle speeds, one through (3) and one through
(1) (which two depending on the positions of the other levers).
The clutch may be used to start and stop the spindle which
makes for convenience and speed in operation.

\begin{figure}[h]
  \centering
  \includegraphics[width=\textwidth]{fig37.png}
  \caption{Fig. 37.}
\end{figure}

The $C-D$ lever operates the sliding member of the positive
clutch $J$ and the sliding pinion (10) simultaneously; to $C$,
throws clutch in and sliding pinion out of mesh with (11) (for
the four direct speeds); to $D$ throws clutch out and sliding
pinion into mesh with (11) (for the four indirect speeds). The
gear runs are shown under Fig. 36.
CHAPTER III

CUTTING TOOLS AND CUTTING SPEEDS

It is very necessary for the student in machine shop practice to realize in the beginning that the cutting tool is a most important factor. There is nothing in shop work that should be given more thoughtful consideration than cutting tools. If one understands the principles underlying the successful action of the cutting tool, he has gone a long way in becoming expert in its use.

Time is always wasted if an improperly shaped tool is used. A dull tool is a disgrace to the shop. It is fairly difficult for the beginner to hold a lathe tool against the grinding wheel and grind it just how and where it should be ground. He must first learn how it should be ground, and he must acquire by practice the knack of grinding it. "Practice makes perfect."

46. Cutting Tool Efficiency.—The action of a cutting tool depends primarily on three things—(1) the rigidity of the work, that is, of the piece itself, and the manner in which it is held in the machine; (2) the rigidity of the tool—its size and the way in which it is held; and (3) the shape of the cutting tool.

The machine tool builder takes care of the design of the machine to give the necessary strength and stability. The cutting action of the tool, however, depends on its shape and its adjustment in the holding device. This is especially interesting to the machinist because most of the cutting tools he uses in the shop must be shaped—or at least sharpened—and adjusted by himself. The experienced workman will never use a dull tool, a poorly shaped tool, or a tool improperly held if he can help it.
FORGED LATHE TOOLS

1. Side Tool.—Used for facing. Has side clearance and front clearance. The front is ground at an angle sufficient to clear a lathe center. It has side rake but no front rake.

2. Diamond Point Tool.—A turning tool which is practically obsolete; largely superseded by a shape suggested by Dr. F. W. Taylor.

3. Bent Diamond Point Tool.—Diamond point tools, side tools, thread tools, cutting off tools, and others are often bent either to the right or left as desired. They are often called off-set tools.

4. Turning Tool.—This is substantially the shape recommended by Dr. Taylor after years of experimenting as being the most efficient form of turning tool. Medium front rake 8°, medium side rake 14°. Sometimes called a bull-nose tool.

5. Cutting Off or Parting Tool.—Explained fully on page 128.

6. Round-nose Tool.—Made with round cutting edge of any required radius and used for filleted shoulders and also for "Necking" or "Grooving."

7. Tool for Turning Brass.—Ground substantially like a small round nose. Has no rake because brass is soft and a tool with rake is apt to dig.

8. Square-nose Tool.—Very useful for truing live centers or turning short tapers. Used also with a coarse feed for a very light finish cut on cast iron. Has no top rake.

9. Goose-neck Tool or Spring Tool.—Used for turning large fillets and other forming work, when a wide cut is required, also for finishing cast iron and for water finish on steel. Any tendency of this tool to spring is away from the work.

10. Dutch-nose or Shovel-nose Tool.—A very efficient tool for facing work of a fairly large diameter when considerable stock is to be removed. It is fed toward the center. Used also for finish turning steel, coarse feed and light chip. Used with sodawater for a high finish. Has front rake but no side rake. Has clearance on both sides and may be used either right or left.

11. Spotting Tool.—This tool is ground to an angle of about 120° to correspond with the angle formed by the lips of a drill. Note that each lip is given clearance but in opposite directions because the cutting force is up on one lip and down on the other. Used to make a central spot in which to start the drill. Often called centering tool.

12. U. S. Standard Thread Tool.—Ground accurately to 60° gauge to form a thread on a cylinder. Explained more fully in chapter on Threads and Thread Cutting.

Fig. 38—Forged tools for lathe work.
47. Cutting Tools Used in Lathe Work.—There are many operations that may be done in a lathe—turning, cutting a thread, boring a hole, etc., and each kind of operation requires its own particular kind of tool. (See Fig. 38.) The chart shows forged tools which in recent years have been superseded to a certain extent, by the more economical tool holder with its "bit" or "blade." (See Figs. 43 and 44.) The various kinds of tools will soon become familiar to the young man in the shop; these charts, however, are instructive at this time as illustrating an excellent example of the importance of a knowledge of principles. The underlying principles of cutting angles, clearance angles, etc., which are explained in the following paragraphs, are the same for any machine shop cutting tool. If one knows why a turning tool is ground a certain shape, why it has certain angles of clearance, etc., and has practised enough to hold it and manipulate it skillfully against the grinding wheel to give it this shape, he has learned 90% of tool grinding. He will soon learn the shapes and angles of the other forms of tools, and he has acquired the knack of grinding that is fundamentally the same for all.

The material of which the cutting tool is made must be hard and tough enough to withstand the resistance of the cutting force, therefore tools for cutting metals, which are essentially machine shop cutting tools, are made of tempered steel, one of the hardest and toughest materials known.

48. Cutting Angle.—The action of any cutting tool in any material is that of a wedge, prying apart or separating the substance of the material. The angle of the wedge is the cutting angle of the tool. (See Fig. 39.)

The harder the material to be cut the more the cutting edge must be supported, that is the cutting angle must be greater. The cutting angle which is correct for wood is not substantial
enough to stand up under the strain of cutting iron or steel, the cutting edge, not being sufficiently supported against such a severe crushing force would soon crumble and the value of the tool would be lost. The proper cutting angle for a metal cutting tool is 60 degrees to 80 degrees depending on the hardness of the metal to be cut.

49. Clearance Angles. —The action of a cutting tool in machine work is a peeling operation. As the chip is peeled off the work, the only friction is that caused by the chip in contact with one side of the wedge. The other side of the wedge clears the work by a small angle which is called the side clearance. (Sec Fig. 39.) In general,

![Diagram showing cutting angles and clearance angles]

Fig. 40.

A front clearance of 10° or 12° is usually ground on a forged tool. Note the position above the center of the new tool A so as not to give excessive front clearance. Note B, which represents the position of the same tool after being sharpened several times, set on center to give correct clearance.

![Diagram showing tool positions]

Fig. 41.

the angle of side clearance should not be more than 6 degrees because the greater the amount of metal a cutting edge has under it, or the more it is backed up, the longer it will stay sharp.
The direction of the force which is exerted against the turning tool is along a line tangent to the circumference of the work at the cutting point. (See Fig. 40.) As the usual practice is to set the turning tool on center or a little above, it is necessary to have clearance at the front of the tool, so that it will not rub. This front clearance is usually about 10 degrees. The design of the tool post (ring and rocker) permits of using tools of various heights. Since 10 degrees is practically a standard clearance, the height of the tool will determine the amount to set the tool above center to resist the tangential force of the cut and still have sufficient clearance. This is illustrated in Fig. 41.

50. Rake Angles.—To obtain the required cutting angle it is necessary to grind the top of the tool on a double slope, from the front and from the side, otherwise the angle will not be acute enough. The slope from the point is called the \textit{front rake} (sometimes called back rake) and the slope from the side is called the \textit{side rake}. (See Fig. 42.) The number of degrees of these rake angles depends, of course, on the cutting angle required, the more rake a tool has the less cutting angle it has, that is, the sharper the wedge is. A cutting angle of 70 degrees is average for turning cast iron and also for turning tool steel. A cutting angle of 60 degrees is average for soft steel. After the clearance angle is ground on the tool the rake is ground to give the required cutting angle for the material at hand. Brass is softer material and therefore would not naturally require as heavy a cutting angle as steel. However, no rake is given a brass cutting tool because of the tendency of the tool to "hook in" or "dig in" to the soft material.

51. Patent Tool Holders.—High speed cutting tools will do practically double the amount of work that can be done with a carbon steel cutting tool, therefore, for most purposes, lathe tools and other cutting tools are made of high speed
steel. This steel is much more expensive than carbon steel and for reasons of economy, tool holders have been invented

![Images of various tool holders](image)

**Fig. 43.—Representative types of patented tool holders.**

![Image of high speed tool set](image)

**Fig. 44.—Lathe Set, O. K. high speed tools and holder. (O. K. Tool Co.)**

which will securely and rigidly hold turning and boring tool "bits," side tool "blades," etc., made of the high speed steel.
This saves not only tying up a great amount of steel but the expense of forging also.

The cuts, Fig. 43, show certain types of patented tool holders. Fig. 44 illustrates a type of tool holder which is

provided with tools of various shapes ready for use. The chart (Fig. 45) shows how the turning tool bit may be ground to serve several different purposes. Remember it is the resourceful man that is valuable and remember also that if one
can skillfully grind one of these tools he can skillfully grind the others.

52. Keep Cutting Tools Sharp.—A cutting tool carefully ground will stay sharp under correct working conditions for a considerable time, but as soon as it is noticeably dull it should be reground or the tool and possibly the work will be ruined. A dull tool tears rather than cuts the material; it springs the work, and does not make a smooth cut. To keep cutting tools sharp is a most important factor in efficient machine work.

53. Judgment in Tool Grinding.—It is an acknowledged fact that lack of judgment in the grinding of tools costs thousands of dollars every year in the wastage of materials alone. To sharpen a dull tool shall it be ground on the top, on the front, or on the side, or a little here and there? No rule can be given. Each tool grinding calls for judgment. Examine the particular tool; how many times may it be ground on the top before it is worn out? How much may it be ground on the side before it is too thin? How much of the life of the tool is lost by grinding it on the front? Keep these things in mind when sharpening a tool.

54. Grinding Cutting Tools.—When grinding carbon steel, care must be taken not to bear on too hard or the edge will become blue and the temper lost. A wet grinder should be used if one is available. It is not so easy to burn the temper out of a high speed cutter but it is easy to cause surface cracks by not having water enough. Have plenty of water and do not bear on too hard; give the wheel a chance.

Do not hold the tool in one place, move it across the face of the wheel. Do not “whittle” the tool but grind one continuous cut tipping it slightly to give the side clearance (see Fig. 46), tipping it a little more as you swing the point.
around as at b and then less again as you finish the round nose c.

A tool bit should not be ground in a holder, first, because it is clumsy and inefficient and second, because one is liable to grind the holder. If occasionally the holder is ground a little, soon it is ruined. Fig. 47 shows the correct way to hold the turning tool bit. The beginner should grind a practice piece of machine steel (preferably a little larger than the tool bit so as not to get them mixed) and acquire the knack before attempting to grind an expensive tool bit. It may be fairly difficult at the start to grind the front clearance properly because the bit when in use is held at an angle in the holder. Until the eye is trained, use a gauge. The 60 degree center gauge is suitable, if 10 degree front clearance is wanted, because in most of the holders the tool bit is set at an angle of 20 degrees with the horizontal (Fig. 48). For any other than 60 degrees cutting angle it is easy to cut out a small sheet metal gauge of the angle desired.

Fig. 47.

Fig. 48.

DON'TS IN TOOL GRINDING

1. Don’t grind stupidly, know where and why and how to take off the metal.

2. Don’t hold the tool as if your fingers were paralyzed, hold it securely enough to control it.
3. Don't whittle the tool, grind it.
4. Don't hold the tool left handed or otherwise awkwardly; hold it properly, it's the easiest way.
5. Don't be afraid to use plenty of water.
6. Don't hold the tool in one place or you will cut a groove in the wheel.
7. Don't use a wheel that is grooved or out of round if you can help it.
8. Don't grind on the side of a wheel except when necessary. When it is necessary you will want a flat surface and it won't be flat if you or anyone else has cut grooves in it.
9. Don't hold the smaller tools on the tool rest, support them in the left hand and rest this hand.
10. Don't use the tool rest with more than $\frac{1}{16}$" space between it and the wheel.
11. Don't make a round nose of a thread tool or a thread tool of a round nose; it is wasteful of material and in the end wasteful of time.

Questions on Cutting Tools

1. What is the general shape of the cutting edge of a turning tool for lathe work? What are the disadvantages of a sharp point on a turning tool?
2. What is meant by cutting angle? How many degrees are included in the cutting angle of an average turning tool? Why not 90 degrees? Why not 30 degrees?
3. What is meant by clearance angle? How much side clearance has a lathe turning tool? How much front clearance? What is the disadvantage of too great a clearance angle?
4. What is meant by front rake? Side rake? What is the object in giving rake to a tool?
5. Other things being equal, will a tool with side rake cut equally well if fed in either direction?
6. What is meant by negative rake?
7. What is a right hand turning tool?
8. What is an off-set tool?
9. It may be stated that a cutting-off tool has five clearance angles; where are they?
10. It may be stated that a side tool has four clearance angles; where are they?
11. When it is necessary to sharpen a side tool, is it ground on the top, on the side, or on the front? Why must judgment be exercised?

12. What are the advantages of a tool that has been oil-stoned?

13. If a tool rubs on the work, what faults may be found?

14. What is the chief advantage of a tool holder and bit?

15. Should the bit be ground in the holder or should it be removed before grinding? Give reasons.

16. Name three things on which the action of a cutting tool depends.

**SPEED, FEED AND CHIP**

55. Definitions of Speed, Feed and Chip.—*Cutting speed* in any machine shop operation is expressed in *feet per minute*. In lathe work it is the number of feet measured on the circumference of the work that passes the cutting edge of the tool in one minute. If it were possible to measure the exact length of the chip removed in one minute it would measure the cutting speed in feet per minute.

The *feed* in lathe work is the amount the tool advances for each revolution of the work. For example, in turning a cylinder with $\frac{1}{32}$" feed it will require 32 revolutions of the work to move the carriage one inch. The machinist speaks of "coarse feed" and "fine feed." These terms mean nothing except when applied to lathes of practically the same size. What might be regarded as fine feed on a large lathe would be a coarse feed on a small lathe.

By the term "*chip*" or "*cut*" in lathe work is meant the *depth of cut*. Suppose a cylinder of machine steel 2" in diameter is put in a lathe and a cut made reducing the diameter to 1$\frac{3}{8}$". Regardless of the speed or feed, the chip, that is the depth of the cut is $\frac{1}{8}$". It should now be clear what the foreman means when he says "Give it a higher speed," "Try a coarser feed," "Take a bigger chip," or "Take a deeper cut."

56. The Time Element.—One of the most important problems entering into machine shop work is the time element. The time it takes to produce a finished piece of work depends largely on the rate at which the metal is removed from the original stock. The rate at which the metal is cut off depends
on three things, namely, the chip, the feed and the cutting speed. Take for example the turning operation.

First.—It is obvious that the cutting edge of the tool takes a deeper chip, if it reduces the diameter \( \frac{1}{4}'' \) than if it reduces it only \( \frac{1}{6}'' \). It will be folly to take two chips if this reduction in roughing size is necessary. One factor then is the size of the chip.

Second.—If every time the work revolves the tool is fed into the metal one-sixty-fourth of an inch, it will remove a chip only half as thick as if it were fed one-thirty-second of an inch. If practicable to set the feed for one-thirty-second why not get the piece turned in half the time? Another factor then is the amount of feed.

Third.—If this work is 2'' in diameter and revolves 70 times in a minute, a point on the circumference will travel about 30 feet in a minute. If the cutting tool will stand 60 feet per minute cutting speed it will not be efficient to turn at half this speed. The third factor then is the cutting speed.

There is a new problem of cutting speed, feed and chip for every job on every machine in the shop. After awhile the workman becomes expert enough to attend to these things without any trouble. At the start, however, these problems require close attention and certain calculations.

57. Cutting Feeds and Speeds.—There are so many conditions that determine the proper feed and chip to use that it is impossible to give any set rule for either. The shape of the tool, the way in which it is held, the kind of steel from which it is made are factors; also the kind of material being cut, whether machine steel or tool steel, brass or cast iron; the shape of the piece being cut, whether it is rigid or inclined to spring; the nature of the cut, whether it is a roughing cut or a finishing cut, are all factors which must be taken into consideration when obtaining an efficient chip or feed.

Conditions also govern the rate at which the tool will cut, and no table can be given that will apply in all cases. Fortunately however, there are certain well established average cutting speeds for various metals.
**Average Cutting Speed with Carbon Steel Tools**

<table>
<thead>
<tr>
<th>Material</th>
<th>Speed</th>
</tr>
</thead>
<tbody>
<tr>
<td>Annealed tool steel</td>
<td>25 feet per minute</td>
</tr>
<tr>
<td>Machine steel and wrought iron</td>
<td>35 feet per minute</td>
</tr>
<tr>
<td>Cast iron</td>
<td>40 feet per minute</td>
</tr>
<tr>
<td>Brass</td>
<td>100 feet per minute</td>
</tr>
</tbody>
</table>

Average cutting speed with high speed steel tools at least double the above.

Cutting speeds must not be confused with revolutions per minute. A piece 2 inches in diameter will have to make 5 times as many revolutions per minute as a piece 10 inches in diameter to give the same cutting speed. In other words, each different diameter must have a different number of revolutions per minute (r.p.m.) to give the same cutting speed. If the beginner will calculate for the first few jobs the r.p.m., necessary to give the required cutting speeds, after awhile he will become so accustomed to seeing the machine work properly that he will be able to set up without calculations and almost without thought.

58. **Cutting Speed Calculations.**—Cutting speed (excepting the shaper and planer) is the rate at which a point on the circumference travels. In the case of a lathe it is the circumference of the work; in the case of a milling machine or drill press it is the circumference of the milling cutter or of the drill. In machine shop practice, when speaking of sizes, the diameter is expressed, not the circumference. Also these diameters are given in inches, while cutting speed is expressed in feet. To find the circumference of a piece of work (or of a drill or milling cutter) multiply the diameter by 3.14 and to reduce to feet divide by 12. The diameter multiplied by 3.14 and this divided by 12 is equal to .26 times the diameter. \[
\frac{\text{Dia.} \times 3.14}{12} = .26 \times \text{Dia.}
\]

Instead of multiplying the diameter by 3.14 and dividing by 12 in every problem it is much quicker to multiply the diameter by .26. That is, the circumference in feet is always equal to .26 times the diameter in inches.

Further, if one had a job that figured 2 feet in circumference it would take 20 r.p.m. to give a cutting speed of 40 feet per minute; if the job figured \(\frac{1}{2}\) foot in circumference it would take...
80 r.p.m. to give a cutting speed of 40 feet. In both cases the r.p.m. is equal to the cutting speed divided by the circumference in feet. From these examples the following may be deduced: To obtain the r.p.m. necessary to give any required cutting speed, multiply the diameter (in inches) by .26 and divide the cutting speed by this product.

Since .26 is so nearly \( \frac{1}{4} \); it may be stated that for all practical purposes the number of r.p.m. may be calculated by the following:

**Rule:** To obtain the number of r.p.m. necessary to give any required cutting speed, divide \( \frac{1}{4} \) of the diameter into the cutting speed.

**Example.**—A piece of machine steel 2\( \frac{3}{4}'' \) in diameter is to be turned in a lathe. What number of r.p.m. is necessary to give a cutting speed of 70 feet per minute?

**Solution:** \[ \frac{2.25}{4} = .56, \quad 70 \div .56 = 125 \text{ r.p.m.} \text{ Ans.} \]

**Questions on Cutting Feeds and Speeds**

1. What do you understand by "time element" in machining a piece of work?
2. Name four things that may determine the proper feed for turning.
3. What is the difference between cutting speed and revolutions per minute?
4. How many revolutions per minute are necessary to turn a piece of work 1\( \frac{1}{2}'' \) in diameter at a speed of 30 feet per minute?
5. What number of r.p.m. is necessary to give a cutting speed of 40 feet per minute on work 2\( \frac{3}{4}'' \) in diameter?
6. What rule is used to find the r.p.m. of a drill to give a required cutting speed?
7. Is this same rule used to obtain the r.p.m. of work in a lathe to give the required cutting speed?
8. In turning a cast iron pulley 12'' diameter, how many r.p.m. will be necessary to give a cutting speed of 40 feet per minute?
9. Why is it that machine steel can be turned at a higher speed than tool steel?
10. If the r.p.m. and the diameter of the work are known how may the cutting speed be found?
11. In cutting speed calculations the constant .26 \( D \) (or \( \frac{1}{4} D \)) is used. How is the constant obtained?
CHAPTER IV

THE SCALE, CALIPER, SNAP GAUGE AND MICROMETER

59. Scales.—Among the most useful tools in the machine shop are the steel rules or "scales." They are made in a variety of kinds as "spring tempered," "flexible," "narrow," etc., and in lengths from 1" to 48", the most popular being the spring tempered, 6" scale. Most steel rules are graduated, that is, marked by fine lines upon each edge of both sides, and often at the end, in different subdivisions of an inch. The different graduations are classified by number; for example: the No. 7 graduation which is perhaps the most used has 64ths, 32ds, 16ths, and 100ths on the four edges respectively. The No. 8 graduations, which many prefer, is graduated in 64ths, 32ds, 16ths, and 8ths. The "flexible" scale, which is very popular with machinists and tool-makers, is graduated on one side (two edges) only.

It will be observed (Fig. 49) that the graduation lines on the scale are of different lengths, the 64th lines (not shown) are the shortest and the lines marking 32ds and 16ths and so on are successively longer, the inch line being the longest. This is for the purpose of quickly reading the measurement.

When reading a scale measurement find the nearest "significant" graduation whether eighths, quarters, halves, and of course whole inches and add or subtract as the case may be, to obtain the fraction of the finer graduation. To illustrate:

\[ 3\frac{3}{16}'' \text{ is } 3\frac{1}{2}'' \text{ plus } \frac{1}{16}''; \quad 4\frac{13}{16}'' \text{ is } 5'' \text{ minus } \frac{1}{16}''; \]
\[ 6\frac{3}{32}'' \text{ is } 6\frac{1}{4}'' \text{ plus } \frac{1}{32}''; \quad 2\frac{15}{32}'' \text{ is } 2\frac{1}{2}'' \text{ minus } \frac{1}{32}'' .\]

A machinist never says "two-eighths" or "four-sixteenths" or "six thirty-seconds" but such expressions as "a thirty-second over half" or "one sixty-fourth under three quarters,"
etc., are commonly used. Also the terms "a scant sixteenth" or a "full thirty-second" and "half a sixty-fourth" are used.

60. Calipers.—A caliper is a tool used for measuring work. Calipers are made in several styles such as "spring calipers," "firm joint calipers," "transfer calipers," for both inside and outside measurement.

Fig. 50 shows "outside" and "inside" spring calipers. The spring at the top tends to keep the legs set taut against the pressure of the adjusting nut.

Fig. 51 shows a firm joint caliper. A caliper of this kind is usually preferred only in the larger sizes.

Fig. 52 shows a "transfer caliper." The special feature of the transfer caliper is that it may be used inside of chambered cavities, over flanges, etc., removed and re-set without losing the size calipered. This is done by loosening the lock nut which binds one leg to the auxiliary leaf and moving this leg (while the joint b is tight) to clear the obstruction, then moving it back against the stop on the leaf where it will show the exact size measured.

The caliper may be used as a measuring tool or as a gauge.
The machinist generally uses it as a measuring tool; he takes a cut for a short distance, measures it with the caliper, reads the measurement on the scale to see how much more of a chip he has to take and proceeds accordingly. When the caliper is used to gauge work, pieces already turned for example, it is first set to the size required.

61. Measuring with a Caliper (Fig. 53).—Accuracy in caliper measurement depends on the sense of touch and to get a delicate touch of the caliper on the work it should be held lightly and not with a "grab grip." It is not difficult to demonstrate the "feel" of one-thousandth of an inch with a caliper. A lathe mandrel¹ tapers about \( \frac{1}{1000} \)" in two inches of its length. Get a mandrel from the tool room and carefully set the caliper as if to measure this mandrel 3" from one end,

¹ Mandrel.—Sometimes called "arbor," a machine shop tool, with accurate centers, which may be forced into the hole in the piece to be machined (such as a pulley or gear), thus providing centers on which turning or other machine work may be done. For complete description of mandrel see page 119.
then without changing the setting, caliper the mandrel 5" from the end and note the difference in the feel.

If sufficient force is exerted a caliper set to an inch may be pushed over a piece 1 1/8" in diameter. On the other hand a caliper set to 1 1/8" can be made to "just touch" a diameter of 1" if held canted as shown in Fig. 54. Hold the caliper exactly at right angles to the axis of the work. *Never caliper the work while it is running*; it is not accurate and the caliper is liable to get caught and broken.

62. Setting and Reading an Outside Caliper.—To set an outside caliper hold the scale in the left hand with the end against the little finger as shown (a Fig. 55) and in such a position that the light falls directly on the scale. Hold the caliper in the right hand, in such a position that it may be adjusted by the adjusting nut between the thumb and finger. Place the end of one leg of the caliper against the end of the scale and against the finger so that it will not slip around,

![Diagram](image)

(a) Correct.
(b) Incorrect.

Fig. 55.

and then adjust the other leg to the desired graduation on the scale. Hold the caliper true and looking squarely at the end to be set to the line, adjust the caliper until that end seems to split the line. In this way a caliper may easily be set to within .002" or .003" of the exact size required. A firm
joint caliper is held in about the same way but must be adjusted by rapping lightly against some solid object.

If a plug gauge of the desired size is available or if a piece is to be duplicated the caliper may be set exactly by adjusting it to the given gauge or piece.

To read an outside caliper it is held substantially as above except that as it is not to be adjusted, the adjusting screw should not be touched.

Calipers are not efficient for accurate measurements but they are efficient for measuring stock, roughing cuts, lengths and any dimensions that need not be extremely accurate. The caliper may be used if necessary for very close measurements but it is easier and quicker and surer to use a micrometer or a gauge.

In rapid production work it is usually advisable to use gauges for determining the correct sizes. For cylindrical work the snap gauge is used and where a slight variation over or under nominal size is allowable a "limit gauge" is used.

![Fig. 56.—Types of limit snap gauges.](image)

Fig. 56 illustrates three styles of limit snap gauges. It will be understood that for rough turning, the difference between the "go on" dimension and the "not go on" dimension is much greater than could be allowed in the finishing operation, and further, in different classes of work the limits allowed for finishing vary greatly. The advantage of the gauge shown at (c) lies in the fact that it is adjustable.

63. The Micrometer Caliper.—Micrometer calipers form convenient and accurate instruments for fine external measurements. They are made in different sizes and styles to measure
all dimensions to twenty-four inches. They are graduated to read to thousandths of an inch, and one-half and one-quarter thousandths are readily estimated. Some of the calipers have verniers by which readings can be obtained to ten thousandths. (See Appendix, page 288.)

The essential parts of the micrometer are: frame \( F \) (Fig. 57) to which are fastened the anvil \( A \) and the hub \( H \), the spindle \( S \) which is threaded \( \frac{1}{40}'' \) pitch about half the length and which is fastened securely to the outer end of the thimble \( T \) and revolves freely in a fixed nut within the hub. By turning the thimble right or left one full turn, the spindle advances toward or recedes from the anvil \( \frac{1}{40} \) or .025 of an inch.

A line is drawn on the front of the hub parallel to the axis of the spindle and this line is divided into 40 equal parts per inch, every fourth part being numbered 0, 1, 2, 3, etc.

The bevelled edge of the thimble is divided into twenty-five equal parts every fifth part being numbered 0, 5, 10, etc.

When the micrometer is closed the bevelled edge of the thimble coincides with the zero line on the hub, and the zero line on the bevel coincides with the horizontal line on the hub. Open the micrometer one full turn or until the 0 on the bevel again coincides with the horizontal line on the hub and the micrometer is opened \( .025'' \).

Open the micrometer three more full revolutions. The bevelled edge of the thimble splits the 1 line on the hub and the opening is \( \frac{1}{40} \) or \( \frac{1}{10} \) or as usually read, “one hundred thousandths.”

To read the micrometer count the numbers 1, 2, 3, etc., as \( .100, .200, .300 \); add each line beyond the figures as, \( .025, -.050 \) \( -.075 \) then add the number of thousandths indicated on the thimble. To illustrate: Fig. 58, \( a \) reads \( .300'' \) even, \( b \) reads
.350" (.300 + .025 + .025), c reads .437" (.400 + .025 + .010 + .002), d reads .687 1/2" (.600 + .075 + .012 1/2).

A thousandth of an inch is considered a very large amount in the estimation of a skilled mechanic. A good idea of the value of a "thousandth" may be had from measuring the large and small ends of a standard mandrel of 3/16" diameter or under. Measure the large end; measure the small end; subtract, and note that the difference is about .002" or .003".

Again measure the large end and without changing the setting of the micrometer place it over the small end and note the "shake" between the measurement obtained at the large end and the diameter at the small end.

64. Holding a Micrometer.

—Fig. 59 indicates clearly the proper way to hold the micrometer in order to accurately measure a piece held in the hand. Note carefully the position of the fingers; the micrometer is held by the little finger or the third finger, whichever is less awkward, against the palm of the hand, which allows the spindle to be operated in either direction with the thumb and index finger. The
correct way to hold a micrometer when measuring work not held in the hand, is shown in Fig. 60.

When making a measurement be sure the micrometer is held square across the diameter. Turn the spindle down to the work but not down too hard. It is easy to spring a micrometer .001 or .002 and this not only gives a false measurement but injures the micrometer.

It seems easy for some people occasionally to read a micrometer .025 over or under; such a mistake is inexcusable. It is even more careless to add 25 and 5 and call it 35, or 75 and 5 and read it 85. Be careful when using a micrometer to hold it properly, to adjust it carefully and to read it accurately.

Questions on Measuring

1. How do you read a scale, using the significant graduations?
2. How do you hold a caliper to read the measurement on a scale?
3. What is the method of adjusting a firm joint caliper?
4. What do you understand by sensitive touch?
5. In the limit gauge illustrated, Fig. 56, what is the reason for rounding two corners and bevelling the other two?
6. In the limit gauges illustrated in Fig. 56, why are the "not go on" gauging surfaces smaller than the "go on" surfaces?
7. What is the difference between the "go on" and the "not go on" dimensions of a limit gauge?
8. What part of an inch does one revolution of the thimble of a micrometer move the spindle? Why?
9. Into how many divisions is the hub graduated? How are these divisions numbered?
10. Into how many divisions is the bevelled edge of the thimble graduated? How numbered? Why?
11. Describe the proper way to hold a micrometer (a) holding the work in the hand, (b) when the work is in the machine.
12. Explain the difference in using a micrometer and a C clamp.
CHAPTER V

CENTERING

65. Holding Work in the Lathe.—Work to be turned in the lathe may be held between centers; fastened in a chuck; clamped to the large face plate; or supported in the steady rest. Work that is to be faced or turned true with a finished hole is held either on a mandrel between centers; or on a special arbor the shank of which fits the spindle; or on a plug which is fastened in the chuck and turned to fit the hole.

![Diagram of workholding in a lathe](image)

Fig. 61.

A large proportion of engine lathe work is machined “on centers.” Sixty degree countersunk holes, called centers, are drilled and reamed in both ends of the piece to be turned. These holes fit the 60° lathe centers and the work is thus supported (Fig. 61).

The work thus centered is usually driven from the small face plate by means of a dog\(^1\) which is securely clamped to it

\(^1\) *Dog* (Fig. 62).—In lathe work, a driver. Made in many shapes and sizes and used to drive the work from the face plate. The work is gripped by a set screw and the finished end should be protected by a piece of copper or soft brass between the work and the hardened end of the screw.
on the live center end. The work turns with the live center which acts as a support only, and on the dead center which acts as a support and also as a bearing. (See Fig. 61.)

66. Importance of Carefully Locating the Center.—It is very important to center carefully a piece to be turned. A piece carelessly centered may not have sufficient stock to clean, that is to finish all over, and therefore be spoiled. Even if it has sufficient stock to clean, the fact that the centers are out of true will necessitate a big chip on one side of the diameter and a small chip on the other. This unevenness of cut takes more time and may cause inaccuracy.

It is especially important to center tool steel carefully. In its manufacture the heating necessary for rolling the steel into bars of various shapes and sizes causes a decarbonization\(^1\) of the outside to the depth of perhaps \(\frac{3}{64}\) of an inch. If this is not altogether removed when machining a piece of tool steel, when the piece is heat treated, that part of the piece from which the decarbonized portion has not been removed, will not harden.

The rolling process also causes a difference of density of the metal toward the center of the bar, and turning off more from one side of the bar than the other will cause the piece to warp in hardening.

Care must be taken in centering cast iron. In making an iron casting the molten metal is poured into a sand mold. The hot metal coming in contact with the cold sand causes the surface of the casting, the "scale" it is called, to become con-

\(^1\) Tool steel has a definite content of carbon, usually about 1 per cent. When the steel is heated and exposed to the air, as in rolling, the carbon on the surface is lost and this surface layer is said to be decarbonized.
siderably harder than the interior. Also a considerable amount of sand is fused with the iron in the surface of the casting. These conditions serve to render the surface of a casting very hard and brittle, and when machining it is important that the point of the cutting tool be well under this scale in order not to rub off the cutting edge and ruin the tool.

67. The Centering Machine (Fig. 63).—Modern machine shop equipment includes a centering machine which automatically holds round pieces central with the drill spindle thus making it unnecessary to otherwise locate the centers.

Fig. 63.—Centering machine. *(D. E. Whiton.)*

However, a centering machine may not be available, and further, certain pieces cannot be held in such a machine even if one is at hand, therefore other methods of locating the centers must be used. Some of these methods are here described.

68. Methods of Locating Centers.—First, measure the stock to see if it will finish to the length required, and then rub chalk on the ends to make the center locating lines show more distinctly.

*Hermaphrodite Caliper Method* (Fig. 64).—Set the caliper

1 *Hermaphrodite Caliper.*—A combination of a caliper leg and a divider leg. Very useful for locating and testing centers and for laying off distances from an edge. Commonly called “morphy.”
to about the radius of the piece. Place the caliper leg on the circumference at the extreme end of the piece and with the point draw an arc near the center of that end. Move the caliper leg about one quarter of the circumference of the end each time and draw three more arcs. The four arcs will form an approximate square, the center of which is the center required.

Center Square Method (Fig. 65).—Hold both limbs of the center head\(^1\) of a combination set\(^2\) tightly against the surface of

---

\(^1\) Center Square.—A blade so arranged with a square as to have one edge of the blade divide the angle of the square equally. It is used in scribing radial lines on the ends of a piece of work.

\(^2\) Combination Set (Fig. 66).—Contains a square, a mitre, a center square, a protractor and a level.
the work and with a scriber, rule lines at about right angles to each other. The intersection of these lines is the required center.

*Surface Gauge Method* (Fig. 67).—When it is required to locate the center in an irregular shaped casting or forging a surface gauge¹ may be used. For example: a bolt blank may be forged with the head off-set. If the head is centered true the body of the bolt will run out of true and may not clean. To properly center such a piece place the bolt on parallels or on V-blocks, adjust the surface gauge scriber to the approximate center of the body and draw lines on both ends forming squares. The centers of these squares are the desired centers.

*Diagonal Line Method* (Fig. 68).—Rectangular pieces are easily centered by drawing diagonal lines.

69. Use of Center Punch.—After the center is located, catch the piece in a vise and select a center punch (Fig. 69) with the point ground to about 90°. Tap lightly at first, being sure that the point is central, then make the indentation large enough to support the piece while it is being

¹Surface Gauge.—A tool with a heavy base supporting a sharpened scriber which is adjustable as to height. It is used for gauging, marking, or transferring the distance from a flat surface to a point or line on the work.
tested between the lathe centers. It is important to test the accuracy of the center punch marks, especially until experience has trained the eye. Do not put on a dog, or do not start the lathe but revolve the work by hand on "dead centers," marking the "high spot" on each end with chalk. Catch the work in the vise again, and by tipping the center

![Diagram of center punch usage](image)

**Fig. 69.**

punch, drive the center enough toward the "high spot" to bring it right. Then hold the punch perpendicular and make the indentation symmetrical.

70. Drilling and Reaming the Center.—The combination drill and countersink\(^1\) (Fig. 70), is the correct tool to use for making the center hole. Obtain a small drill chuck with a

![Diagram of drill and countersink tools](image)

**Fig. 70.**
taper shank that will fit the taper hole in the main spindle of an engine lathe (or a speed lathe) and, first making sure that the centers are in approximate alignment, remove the

\(^1\) *Combination Drill and Countersink a, (Fig. 70).*—A tool which combines the center drill and the 60° center reamer (or countersink). It is more efficient and has largely superseded the small drill, and the center reamers shown in b and c.
center and put in the chuck. See that the drill is sharp, then grip it firmly in the chuck. Put one of the center punch marks on the drill point and steadying the work, bring the dead center carefully into the other center punch mark.

Hold the work loosely with the left hand allowing the tool-rest to support the wrist. Start the lathe (fastest speed) and feed by turning the tail spindle hand wheel slowly until the center is reamed to the correct size. (See paragraph 71.) Be careful not to break off the drill when drawing the work back from the center reamer. It is best to keep the work against the tail center as it is backed away from the drill thus avoiding any tendency to pry off the drill. Use lard oil or cutting compound for steel or wrought metal. Drill cast iron dry.

The dead center and live center should be in line for all operations except taper turning. Form the habit, when going to work on a lathe, of making sure that the centers are in approximate alignment.¹

71. Size of Centers.—Combination drills and countersinks are furnished in various sizes. There is no rule for the sizes of centers. The proportion of the size of the center to the diameter of the work is largely a matter of judgment and depends on the material, the amount of stock to be removed, the cut to be taken, the number and kinds of operations to be performed and the shape of the piece.

Do not make the centers too big, they should be just large enough to withstand the resistance of the cut. If, after facing, the center is too small it is easy to make it larger. The size of the drilled hole a, (Fig. 71) should be about \( \frac{3}{4} \) to \( \frac{1}{2} \) the diameter of the reamed hole b. It should be sufficiently deep that the point of the lathe center does not touch, thus insuring a real bearing on the cone surface. This extra depth also provides a reservoir for oil.

¹To ascertain if a lathe is in approximate alignment.—(1) Note that live center runs true and carefully bring the dead center to within \( \frac{1}{16} \)" of the live center. A very little off-set will be easily seen. (2) "Witness marks" on slide and base of tailstock will usually indicate whether or not the alignment is close. Setting a lathe in accurate alignment will be explained later.
For the preliminary cylindrical turning operations the following may be used as a guide:

<table>
<thead>
<tr>
<th>Finished diameter of work</th>
<th>Diameter of center</th>
<th>Diameter of center drill</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\frac{1}{2}''$</td>
<td>$\frac{1}{8}''$</td>
<td>$\frac{1}{16}''$</td>
</tr>
<tr>
<td>$1''$</td>
<td>$\frac{3}{16}''$</td>
<td>$\frac{3}{32}''$</td>
</tr>
<tr>
<td>$1\frac{1}{2}''$</td>
<td>$\frac{1}{4}''$</td>
<td>$\frac{1}{8}''$</td>
</tr>
</tbody>
</table>

Too much emphasis cannot be placed on the importance of having correct centers—the right shape, the right size, smooth and clean.

72. Removing a Broken Center Drill.—If the drill should be broken in the center hole it may often be removed by a sharp blow on the side or end of the piece of work, if not it must be softened by annealing\(^1\) when it may be drilled out.

---

To Anneal Steel: (Usual Method).—Heat thoroughly to a dull red, then cover with ashes, powdered charcoal, lime or asbestos and allow to cool.

"Cold Water" Annealing (which is much quicker than above).—Heat to a dull red, then allow to cool slowly until no red can be seen in a fairly dark place, then plunge into cold water.

---

Questions on Centering

1. Does it make any difference whether the "morphy" is set a little more or a little less than the radius?
2. Why do you rub chalk on the end?
3. How large an indentation do you make with the center punch before testing? Why? What is the angle of the point of the center punch?

\(^1\) anneal.—To reduce brittleness and increase softness. (Also eliminates strains in steel caused by forging and rolling.)
4. After both centers are located and center punched, how are they tested for accuracy? What does the "high spot" indicate?
5. If the center punch marks need changing slightly, how is this done?
6. Why should care be taken to have the centers in the work fairly true?
7. Why is the combination drill and countersink an efficient tool?
8. How many r.p.m. should the combination drill and countersink be run?
9. Name at least two things that determine the proper size of the centers to be reamed.
10. Why does the drill break so easily? What is a good way to pry it off?
11. How tightly do you grasp the work when you are drilling it. Where do you rest your hand? Give reasons.
12. On what materials is a cutting lubricant used when drilling or reaming? What is the use of the lubricant?
13. How is the center square used for finding the center?
14. If the nature of the work is such that the use of a "morphy" or a center square is not practicable, for example, a bolt head that is forged off center from the body, how may the centers be located so that the body will run true?
15. What is a centering machine? What are its advantages?
16. Give two reasons for the drilled hole being deeper than the point of the center reamer.
17. If a drill has been broken off in the center hole, why must it be annealed before it can be drilled out?
CHAPTER VI

FACING

73. Adjusting the Work on Centers.—After the work is carefully centered, tighten a dog firmly on one end. (A protecting piece of copper under the dog screw is unnecessary unless that end of the work is finished). Put a drop of oil in the center hole in the other end. Place the work between centers and adjust the tail center. The experienced machinist always puts the work on the live center first, being sure the tail of the dog does not bind in the slot of the face plate (Fig. 72). Then he runs the tail center into the center hole, and is careful that it enters without hitting around the hole two or three times. To avoid chatter, he brings the center up until there is no shake of the work between centers, but not too tight because the work must be free to turn or the center will be scored and possibly spoiled. That is, he adjusts the work between the centers carefully. Unless the work is too large, the machinist always tries this adjustment by wiggling the tail of the dog in the face plate slot.

When placing the work between centers do not hold it at arms length or try and reach over the tool post. Hold the piece steady by resting the hand against the tool rest or the tail spindle or in any convenient way.
74. The Facing Operation.—After centering and adjusting the work on centers the next operation is facing the ends. Use the right hand side tool for facing. (See Fig. 73.) The whole cutting edge of the side tool should not be set at right angles to the center line of the work, but the point of the tool should be slightly in, that is, toward the work.

To obtain a smooth finish the point of the side tool should be slightly rounded with an oil stone, or so ground as to present a short flat surface to the work. (See Fig. 74.) The length of this flat should be greater than the amount of “cross feed.” The work should be faced from the center hole outward and not from the circumference toward the center. Usually if considerable metal is to be removed by facing, it is advisable to rough by feeding the tool sideways and perhaps a turning tool bit properly ground for this purpose may be more efficient.

If there are several pieces to face, it will be advisable to get a half center (Fig. 75) from the tool room, but having only one or two pieces, or in the event of a half center not being available, the following method is used.

Adjust the work between the centers (no slack and not too tight) start the lathe and with the right hand on the cross feed
handle and the left hand on the long feed handwheel, bring the tool as close to the center as convenient, and then into the work until a light chip is taken around the center hole. Stop feeding with the left hand (long feed) and with the right hand feed slowly outward, that is, toward the circumference, thus squaring or facing the end. The amount of chip and feed must be determined by conditions. The machinist usually prefers to face the smaller pieces (under 1" diameter) by hand feed. The beginner will go slowly on the first piece.

The above method of facing will leave a slight fin around the center hole. To remove this fin bring the side tool to the surface just faced and to the fin. Ease the tail center about $\frac{1}{32}"$ and clamp the tail spindle; when the lathe is started the work will have a tendency to ride on the tool and make a clean cut around the center hole. Do not feed toward the circumference again but stop the lathe and remove the work.

The end, especially around the center, should always be faced clean to give an equal bearing on the center. If the work is not faced clean, the center hole will have a tendency to wear unevenly and become out of round. If there is any doubt about the stock being long enough, a tell-tale (an unfinished portion of the end) should be left to prove, if necessary, that it was short.

Facing is often called squaring. The ends of a properly faced piece will be square (flat). The surface may be tested with the edge of a scale. To produce a flat surface the lathe centers must be in line; a surface like a, Fig. 76, indicates that the dead center is off-set towards the operator, and a surface like b shows the center is off-set away from the operator.

If considerable metal is to be removed later in the turning operation it is better not to face to exact length until all of the roughing cuts are taken. This will provide a larger center for
the roughing cuts than would look well in the finished piece, and also, finishing the ends after roughing out the work will remove the burr around the center caused by the great pressure of the heavy roughing cut.

A piece of steel is usually cut off somewhat over the finished length because the hack saw or cutting off machine of any kind is not intended for accuracy. The machinist before starting to work on any special piece measures the stock to be sure it is long enough. Face off about half the over length from one end; remove the dog; lay off the length; (set the hermaphrodite caliper to the length required if it is a short piece); place the dog on the end that is faced and finish the other end nearly to the line.

An accurate way of measuring the length is with an outside caliper. Sometimes on the longer pieces it is necessary to use a scale to measure direct or possibly two scales end to end. When using this method one must be particularly careful to have the end of the scale square and flush (even) with the finished end of the work. Face one end, then using the scale to measure with, scratch a line on the other end to indicate the amount to be removed, and face to this line.

Remember always that the function of the tool is to cut the metal, not to scrape it. Do not take a great number of chips. Learn as quickly as possible to judge the facing chip that will remove a given amount of stock.

Facing accurately and quickly is a job which requires attention to business and good judgment. The beginner who with reasonable speed can face a piece to length and have the ends square and smooth, with no ridges, no undercut near the center, and no fin, has a right to be proud of his work.

The term radial facing is applied to work of a comparatively large diameter and will more properly be discussed under the subject of chucking. (See page 128.)
Questions on Facing

1. Why should the piece be measured before proceeding to face?
2. It may be stated that a side tool has four clearance angles; where are they? What three are ground on the tool?
3. If a patent holder is used, why should the blade be removed from the holder before grinding?
4. If it is necessary to sharpen a side tool, should it be ground on the top, on the front, or on the side? Why?
5. Should a side tool be oil-stoned? Give reason.
6. Why must the centers be in line? What is the effect if the tailstock slide is off-set toward the operator? Away from the operator?
7. How do you adjust the work between centers?
8. Do you always finish one end before rough facing the other end? Give reasons.
9. What is a “tell-tale”?
10. If you have several pieces to face, is it advisable to rough face them all before finishing any? Give reason.
11. When is hand feed used in facing? When is power feed used? Give reasons for both.
12. Is the tool fed in a direction toward or away from the center? Why?
13. Why is it advisable to slightly round the point of the side tool?
14. What do you mean by setting the side tool “on center” and pointing “slightly in” toward the work?
15. For a facing cut, how do you lay off the length? How do you measure the length?
16. When should a piece not be faced to exact length before the turning operation?
17. When facing an end, one roughing cut and one finishing cut is usually sufficient. Why take more?
18. If considerable metal is to be removed what tool other than a side tool may be used? Why is it more efficient? How may it be fed?
19. What causes a wavy or shattered appearance of the surface being squared?
20. What is a half-center? Why is it more than half?
21. What is a “fin”? What is the best way of removing a fin? Another way?
22. When is a lubricant used in facing?
23. Name at least two reasons why a piece is faced.
CHAPTER VII

TURNING IN A LATHE

Turning in a lathe is accomplished by causing the work to revolve while the tool being fed longitudinally peels off a chip. The same definition may be applied to turning in a turret lathe or to turning outside or inside (boring) in a "boring mill." The principles and methods involved in turning and boring are the same for the larger machines as for the smaller machines. The cutting action of the tool is the same whether the work is held between centers or in a chuck; on the face plate in a lathe; or on the table as in a boring mill; or in a special fixture in either machine. The principles of feeds and speeds, of alignment, of measurement, etc., etc., are fundamental machine shop principles.

In rapid production work where hundreds or thousands of duplicate pieces are made within narrow limits of exactitude, the turret lathe with its special holding fixtures, its facing, turning,
Fig. 78.—Shows a bank of turret lathes set up for rapid production of duplicate pieces. (Courtesy Warner and Swasey Co.)

Fig. 79.—Boring in a boring mill. (Courtesy Universal Boring Mill Co.)
boring, and other tools, each arranged in the turret head or on the cross slide for its particular operation, is most advantageously used. For the larger castings and forgings the boring mill is most adaptable for many turning, facing or boring operations. It must be remembered that these machines are after all modifications of the lathe. The lathe is the most important machine; it is the most widely used and the most adaptable of machine shop tools and turning in a lathe is one of the most important operations in a machine shop.

In order to be able intelligently to set up a lathe for an accurate turning job it is necessary to understand certain of the principles and methods involved.

**PRINCIPLES OF TURNING**

75. Position of the Dead Center.—The center line of the lathe is determined by the center of rotation, or the axis, of the main spindle, and is parallel to the ways. It is therefore parallel to the line of travel of the carriage which moves on the ways, and also parallel to the line of travel of the turning tool.

The live center and the dead center are equidistant from the horizontal plane of the ways. The live center having no adjustment, is fixed in its position.

The design of the tailstock, however, permits of transverse adjustment of the dead center, that is, the dead center may be moved off center toward or away from the operator, or it may be adjusted from off center to an exact central position.

When turning work in a lathe, if the dead center is exactly in line with the live center, the distance from both centers to
the line of travel of the tool is the same. In this case the radii and consequently the diameters of the turned work are everywhere equal, and the turned piece is a perfect cylinder. It is said to be turned straight. (See a, Fig. 80.)

If, when turning, the dead center is "off-set" or "out of line," the distance from the line of travel of the turning tool to one center is greater than it is to the other, and the diameter of the work which is being turned changes constantly from one end to the other and the work is not straight. It is said to be turned taper. (See b, Fig. 80.)

76. Accuracy of the Live Center.—The method of procedure in turning a cylinder in a lathe, is to turn half its length or more from one end, then reverse the piece on centers (changing the dog to the opposite end of the work) and turn the rest of the cylinder. The circumference of any properly turned piece is concentric with its center line. If the live center runs true and a piece is turned as above, it will be found that the part of the cylinder first turned will run absolutely true when reversed, and when the remainder is turned the two cuts will meet exactly. This work is right, and is possible only when the live center runs perfectly true.

Suppose that the live center runs out and the above operations are made. When the piece is reversed it will be noticed that the middle of the work runs out one-half the amount the live center is out of true. It is impossible to have the cut which is made after the piece is reversed meet flush with the cut already made and the piece cannot be turned straight to size. (See Fig. 81.) Further, suppose the live center runs out of true and a portion only of the length of the work is turned. If the center does not run out too much, and the stock is large enough to clean, the turned portion will be round and straight but it will not run true on dead centers and it will not run true
in a lathe or other machine in which the centers are true. Many times it has been discovered after several later operations (in which the centers were not used) that the work has been spoiled in turning; this means delay and waste. The live center must run true.

*How to Determine if the Live Center Runs True.*—One method of determining if the live center runs true is as follows: Tighten a tool holder (reversed) in the tool post and bring the end fairly close to the center; hold a piece of paper under the revolving center and look down between the tool holder and the center. Any eccentricity of the center may readily be observed.

**77. Cleaning and Truing the Lathe Centers.**—The shanks of the lathe centers are taper and fit taper holes in their respective spindles. Any dirt, chips or burrs, either on the shank or in the taper hole will cause the center to run out of true.

![Square Nose Tool](image_url1)

![Side Tool](image_url2)

*Fig. 82.*—When truing the center, have the tool exactly "on center," do not take too wide a chip, run the lathe slowly and apply lard oil or cutting compound with a brush.

Clean the center with the palm of the hand, and if any nick or burr is felt, oil-stone it off. Clean the tail spindle hole with clean waste on a stick, or with your finger. To clean the main spindle hole, push a piece of waste through the whole length and wipe the taper hole with your finger but never while the lathe is running. *Never clean any moving part of any machine.* If the live center runs out after cleaning, it may be trued up with a square nose tool *a* (Fig. 82) or a side tool *b* (Fig. 82). If a right hand side tool is used the direction of rotation of the spindle is reversed, that is, in shop language

*Note.*—To turn center with compound rest see page 166.
the lathe must "run backward." A center should be filed very little, if at all, after turning. Test the angle of the center with a center gauge as shown in Fig. 83. Some lathes have been abused to such an extent that the taper hole in the main spindle runs out of true. In such a case it is necessary to have a "witness mark" (perhaps a center punch mark) on both the spindle and center, and bring these two in line when putting in the live center.

The dead center is hardened and if damaged, must be ground. It should be smooth. The dead center is a bearing on which the work revolves, therefore it must be kept well oiled or it will run dry in the center of the work and become roughened and probably twisted off.

1 Grinding Centers.—There are several styles of grinding attachments that are designed especially for truing centers. If one of these attachments is not available the dead center may be softened (annealed) and trued up, and then rehardened. (For annealing see page 89.)
78. Setting the Tool.—First: Catch the Tool Short. (See Fig. 84.) The further the cutting edge of any tool projects from the tool post the greater the leverage and the more the spring of the tool. This causes chattering and often worse evils and should be avoided wherever possible.

Second: The tool post should be located at the left hand end of the T slot in the tool rest (Fig. 85). If it is clamped in the middle or right hand end of the slot the danger of the dog hitting the tool-rest is greatly increased. If the dog hits the tool-rest when the lathe is running, the point of the live center will be broken off and the center hole in the work spoiled.

Third: The position of the cutting edge as presented to the work has a considerable influence on the finished appearance of the work and also on the life of the tool. It may be easily proved that a cutting edge at right angles to the center line of the work will not cut as efficiently as when arranged at an angle of about 20° with this perpendicular. (See Fig. 86.)

Fourth: The usual practice with modern lathe turning
tools (tool holders with high speed tool bits) is to set the tool point toward the dog (see a, Fig. 87) thus obtaining approximately the proper side rake with less grinding on the top

![Diagrams](image)

Most efficient turning tool. Does not remain sharp as long or produce as good work as (a). Very inefficient except for certain finishing cuts.

Fig. 86.

of the tool bit. This is all right for light cuts if care is taken to tighten securely the tool in the tool post. For heavy cuts, however, it is best, if possible, to have the tool point a little away from the dog for if it slips it will move away from the work and not into it. (See b, Fig. 87.)

Fifth: Have the point of the tool on center or a trifle above.

79. Direction of the Feed.—In machining a piece of work on centers the feed should be toward the headstock because then the pressure is on the live center, which revolves with the work. The dead center is a bearing on which the work revolves and if a heavy feed is directed against it, undue friction will result which will tend to score the center hole in the work and may possibly twist off the end of the lathe center.
80. Oiling the Centers.—It is very important to keep sufficient oil (or white lead) on the dead center. The heat generated in turning, especially during a heavy roughing cut, will quickly burn up the oil on the bearing surface and a dry bearing always cuts. Also the heat of a heavy cut may expand the work enough to unduly tighten it between the centers. It is not enough to squirt some oil at the center: stop the lathe, draw back the center, put the oil where it will do the most good and re-adjust the dead center. A dry bearing will sometimes give warning by a faint squeak. *Always investigate a squeak.*

81. Lubricating the Tool.—There is no doubt that a good flow of oil or cutting compound on a turning tool will make for longer life of the tool, more work, and a better finish, when machining steel or wrought iron. Most engine lathes, however, are not equipped to obtain a flow of oil and the small amount that can be applied with a brush is not usually considered worth while except when cutting threads.

82. Graduations on Cross Feed Screw.—Most lathes are equipped with a graduated bushing on the cross feed screw which will show in thousandths of an inch the movement of the cross slide. For example, the cross feed screw in Fig. 88 has 10 threads per inch; one complete turn of the handle advances the cross slide \( \frac{1}{100} \) ". The bushing is graduated into 100 equal divisions, therefore a movement of one of these divisions past the line on the nut (shown in line with the 0 graduation on the bushing) will indicate a movement of the cross slide of \( \frac{1}{100} \) of \( \frac{1}{100} \) or \( \frac{1}{10000} \) of an inch. Remember that this movement affects the radius of the work and is doubled on the diameter.

Caution.—All lathes are not graduated to read thousandths,
but the numbers on the graduations read thousandths. For example: if there are ten graduated spaces between the numbers 0 and 10 the graduations read thousandths but if there are only five spaces between 0 and 10 the graduations read 0.002”. When going to work on a strange machine note the graduations.

83. Lost Motion in the Cross Feed.—There is always “backlash” or lost motion between any freely revolving screw and the nut. The amount of lost motion depends on the looseness of the thread in the nut and is of course increased by wear. The amount of lost motion in the cross feed screw in a new lathe may be .005” and in an old lathe .020” or more. Suppose the lathe operator runs the cross-slide in .005” too far and then merely moves the handles back .005”; the cross slide has not moved and will not move until the lost motion is taken up. The best way to correct an error of this kind is to move the handle back more than is really necessary, then take up the lost motion in the other direction and feed in again to the proper mark.

METHODS OF TURNING

As an example let it be supposed that a machine steel cylinder is to be finished 6½” long and 1¾” diameter. The stock furnished is 1½” diameter and 6¾” long. Be sure the live center runs true and note that the lathe centers are approximately in line. Center the work carefully (centers about ¾” diameter) and, in this case, face to length. The next operation is rough turning. In order to turn straight it is necessary to have the lathe centers exactly in line.

84. Alignment of Centers.—With the turning tool take a cut (No. 1, Fig. 89) quite near the dog just deep enough to get under the scale and about ¼” wide. Without changing the position of the cross feed, move the dead center away from the
work, swing the piece clear from the turning tool and run\(^1\) the carriage back to take cut No. 2. Put the work back on the centers and take cut No. 2. Caliper both cuts. If they are of the same diameter the lathe is turning straight because the two cuts indicate exactly the same effect as if one cut had been made the whole length to the dog. If the two cuts do not measure alike and cut No. 2 is the larger diameter, the tailstock slide should be moved toward the operator. If cut No. 2 is the smaller the tailstock should be moved away from the operator.

85. Quicker Methods of Aligning Centers.—If a “test piece” (a piece 8” or 10” long, turned round and straight) and an indicator\(^2\) are available the lathe centers may be quickly aligned.

\(^1\)“Run” means to cause to perform a characteristic motion, hence the shop terms “run the tail center back;” “run the carriage back;” “run the tool in,” etc.

\(^2\)Indicator.—An instrument which with multiplying levers plainly shows the slightest movement of the point. See Fig. 91. There are many kinds of indicators ranging in price from a few cents to several dollars. The principle involved in all of them is, that the amount the work is out of true is magnified many times and indicated by a needle, usually on a dial reading thousandths of an inch. An indicator is intended for use in setting centrally any point or hole in a piece of work to be operated upon in a lathe or upon a face plate, also for testing lathe centers, shafting and other work held between centers, the inside and outside diameters of cylinders, pulleys, etc., and work of a similar nature.

It is also very useful when aligning vise jaws or angle irons in shaper, planer or milling machine work or in aligning a finished surface preparatory to taking cuts which are to be parallel or at right angles to this surface. Other types of indicators are illustrated on page 209.
Place the test piece on centers without a dog and the indicator in the tool post. Run the indicator along the length of the test piece (Fig. 90) and when no movement of the indicator needle is observed the centers are in line. If no indicator is available, lightly pinch a piece of paper between the butt end of a tool holder and the test piece near one end and then near the other end and note the graduations on the cross feed screw for each position. The reading of the graduations will indicate whether or not the centers are in line.

86. **Setting the Speed.**—The next step is to get the proper number of revolutions of the spindle to give a correct cutting speed (in this case, with a high speed tool, say about 60 feet per minute).

The formula is: \( \text{r.p.m.} = \frac{\text{Cutting Speed}}{\text{Dia. of work} \times .25} \)

Substituting values and solving: \( \text{r.p.m.} = \frac{60}{1.5 \times .25} = 160 \)

Then fix the spindle speed as near 160 r.p.m. as possible.

87. **The Roughing Cut.**—An accurate job is required so it will be necessary to take two cuts, a roughing cut and a finishing cut. The diameter is to be reduced \( \frac{1}{8}'' \) (stock \( 1\frac{1}{2}'' \) to finish \( 1\frac{3}{8}'' \)) that is, the roughing chip is less than \( \frac{1}{16}'' \) so a coarse feed may be used for roughing without undue strain on tool or work.

With the tool post at the left-hand side of the tool rest set the tool well back in the tool post. For roughing, it usually is set square or pointing a little to the right and about \( \frac{1}{64}'' \) or \( \frac{1}{52}'' \) above center.

Tighten the dog securely on one end of the work, put a drop of oil in the opposite center, and adjust between lathe centers, (no slack and not too tight). Leave \( \frac{1}{52}'' \) for finishing. This means \( \frac{1}{52}'' \) on the diameter or \( \frac{1}{64}'' \) chip for the finishing cut.

Being careful not to turn undersize, feed by hand for \( \frac{1}{8}'' \) or \( \frac{1}{16}'' \), stop the lathe and caliper the cut. When it measures about \( \frac{1}{52}'' \) oversize, throw in the feed and turn about half the length. Then take out the work and without changing the cross feed run the carriage back to dead center, change the dog and turn the other end to meet the cut already made.
88. The Finishing Cut.—Approximately $\frac{1}{32}$ of an inch is left on the diameter for finishing. The amount to leave for the finishing cut depends largely on the character of the roughing cut (a very coarse roughing feed has a tendency to tear the surface even if a fairly sharp tool is used). At least $\frac{1}{64}$" should be left for the finishing chip ($\frac{1}{32}$" on the diameter) because less than $\frac{1}{64}$" does not give the cutting point a chance to get under the chip and the result is a rubbing or burnishing effect which rapidly dulls the edge and produces a poor finish.

A keen cutting edge is essential for a good finish. If several pieces have been roughed with the tool it may be necessary to re-grind it but if only one or two probably a few rubs with the oil stone is sufficient. Adjust the cross feed until the tool just touches the rough turned surface of the work, then run the tool off the work toward the dead center. Using the graduations on the cross feed run the tool in, but not quite the full amount. Take a cut wide enough to caliper and measure the work again. (Note:—This precaution only takes a moment and is always advisable.) Having noted the extra amount to be removed start the lathe, throw out the feed, run the tool off the work again, move the cross slide in the required amount, and turn half the length of the work. Throw out the feed, then stop the lathe, and, being careful not to touch the cross feed handle, remove the work and run the tool back to the starting point. Change the dog to the finished end and put a protecting piece of brass or copper under the screw. Oil the dead center, adjust the work between centers, and turn to meet the finished cut already made. If the position of the cross slide has not been changed, the ends of the piece will measure the same; if the live center runs true, the two cuts will meet exactly; if the centers are in line the piece will be straight.

89. Turning Duplicate Pieces.—When turning a number of duplicate pieces, set the tool for the first piece, cut the required length, throw out the feed, then stop the lathe, remove the piece, run the carriage back to the starting point and put in the next piece. Measure each piece when it is taken from the lathe to make sure the setting is unchanged.
If the cut is of sufficient length to give the necessary time, put a dog on the next piece and oil the center so that it will be ready to put in the lathe when the other is taken out. "Time is money."

When the first cut is made on all the pieces, take the next operation or cut in the same way—piece by piece, and so on till all are roughed, then take the finishing cuts by the same method of procedure.

These directions apply in general to any lathe operation, and as a matter of fact to most machine shop work involving more than one piece. Machining several pieces without changing the setting of the tool or the adjustment of the machine makes for speed and efficiency.

90. Filing in a Lathe.—While it is true that a very large percentage of the round work in the machine shop is now finish turned in the lathe, or ground to size in a grinding machine, it is yet important that the machinist shall know how to file and polish in the lathe as well as at the bench.

Very little work requires any filing, although it often happens that a few brushes with the file will save considerable time in turning. For example, if a special taper is turned nearly correct, a few strokes of the file will make it right much more quickly than it could be turned, or if a shaft is nearly exact a very little filing will make it correct. Sometimes it may be necessary to file and polish such work as a filleted corner, or rounded edge or end, or some special part of a machine such as a bushing or a handle or a pulley.

Filing in a lathe is accomplished by moving the file slowly on the revolving surface. The file mostly used for lathe work is the mill file.\(^1\) This file is single cut and does not pin as readily as the double cut files.

The mistake is usually made of having the work, especially cast iron work, revolve too fast. A surface speed about double that of turning will usually be found sufficient.

Another mistake that the beginner often makes is to move the file across the work too rapidly, thereby filing it out of

\(^1\) For description of files see page 240.
round. A long slow stroke will always give the best results. The file should not be lifted on the return stroke but the pressure should be relieved.

Another fault is to bear on too hard when filing round work. Remember that the surface of the file at any given time touches only a very small surface of the work and bearing on too hard will tend to file the piece out of round and also tend to pin the file and scratch the surface.

The beginner should be warned against the tendency to bring his left arm in contact with the revolving dog or face plate or chuck. It often happens that in order to file a surface fairly close to the dog or chuck jaws, the operator must assume a most awkward and tiresome position in order to reach over the chuck or face plate with his left arm. It is an excellent idea to learn to file "left handed" in a lathe. It is about as easy to learn to file left handed as right handed and when the knack is acquired, a habit is formed that will make for efficiency.

Keep the file pointed practically straight ahead but the stroke should be from right to left or from left to right, to avoid the tendency to file more from one portion than another. Crossing the strokes in this way will help also to keep the file clean.

A better appearance of the work will result if the file (especially when new) is chalked when used on cast iron and oiled or chalked when used on steel.

Hold the file with the palm of the hand against the end of the handle and the thumb on top. Cover the end of the file with the thumb of the other hand and curl the fingers under.

Many machinists leave too much for filing. With a fairly smooth cut, two thousandths of an inch is enough to leave for filing and polishing.

91. Polishing in a Lathe.—Most polishing in the lathe is done with emery cloth. Emery cloth may be obtained in various sizes of grains from fine to coarse. Usually polishing is done with fairly fine grain of emery cloth and the best finishes are obtained with a piece of emery cloth practically
worn out. A better finish and a more lasting polish may be obtained by applying a reasonable amount of lard oil.

As far as the operation of polishing is concerned, the speed of the work can hardly be too great, but if a large piece is in the lathe and a high speed is wanted, care must be taken to have this work well balanced, and if it is on centers have plenty of oil in the tail center and do not adjust the tail center too tight.

Do not try to file or polish work with grooves or slots or holes in the surface. If it is necessary to polish such a piece in a lathe, plug these holes or grooves with hard wood.

Questions on Turning—I

1. What do you mean by “adjusting” the work between centers?
2. How do you make sure the dog does not bind in the slot of the face plate?
3. Why is it necessary to have the live center running true? Name two methods of truing a live center?
4. Why must both centers be in line? How are the centers aligned approximately? How are they aligned accurately?
5. Why is it necessary to put oil in the dead center hole? Why is the dead center hardened?
6. What occurs if the tool is not caught short?
7. What occurs if the tool is set below the center? If set too far above the center? What is the best way of determining when the tool is set right as regards the center?
8. What occurs if, when turning, the dog hits the tool rest? How does laxity in setting the tool often account for an accident of this kind?
9. How may the proper cutting speed be calculated?
10. What determines the roughing chip? The roughing feed?
11. What determines whether the finishing chip shall be \( \frac{1}{64} \) inch or more than that?
12. Why does a finishing chip of \( \frac{1}{64} \) inch give better results than a finishing chip of .002 inch?
13. How is the roughed diameter measured? When is the measurement made? Why?
14. Explain the method of using, and the value of the graduations on the cross feed screw.
15. If it is advisable to oil the center during the cut, how is it done? Why not simply put oil on the outside?
16. If machine oil is not heavy enough what may be used instead?
17. What determines the proper feed for the finishing cut?
18. How is the finished diameter measured? When is the measurement made? Why?

19. When is a protecting piece under the dog screw necessary? Why is a piece of copper or brass better than sheet iron?

20. Describe in detail the method of turning duplicate pieces, and explain the advantages of this method of procedure.

21. Why is a mill file best for filing work in a lathe?

22. What causes "pinning" in a file?

23. What two mistakes are often made when learning to file in a lathe?

24. When is it practicable to finish round work by filing in a lathe?

SHOULDER TURNING

92. Definitions.—In Fig. 92, a represents a shoulder with a "filleted" corner, b with a "square" corner, c illustrates a shoulder with a "rounded" edge, d with a "chamfered" edge and of course the last two terms apply as well to the ends of a piece as to the edges of a shoulder. To "break" an edge is to touch it lightly with a file or emery cloth and take away the extreme sharpness.

![Fig. 92.](image)

93. Roughing to the Shoulder.—When it is required to turn to a shoulder, first lay off the distance from the end. Use a hermaphrodite caliper, or a scale and scriber and make a clean sharp line. Chalking the spot in which the line is to be scribed will serve to make it more distinct. Rough turn in the usual way, with the regular turning tool, until within \(\frac{1}{8}\) inch or \(\frac{1}{16}\) inch of the line, then throw out the power feed and feed to the line by hand. It is well to grind the turning tool with a fairly small round nose and to set it in such a way as to leave as little stock at the shoulder as possible for the finishing tool.
94. Finishing the Small Diameter and the Shoulder.—The tool shown in Fig. 93 is the best form of tool point to make the finishing cut. It is ground to 90 degrees for a distance of \( \frac{3}{16} \) inch or more as shown, and then a trifle less than 90 degrees for clearance. It may be given a little side rake, and the corner may be slightly rounded with an oil stone. If properly ground and set, the front of this tool makes an excellent finishing tool for turning, and the side a most efficient facing tool. Using this tool, with hand feed, carefully cut out the fillet left in the corner by the roughing tool, to leave the corner fairly square. Then take the finishing cut on the diameter in the usual way, with this tool, taking note of the cross feed setting (graduations); throw out the feed when about \( \frac{3}{16} \) inch away from the shoulder and feed by hand practically to the line, then face the shoulder and split the line. The reason for noting the cross feed setting is this: it may be necessary to take a second facing cut to get the right dimension and it will be easy to run the tool in exactly far enough and not risk under-cutting the diameter. Also, if several pieces have been roughed, the finish cuts can be made quickly after the first setting because it has been determined how far in to run the tool. The smaller filleted corners are turned in the same way as explained above. The finishing tool is substantially the same except that it is rounded to give the required radius.

Many machinists prefer to face a shoulder by feeding in from the circumference. In this case the tool is set to split the line and fed towards the corner. In either case rough practically to the line; do not leave too much for the finishing cut or the corner of the finishing tool will become dull sooner than need be.

If possible a piece should usually be roughed all over before any finish cut is taken. Where the end is turned smaller to
a shoulder it should be roughed first thereby possibly saving
a second roughing cut over this portion of the piece, that is, a
piece need not necessarily be roughed to a straight cylinder
before roughing the shoulder cuts. After roughing, face to
length. The next operation is to finish turn the larger diam-
eters. Then lay off the exact shoulder dimensions on these
finished diameters, finish the small diameters and face the
shoulders to the lines, checking the diameters with a micro-
rometer and the lengths by scale measurement.

95. The Forming Tool.—The larger filleted corners are
best roughed out with a large round nose tool using both hand
feeds simultaneously. An approximately correct gauge cut
from sheet metal may be advisable. The best tool to
use for finishing a large fillet or a rounded corner of al-
most any size over $\frac{3}{8}\$" radius is termed a forming
tool. Forming tools are
often forged and then filed
to the desired shape. Fig.
94 illustrates a forming tool
holder. In principle it resembles the old fashioned "goose-
neck" or "spring tool" in that any tendency of the tool to
spring is away from the work and not into it. In the same
figure are shown cuts of typical forming tools to be used with
the holder. Usually a better finish is produced on steel if
oil is used to lubricate the forming tool. The cutting speed
is fairly slow to avoid chattering.

96. Necking (Fig. 95).—Occasionally it is desirable when
turning shouldered work, especially when roughing, to cut a
groove or "neck" of nearly the correct depth and turn either
to the groove or from the groove depending on which end of
the piece it is most convenient to place the dog.

When a piece is afterward to be ground to a shoulder it is
often advisable to neck it just under the size to be ground (b,
Fig. 95) so that the grinding wheel will not have to be fed close
to the shoulder. If a thread is to be cut to a shoulder it usually is considered better to cut a neck for the thread tool to run into (c, Fig. 95). The necking tool may be round nosed or square nosed, as desired.

Fig. 95.

97. The Center Rest.—The center rest, or steady rest (Fig. 96), is a very valuable accessory to the lathe when slender pieces of any considerable length are to be turned. The base is planed to fit the inside ways of the lathe and by means of a clamp the center rest may be held in the required position. The three jaws are adjustable.

A "spot" slightly wider than the jaws of the steady rest, is very carefully turned near the center of the work. (See Fig. 97.) If the piece is to be turned the whole length and other operations afterward made on one or both ends, the machinist usually prefers the center rest to the follower rest (Paragraph 98) for steadying the work during the first operation of turning the cylinder as well as for the
later operations. In such a case the spot should be turned somewhat nearer the live center than the middle of the piece in order that the first cut may be turned at least half the length of the work. When one-half the length of the piece is turned, reverse the piece and readjust the jaws to the diameter of the turned part of the work. To protect this part from being roughened a thin piece of brass or copper should be placed around the piece under the jaws; use oil to lubricate, and do not adjust the jaws too tight.

![Diagram](image)

**Fig. 97.**

The spot should be turned smooth and preferably of some standard mandrel size. The reason for the particular diameter of the spot is to enable the operator to adjust the jaws on a short mandrel rather than on the piece itself, because a long slender piece springs so easily that it makes a true adjustment on such a piece very difficult.

Do not adjust the jaws without first securely clamping the center rest to the ways, for unless this is done, the adjusting of the jaws is apt to lift it from the ways, and when it is finally clamped down the work is sprung.

A long piece of work placed in a chuck for the purpose of boring or turning may not be held rigidly enough, in which case, the center rest may be used as a support for the outer
end (Fig. 98). A spot for the jaws is first carefully turned concentric with the axis of the work.

If one end of the work is supported by the live center, drilling and boring may be done on the other end when the center rest is used as a support. (See Fig. 99.) In this case it is necessary to tie the work to the face plate (usually with a belt lacing) in order to keep it tight against the live center. To tie the work to the face plate, unscrew the face plate 3 or 4 revolutions and tie the lacing as tight as possible. After thus tying, screw the face plate home; this draws the work securely against the live center.

The cat head (Fig. 100) may be used where it is impracticable to turn a "spot" for the center rest. Great care must be
taken to adjust the cat head before putting it in the center rest so that it will run perfectly true when the work is revolving on centers.

98. The Follower Rest.—The follower rest (Fig. 101) is used to prevent slender work from springing away from the tool during the cut. The diameter is turned for a short distance to the desired size and the two jaws are adjusted to this diameter. As the follower rest is bolted to the carriage and moves with it, the two jaws constantly offer resistance to the spring of the work away from the tool.

99. Knurling.—Knurling may be defined as the process of checking the surface of a piece by rolling depressions into the surface. The knurled portion of a nut or of a screw that is to be adjusted by hand; or of a handle, as for example a tap wrench, gives an excellent gripping surface.

Most knurled jobs done in machine shop work are of the diamond pattern in "coarse," "medium" and "fine" as shown in Fig. 102. The knurls are small wheels with the marking cut in their faces and hardened.

The advantage of the particular kind of knurling tool shown in the figure lies in the fact that the wheels work opposite each other, and thus do not distort the center holes in the work. It is possible with this tool to knurl a piece of small diameter
projecting from a chuck and this cannot be accomplished by a tool in which both knurls tend to push the work away from the tool.

Care must be taken in starting the knurl to see that the one wheel does not split the diamond. The knurls should be pressed hard into the work at the start and the pressure relieved somewhat after making sure they "track." The

Fig. 102.—Knurling tool (Billings and Spencer) and illustrations (full size) of knurled work.

finished job should show as in the illustration with the diamond shape clean and sharp.

Coarse power feed may be used and usually two "cuts" are enough. Oil is used to lubricate when knurling any kind of material.

THE MANDREL

A mandrel (sometimes called "arbor") is a tool which when pressed into a finished hole in a piece of work provides centers on which the piece may be turned or otherwise machined.

Fig. 103.

100. The Standard Mandrel.—Standard hardened and ground lathe mandrels (Fig. 103) are manufactured in various sizes. The mandrel tapers about .008" per foot, and the nominal size is near the middle. It is driven or pressed into the hole in the work and holds by friction. To prevent damage
to the work, the mandrel should always be oiled before being forced into the hole.

The ends are turned somewhat smaller than the body, so that any nicks or burrs, caused by the clamping of the dog, will not injure the accuracy of the mandrel.

It is very important to have the centers large enough to withstand the severe strain that may be caused by turning a heavy piece; that they are exactly 60° to fit the lathe center; and very smooth to ensure a good bearing. The centers are recessed for protection, but even so, a mandrel should never be driven with a steel hammer without protecting the end. It is better to use a Babbitt hammer, or a press (see Fig. 104) made especially for forcing a mandrel in or out. Be sure the mandrel is started straight or it will score one side of the hole. If the hole is properly sized the mandrel should enter a considerable distance before it begins to bind, thus serving to start itself square. It is important to remember that a mandrel is always marked on the large end. To avoid pressure against the arms or the web of a pulley when forcing a mandrel in or out put a collar under the hub; if a collar is not available a pair of short parallel pieces will answer.

101. Using a Mandrel.—Turning and facing operations on work held on a mandrel are not different from other turning and facing operations, and require the same conditions of tool grinding and setting. When turning work on a mandrel feed toward the large end of the mandrel, if convenient.
tends to tighten the work on the mandrel. When turning comparatively large work on a small mandrel care must be taken that the mandrel does not spring or bend and also that the work does not turn on the mandrel. It is often advisable in such a case to drive the work directly from the face plate instead of by means of a dog. (See Fig. 105.)

Always remember, that even if the mandrel is hard, the cutting tool is harder and it is easy, through carelessness, to spoil the mandrel. Do not put your trade mark on the mandrel.

102. Other Forms of Mandrels. Home Made Mandrels.—A machine steel mandrel with case-hardened ends is often found efficient; or if it is a rush job a piece of machine steel carefully turned (on good sized centers) to fit the hole will answer. On large sizes a satisfactory mandrel may be made of cast iron by inserting in the ends hardened tool steel plugs with centers.

Expansion Mandrels of various types are manufactured. These mandrels are expensive and are not as accurate as the solid ground mandrel. Taper Mandrels with cast iron expansion bushings a, Fig. 106 are extensively used, especially in the larger sizes. Do not try to force the bushing to bind in a hole that is oversize or it is likely to ruin the bushing.
Another form of expansion mandrel is shown in (b) Fig. 106. It consists of the mandrel proper which has four or more grooves cut length-wise uniformly deeper toward one end; a sleeve with slots opposite the grooves in the mandrel and of the same width; and the jaws which fit into the grooves of the mandrel and through the slots in the sleeve. The jaws taper in length to correspond to the incline of the slots and therefore bear evenly in a straight hole. The take-up or release is made by sliding the mandrel in the sleeve. The particular value of this mandrel lies in the fact that it may be adjusted to bind in a hole somewhat larger or smaller than its nominal size.

A Gang Mandrel (Fig. 107) is useful for turning or milling several pieces such as gear blanks or cutter blanks at the same time. It is especially useful when turning thin pieces.

The Threaded Mandrel or "Nut Arbor" (Fig. 108) is used for facing nuts or otherwise machining inside threaded pieces. Unless one face of the work is square with the axis of the thread an equalizing washer $W$ is necessary to make sure that the nut takes up true on the thread of the mandrel and is not canted. Such a washer is quickly made from an ordinary iron washer by bending it slightly.

The Taper Shank Mandrel (Fig. 109) may be fitted to the
taper hole in the spindle of the machine. The projecting portion may be of the form desired. A nut mandrel made in this way is used for machining "blind nuts," that is, a nut in which the hole does not go through.

103. To Turn a Crank Shaft or an Eccentric.—When a cylindrically turned surface of a piece has an axis parallel to, but not coincident with the normal axis of the piece, this surface is said to be turned eccentric. The work itself may be called an "eccentric" a or a "crank shaft" b and c, Fig. 110. Both are much used in machine construction to convert rotary
into reciprocal motion or the contrary. To turn the eccentric surface it is necessary to provide centers off-set from the centers of the normal axis an amount equal to one half the "throw" desired, that is, one half the amount of the reciprocating motion to be imparted or converted as the case may be. Three methods of providing the off-set centers for turning eccentric cylindrical surfaces are illustrated in Fig. 110. An eccentric \( a \) is usually keyed to a shaft when in use and consequently is located on a mandrel, by a key or by a witness mark and then turned. A mandrel with off-set centers is shown in \( a \). The crank shaft \( b \) is itself provided with an extra pair of centers. When the amount of eccentricity is too great to allow the extra pair of centers within the diameter of the normal bearing, special pieces with centers properly arranged may be fitted on the ends of the shaft as shown in \( c \). A suitable counterweight should be provided, and also braces to eliminate spring.

Questions on Turning—II

1. If considerable metal is to be removed, is it advisable to face to exact length? Give reason.
2. Why is it usually advisable to lay off the shoulder distance from one end of the work?
3. What tools may be used to lay off these distances? When is chalk used? When is blue vitrol used?
4. Is care used in laying off shoulder distances when roughing? Why?
5. If you are going to rough a square shoulder, how does the shape of the tool differ from a regular turning tool? How is the tool set? Why?
6. If one or more diameters are to be turned on a piece, is it necessary to rough to straight cylindrical shape before roughing the smaller diameters?
7. When roughing out how much stock do you try to leave for finish facing the shoulder? Why?
8. Why is it good practice to throw out the power feed when \( \frac{1}{8} \) inch or more from the shoulder, and feed the rest by hand?
9. When turning how much do you leave for a finishing chip?
10. How do you grind a combination turning and facing tool for finish turning the diameter and finish facing the shoulder? Is such a tool efficient? Why? Explain its action in detail.
11. Explain how the graduations on the cross feed screw may be of great help in turning shouldered pieces.
12. Sometimes when turning shoulders it is convenient to neck the work. How is this done?
13. If the work, when faced, is left somewhat over length because of the amount of stock to be turned off, when should it be faced to length? Why at this time?
14. What is meant by a square corner when turning a shoulder? A filleted corner?
15. What is a square edge? A rounded edge? A chamfered edge?
16. Why must care be taken when starting the knurling tool?
17. How do you tell when a piece is sufficiently knurled?
18. Why are two wheels used to produce the diamond pattern knurl?
19. What is meant by turning a “spot” for the steady rest?
20. Why is it advisable to make the spot a nominal size if convenient?
21. Why not turn the spot in the middle of the piece?
22. When is it necessary to tie the dog to the face plate? How is it done?
23. What is the difference between a steady rest and a follower rest?
24. Why is a mandrel tapered slightly?
25. Why is not a mandrel marked on both ends?
26. How is a mandrel started and how is it pressed in the hole?
27. What precaution must always be taken before forcing a mandrel in a hole? Why?
28. Name three kinds of mandrels and explain the particular value of each.
29. What is a “nut arbor”? What do you understand to be the value of the bent washer on the nut arbor?
30. What do you mean by a blind nut? Can you find a blind nut holding any part of the lathe?
31. How may a blind nut be held while being faced or turned?
32. A special bushing 3” long has a hole .990” in diameter. What kind of a mandrel, if available, could be used?
33. If no mandrel for a hole .990” in diameter were available, what would you do?
34. Why should the center holes in a mandrel be clean cut, smooth and fairly large?
35. Why are the ends of a mandrel recessed around the centers?
36. Explain how work of fairly large diameter, mounted on a mandrel, may often be driven direct from the face plate. What is the advantage of this method of driving?
37. What is an eccentric? What is an eccentric strap?
38. What do you understand as the difference between an eccentric and a crank shaft?
39. What do you mean by turning eccentrically?
40. Using one or more V blocks and a surface gauge how would you proceed to lay out the positions of the two pairs of centers in b Fig. 110?
CHAPTER VIII

CHUCKING WORK

Engine lathe work may be divided into three general classes; work on centers, chucking work, and face plate work. Each of these classes of work may involve one or all of the operations of turning, threading, boring, etc. Certain operations relative to work on centers have been explained and chucking work will now be considered.

104. Kinds of Chucks.—The larger chucks (Fig. 111), those that will hold a piece of a diameter of 2" or over, are usually mounted on a chuck plate that screws on the end of the machine spindle. In lathe work they are called "lathe chucks."

![Fig. 111.—Lathe chuck and drill chuck.]

The smaller chucks are provided with a taper shank arbor which fits the taper hole in the end of the spindle. (See Fig. 111.) Because their chief function is that of holding small drills, etc., these chucks are generally called "drill chucks."

If the jaws move independently of each other the chuck is called an "independent chuck;" if they all move together, it is called a "universal chuck." A chuck that may be ar-
ranged as either independent or universal is called a "combination chuck." Practically all of the larger chucks are independent or combination while the small lathe chucks and the drill chucks are universal. Most lathe chucks have reversible jaws. In some chucks the jaw is removed and reversed and in other designs (see Fig. 111), a part of the jaw is reversed. The object of the stepped reversible jaw is to enable pieces of various diameters to be held. The work may be gripped either inside or outside as desired. (See Fig. 112.)

The spring chucks (Fig. 113) are very handy for small round work. The adapter fits the taper hole in the main spindle and the chuck fits into the adapter. The hollow draw-in spindle goes through the hole in the lathe spindle, and the
end, being threaded inside, screws over the threaded portion of the chuck and draws the chuck back into the adapter until it grips the work. This form of chuck is often called a spring collet or a draw-in chuck.

105. Mounting the Chuck on the Spindle.—Remove the live center and if short work is to be machined put waste in the center hole to keep out chips and dirt. Remove the lathe face plate and place it where chips and dirt will not get in the threaded hole. Be sure the thread on the spindle, and also the shoulder of the spindle, are perfectly clean and free from any burr; likewise the chuck plate. Oil the thread thoroughly. A chuck 10″ diameter or over should be brought in line with the spindle by placing it on a suitable block laid across the ways of the lathe. Start it square on the thread, turning the spindle by hand. (The same care should be taken when removing the chuck.) The chuck should screw on the spindle easily. It should be screwed tight against the shoulder by hand but must not be brought up with a bang.

106. Adjusting the Work in a Chuck.—Adjust the jaws to about the size of the piece to be held and approximately central with the concentric grooves turned in the face of the chuck. Tighten the jaws on the work, and turn the chuck around by hand to be sure it will revolve without trouble; next start by power and mark the “high spot” on the work with chalk as the work revolves. Then loosen the jaw furthest from the high spot, tighten the opposite jaw, thus pushing the work to the center.

Another way of trueing the work is to use the butt end of the tool to gauge the eccentricity, and then adjust the work the amount it is off center.

Tighten all screws evenly after the piece runs to suit. Mark two adjacent jaws, and for subsequent pieces of the same size

1 Removing the Face-plate or Chuck.—Put in the back gears, and turning the lathe backwards by hand, allow the edge of the slot in the plate to bring up against a substantial block of wood resting on the back ways. A chuck is removed in the same way by bringing one of the jaws against the block.
to be handled, use only these two jaws. If a three jaw chuck is used, one jaw only need be loosened to release the piece. This will save time.

If it is required to have the work "trued up by" a finished surface (outside, inside or faced surface) an indicator may be used to obtain a fairly accurate result. It is, however, practically impossible to make it run exactly true. For this reason, if part of an operation is finished, do not remove the work from the chuck until the whole operation is finished.

107. **Radial Facing.**—The term Radial Facing may be applied to the truing of the faces of work of a comparatively large diameter when held in a chuck or on a face plate. (See Fig. 114.) The Shovel-Nose Tool (see page 60) may be used for the roughing cut. When using this sort of tool or a similar tool, face from the circumference of the work toward the center. When finishing, an ordinary turning tool, or side tool, whichever is preferred, may be used. In radial facing care must be taken that there is little or no end motion in the main spindle bearings. Also, to prevent the tool working away from the cut, and thus producing a surface which is not flat and true, it is necessary to tighten the carriage clamping screw. A much better finished appearance may be obtained on cast iron if the chip is very light and the feed is very coarse. A square-nosed tool makes an excellent cast iron finishing tool. It is good practice when facing steel in a lathe to apply a suitable amount of lard oil or other cutting compound with a brush. Cast iron and brass are machined dry.

**CUTTING OFF BAR STOCK IN A LATHE**

108. **The Cutting-off Tool.**—Select an off-set cutting-off tool (sometimes called a parting tool) with the blade only a trifle longer than half the diameter of the stock.
The cutting edge of a cutting-off tool (Fig. 115) is the widest part of the blade. The front clearance is the same as for any turning tool, about 10°. Besides the front clearance it has clearance on both sides, toward the bottom, and also toward the body. The width of the cutting edge depends on various conditions but should always be wide enough to permit of the blade being strong, and not so wide as to unnecessarily waste the stock. A cutting-off tool never has side rake and seldom if ever any front rake. The reason for no front rake is the decided tendency for the tool to "hook into the work" due to the slack in the cross feed screw. To sharpen a cutting-off tool, grind it on the front.

Fig. 115.—The cutting-off tool. The amount of clearance at (a) and also at (b) is very small, just enough to clear the sides of the groove being cut. Excessive clearance weakens the tool. One particular advantage of the patent cutting-off tool holder and blade is that the blade may be adjusted to project only the distance necessary.

109. The Cutting-off Operation.—Set the tool on center; if too high it will "ride" as it approaches the small diameter, if too low it will "dig." If the work is steel or wrought metal, use plenty of lard oil or some good cutting compound, and provide means of catching the surplus thus keeping the lathe clean. Feed by hand and be sensitive to the cutting conditions. The lathe, especially if motor driven, may be speeded faster as the diameter is decreased. It requires experience to do a good job in grinding, setting and using a cutting-off tool. "Practice makes perfect." When cutting off stock held in a chuck it is best to use an off-set tool to permit of working close to the chuck (Fig. 116).
It should be emphasized here that spring of the work or of the tool is always a decided disadvantage. When using the cutting-off tool particular care should be taken that the work and the tool both be as rigid as the nature of the job will permit.

Do not attempt to cut in two a piece of work held between centers, do not attempt to even neck the piece if it is slender, because it will almost certainly be bent and ruined and the tool will be broken (Fig. 117).

Fig. 116.  
Fig. 117.—A sure way of spoiling the work and breaking the tool.

110. Chattering.—The rapid vibration of the tool and the work which is called chattering frequently takes place when using a cutting-off tool and may be due to one or more of several reasons; a tendency of the tool or work to spring; to the fact that the tool is set too high; to the looseness of the cross slide; or to the looseness of the lathe spindle in its bearings.

Questions on the Use of Lathe Chucks

1. How is the chuck started on the thread? If it is a heavy chuck how is it best held when starting?
2. What is the danger of having the spindle revolve by power when starting the chuck on the thread? When screwing it home?
3. Why should not the chuck be screwed against the shoulder with a bang? Why should it be screwed up tight?
4. To make the work run more nearly true, shall you adjust two opposite jaws to push the high spot toward the center or away from the center? Why?
5. If the high spot should come between two jaws, how would these jaws be adjusted?
6. What is the purpose of the rings in the face of the chuck?
7. If you had several pieces of the same size to machine in the chuck explain how you could save time by marking two jaws of the chuck.
8. Find as many of the following chucks as are available: 6 inch 3 jaw universal; 6 inch 2 jaw universal; 8 inch 4 jaw independent; and a combination chuck.
9. What is the advantage of the universal chuck?
10. What advantages has the independent chuck?
11. How is the combination chuck changed from independent to universal? How are the jaws adjusted, to make them true, before the change is made?
12. There is a proper method of removing a chuck from the lathe spindle. How is it started? How is it held when being unscrewed? What is the danger of starting the lathe by power?
13. What type of chuck jaw is used to hold bar stock or pieces of small diameter?
14. How would you hold in a 6-inch chuck a piece 6 inches in diameter to bore it? How would you hold it to turn off the circumference?
15. What is meant by reversible jaws?
16. What is meant by jaws with reversible tops?
17. How do you size up the workman who hammers the chuck wrench or who uses a pipe extension to the wrench?
18. What is the danger of stopping the chuck by clapping the hand on it?
19. What is the small chuck with a taper arbor which fits into the tail spindle usually called?
20. What are two advantages of the split chuck (spring collet)?
21. Explain in detail the action of the draw-in sleeve in closing the split chuck.
22. What is the advantage of an off-set cutting-off tool in chuck work?
23. How many clearance angles has a cutting-off tool?
24. When is a cutting-off tool set properly?
25. What is the objection to holding work between centers to cut in two?
26. What is meant by chattering? How may it be prevented?
27. When radial facing, what tool may be used for roughing?
28. What is the object of clamping the carriage when radial facing?

DRILLING AND REAMING IN A LATHE

While the drill-press is essentially the machine for drilling and reaming, it often happens that it is more profitable to finish the holes in a lathe.
The twist drills and reamers used in lathe work are the same as those used in drill-press work. The flat drill, while considered old fashioned by some is however a very efficient tool for drilling in a lathe.

111. Flat Drill.—Fig. 118 illustrates the flat drill, and the "holder" which is clamped in the tool post. The two cutting edges of the drill (called the lips) are at an angle of 120° to each other, and are of equal length, which keeps the point central and the strain equalized thereby drilling a hole approximately the size of the drill. Note the clearance angle of the lip, also the rake angle. The cutting angle of the lips is about 60° which is correct for metal cutting.

112. The Twist Drill.—The twist drill (Fig. 119) has almost entirely superseded the flat drill. The spiral flute forms the rake angle and gives a cutting angle of 60°. Examine a twist drill and note that after the flutes are cut the remaining portion of the surface of the body is backed off or relieved and only a very narrow "land" is left. This body clearance reduces the friction of the drill in the hole.
Twist drills are made in number sizes; No. 1 (.228" dia.) to No. 80 (.0135" dia.). They are made in letter sizes A; (.234" dia.) to Z (.413" dia.). (See Table 23). They are also made in sizes ranging by 64ths of an inch from \(\frac{1}{64}\)" to 2" or more, and in metric sizes, ranging from .4 mm. (.0157") to 50 mm. (1.068") by .1 of a millimeter (about .004") on the smaller sizes, and by .5 millimeter on the larger sizes. The smaller drills are not marked and the size is found by the use of a drill gage, Fig. 120.

**Straight and Taper Shanks.**—Drills are made with either straight shanks or taper shanks (Morse Tapers). It is not usually considered economical to buy taper shank drills of a size smaller than \(\frac{3}{8}\)" dia. because of the added cost of the shank. Small drills are better held in a chuck. In the larger sizes (over \(\frac{1}{2}\)") the difference in price between straight and taper shanks is not noticeable and it is nearly always more convenient to hold them by means of the taper shank.

113. Sharpening a Drill.—The clearance angle on a drill is about 12° at the cutting edge (see Fig. 121). If correctly sharpened, the edge of the angle across the web of the drill (the cutting point of the drill) will be about 45° with the line
of the cutting edges (see a, Fig. 121). The appearance of the cutting point is therefore an index to the clearance; when it is like "b," the lip has too much clearance and when it is like "c" the lip has no clearance. It is very important that a drill should have sufficient lip clearance as it takes considerable pressure to feed the drill into the work under the best possible conditions, owing to the nature of the point, and if the lips are not properly backed off the drill will break under feeding pressure simply because it cannot cut.

Extreme care must be taken to get the lips exactly the same length and both at the same angle with the axis of the drill or the hole will be oversize. This is illustrated in Fig. 122.

Theoretically if the drill is ground at an angle of 59° with the axis, a straight lip will be the result. Grinding the drill at any other angle with the axis results in a slightly curved lip. Therefore the drill should be held at about 59° or 60° with the face of the grinding wheel as shown (Fig. 123). The observant student will soon learn to notice any inequality in the lengths of the lips or in their angles with the axis of the drill.
When sharpening the drill place the left hand on the rest, hold as shown in Fig. 123 with the right hand slightly lower than the left hand and the cutting edge of the drill up and in a horizontal position. As the right hand is lowered the drill fulcums in the left hand, and with the necessary pressure against the wheel the required clearance is obtained.

To strengthen a drill the web is made thicker toward the shank. This is not noticeable on drills under $\frac{3}{4}$" diameter but on larger sizes, as the drill is shortened, it becomes necessary to grind the point somewhat thinner. Care should be taken to grind an equal amount from each side.

For drilling brass, the lips of the drill should have no rake, and a straight fluted "Farmer" drill, Fig. 124, is the best to use, but a twist drill may be used if the front of the lips are ground as shown in Fig. 125. It is also advisable to use a drill without rake when drilling very thin pieces owing to the tendency of the drill to "hook into" the work when it is breaking through.

It often happens through ignorance or carelessness that a drill is used after it becomes dull, which causes the land to become worn away for a distance back from the cutting edge. The diameter of the end of the drill is reduced, thus making the drill bind and squeak. It will be necessary to grind off the undersize part and then sharpen the drill. Always examine a drill before using it.

**Drill Grinding Machine** (Fig. 126).—In shops where any considerable amount of drilling is done it is economical to have a drill grinding machine. This machine may be quickly adjusted to support a drill of any length or diameter in a wide
range of sizes and is so designed that it is a very simple matter to grind the drill properly, that is, with lips of equal length, at the correct angle with the axis, and with the correct clearance.

![Image](image_url)

FIG. 126.

114. Speeds and Feeds of Twist Drills—Cutting Compounds.—Owing to the variations of the hardness and toughness of the materials used in machine shop practice, no hard and fast rule can be given for the speeds and feeds of twist drills.

The correct speeds and feeds must be determined by the judgment of the operator, and the following hints will help the beginner to obtain this necessary knowledge.

When the lip breaks off the feed is too heavy, or the drill has been given too much clearance as shown in b, Fig. 121.

When the drill splits, there is too much feed or the drill has not been given enough clearance. There seems to be a tendency for the beginner to give insufficient lip clearance toward the center of the drill. The whole length of the lip must be backed off or the drill will surely break under the feeding pressure.

The rapid dulling of the drill especially at the outer ends of the lips (the corners) is evidence of too much speed.

When a drill squeaks it is usually an indication of a crooked hole or dullness. Never allow a drill to squeak.

The following tables of feeds and speeds are here given as having proved practical for average conditions:
Feeds for Drills (Average).—
.004" per revolution for drill \( \frac{1}{4}" \) or smaller to .015" per revolution for drills \( 1\frac{1}{2}" \) and larger.
Same feeds for high speed drills.

*Speeds of Carbon Steel Drills (Average).*—
20'–25' per minute for annealed carbon steel.
30'–35' per minute for soft steel or cast iron.
60'–100' per minute for brass.
These speeds may be doubled or more for high speed drills.

Cutting Compounds used in Drilling.—
Unannealed Steel.—Turpentine.
Carbon Steel.—Lard oil or a reliable cutting compound.
Soft Steel or Wrought Iron.—Lard oil or soda water.
Malleable Iron.—Soda water.
Brass.—Dry, or a flood of paraffine oil.
Aluminum, Copper and other Soft Alloys.—Kerosene.
Cast Iron.—Dry, never use any cutting compound when drilling cast iron.

Questions on Drills and Drill Grinding

1. How is the drill held when grinding by hand, is it placed on the hand rest, or held in the left hand? Why?
2. How is the drill grasped with the right hand?
3. What angle does the center line of the drill as properly held make with the face of the grinding wheel? Why not 45 degrees? Why not 30 degrees?
4. Why is the drill held with the cutting edge up and in a horizontal position?
5. How is the drill moved against the wheel to "back off" the cutting edge? Why not give it a twisting motion?
6. What do you mean by fulcruming the drill in the left hand?
7. Why must care be taken to have plenty of water available when grinding a drill?
8. If a properly ground drill is held perpendicular to a flat surface, what angle will the cutting edge make with the surface?
9. How much clearance has the cutting edge of the drill?
10. What part of the twist drill is the lip? The point? The land?
11. What is the effect of too much lip clearance? Of not enough lip clearance?
12. How can you tell by looking at the point of a drill whether or not the drill has been given sufficient lip clearance?
13. Has the drill any other clearance?
14. What is “rake” on any cutting tool?
15. What governs the amount of rake angle on a twist drill?
16. Why can not a set rule be given for the speeds and feeds of drills?
17. What does a squeak indicate in drilling?
18. What do you mean by the land of the drill being worn away? What causes this?
19. How many r.p.m. should a ¾” drill be run to give a cutting speed of 35 feet per minute?

115. The Reamer.—It is practically impossible to drill a hole to the exact size of the drill. Therefore to obtain a hole of standard size, round and smooth, it is practical to drill or bore to ½” undersize and then machine ream, and if extreme accuracy is required, it may be hand reamed.

Chucking or Machine Reamers (Fig. 127).—Machine reamers are largely used in drill presses, lathes and similar machines.

There are two types of machine reamers, rose reamers and fluted reamers. In the rose reamer, the teeth are beveled on the end and “backed-off;” they cut only on the end. The lands\(^1\) are nearly as wide as the grooves and are not relieved (backed-off). The flutes or grooves are provided for conveying oil to the cut and chips away from the cut. The rose reamer tapers slightly smaller toward the shank (about .001”) to prevent binding; it does not cut a particularly smooth hole but is very useful to bring the hole to within a few thousandths of size when it may be finished with the hand reamer. Rose

\(^1\) *Land.*—In reamers, milling cutters, etc., the width of the top of the tooth is called the land.
reamers, therefore, are usually made .003" to .005" under nominal size.

The fluted reamer has more teeth for a given diameter than the rose reamer. The lands are narrower, and are backed-off the whole length. The front ends of the teeth are beveled or rounded and then relieved. It is a valuable finishing reamer when extreme accuracy is not required.

Both the rose reamer and the fluted reamer are made with either straight or taper shanks. It is not usually advisable on account of the extra cost to buy taper shank reamers under 3/16" diameter; and it is not usually good practice to buy straight shank reamers of over 1" in diameter on account of the difficulty in holding them.

![Fig. 128.—Hand reamer.](image)

**Hand Reamers** (Fig. 128).—Where a particularly accurate hole is required it is first drilled or bored or machine reamed to .002" or .003" undersize and then hand reamed.

A hand reamer is essentially a finishing tool, a scraping tool. It is ground straight for nearly the whole length of the teeth, being slightly tapered, smaller toward the end, for a distance about equal to its diameter, to permit of its entering the hole to be reamed. The teeth are relieved a very little for clearance. The shank end is machined square to receive the wrench. The hand reamer should never be operated by mechanical power. Care should be exercised to start it true and keep it straight. It is often advisable to start the hand reamer when aligned and steadied by the dead center of the lathe or a center placed in the drill press spindle as the case may be. Do not leave over .005" for a hand reamer.

**Shell Reamers.**—For reasons of economy, many manufacturers prefer the shell reamers and arbors illustrated in Fig. 129. These reamers are made in either rose reamer style
or fluted reamer style and the arbors with either straight or taper shanks, and differ in no particular respect from the ordinary solid reamer except that one arbor may be fitted to a number of reamers and when a reamer is worn out it may be thrown away without discarding the arbor, making for economy in the end.

![Fig. 129.—Shell reamer and arbor.](image)

**Adjustable Reamers** (Fig. 130).—Probably the most efficient kind of reamer for any purpose is the adjustable blade reamer. The best types of these reamers can be adjusted to sizes within a considerable range over or under nominal size; often a valuable feature. While their first cost is considerably in excess of the solid type of reamer, the fact that they may be easily sharpened and quickly adjusted to an exact size, and their corresponding long life, makes them a particularly efficient tool. These reamers are made in all standard sizes, either hand or machine, with the body and shank in one piece or of the shell reamer variety.

**Taper Reamers** (Fig. 131).—Taper reamers, both for roughing and finishing, are made for all of the standard sizes of tapers. The end of the shank is cut square to receive the wrench and the reamer should always be turned by hand. As the chips do not fall out readily a taper reamer should be removed often and cleaned.

**Unequal Spacing of Teeth.**—To prevent any tendency to chatter, the teeth of all reamers should be *increment cut*, that
is, unequally spaced. This may be accomplished by cutting each successive tooth .004"-.007" deeper until half of the teeth are cut, then, starting at the original setting for the next tooth, cut the rest of the teeth successively deeper.

**Cautions.**—It should be emphasized that, never, under any circumstances should a reamer of any kind be turned backward.

![Taper reamers](image)

**Fig. 131.**—Taper reamers. (a) Finishing. (b) Roughing.

A burr on the tooth of a reamer will spoil the hole. When a reamer is obtained from the tool room, feel along the cutting edge of each tooth and if any burr is noticed, oil-stone it off.

**116. Drilling in a Lathe—Spotting the Center.**—Suppose it is required to finish a hole, say 1" diameter, in a piece of solid metal. The first operation usually, after trueing the work in the chuck, is to face the piece, especially if it is cast iron, and the next operation is spotting a center for the drill, using the offset spotting tool. (See Fig. 132.)

This tool is ground to an angle of about 120° to correspond with the angle formed by the lips of the drill. Note that each lip is given clearance but in opposite directions, because the cutting force is up on one lip and down on the other. Set the spotting tool "on center" and "square" that is, in such a position that both lips will cut evenly. Start the lathe and run the tool up to the work. If the point of the tool is not exactly in the center a small ring will be turned in the face of the work. It is very easy to adjust
the point to the center of this ring. Make the spot nearly as large as the diameter of the drill to be used.

A tool bit may be ground and used for a spotting tool if it is given sufficient side clearance (Fig. 133). Care must be taken or the point, being very delicate, will break.

117. Operation of Drilling the Hole.—Select a drill somewhat undersize (\(\frac{3}{64}\)" or \(\frac{1}{8}\)"), to allow for reaming, and a drill holder (Fig. 134) to fit the taper shank of the drill. Be sure that both the taper shank of the drill and the taper hole in the holder are clean and free from oil. Tapers will not hold if oily, and if the taper does not help hold the drill from turning, the tang of the drill may be twisted off under the pressure of the cut.

It is important before starting the drill to note that the tail center is not off-set.

Place the point of the drill in the "spot" and the center of the drill holder on the tail center of the lathe (Fig. 135). Have the spindle well back in the tail stock, and the tail stock tightly clamped to the bed. Upon revolving the work the drill may be fed into the work by turning the tail-stock hand wheel.

As the drill "breaks through" the inside end of the piece, it has a tendency to pull away from the dead center, due to the spiral and to the lack of resistance. Do not try to hold the drill back against the center by hand because it is dangerous. Many drills have been broken and many hands have been severely injured by ignorance or carelessness in this respect.

Note.—If the drill has a straight shank a dog may be used to keep it from turning, provided the drill has a good center hole. Place a protecting piece of brass or copper under the dog screw.
There will be no trouble if a tool clamped in the tool post is arranged against the handle of the drill holder so that the drill cannot pull away from the tail stock unless it pulls the carriage along. This set-up is shown in Fig. 135. Do not wait until the drill starts to break through but arrange as above when making the set-up. While the carriage moves easily enough by the screw pressure against the drill holder and does not make the feeding of the drill noticeably harder, it still offers resistance enough to keep the drill holder against the center when the drill has a tendency to dig. A further caution may here be emphasized: *Never loosen the tail stock, or withdraw the dead center from the drill while the lathe is running.*

After the hole has been drilled, *stop the lathe* and keeping the drill holder against the dead center, either run the tail spindle back or loosen the tail stock and pull it back until the drill can be removed.

The smaller sized holes may be drilled by holding the drill in a drill chuck having a shank which fits the tail spindle.

In lathe drilling the speed is nearly always too slow, probably due to the fact that the chuck being so much larger than
the drill seems to be revolving fast enough. Count the r.p.m. and make sure of the correct speed.

118. **Machine Reaming.**—Reaming is the next operation. The machine reamer is held in a drill holder or by a dog, and should be held against any tendency to pull away from the center as explained in paragraph 117, not only when the reamer breaks through but during the whole length of the cut. This precaution is absolutely necessary or the reamer will catch and bend or break and the hole will be spoiled.

The speed for reaming is usually somewhat slower than for drilling especially in cast iron, to avoid any tendency to overheat and ruin the reamer. The feed should not be crowded or the reamer is likely to tear the surface of the hole.

Place the reamer in position with the end in the hole, get everything ready, then start the lathe and start to feed immediately by turning the hand wheel. Ream cast iron dry, except when sometimes a little oil may be rubbed on the lands of a rose reamer to keep it from scoring. Always use a lubricant when reaming wrought metals or steel.

119. **Hand Reaming.**—If for any reason it is necessary to finish the hole with a hand reamer, it is usually better to start the reamer while the work is in the lathe. Keep the dead center against the reamer and turn the reamer by hand with a wrench. After the reamer is well started the work may be removed and the hole finished in a vise.

It is especially important to know that the dead center is in line when starting a reamer in a lathe as above. If the dead center is even slightly off-set the reamer will not start true.

**Questions on Drilling and Reaming**

1. If the point of the centering tool is not exactly on center, what kind of a cut will be made in the face of the work?
2. How may the center then be easily found?
3. What shape is the end of the spotting tool? Why?
4. How is clearance on the cutting edge of the spotting tool nearer the operator ground? How is it ground on the other edge? Why?
5. How large a spot should be made?
6. If the work is a rough casting or any piece that is not fairly square, why should it be faced before drilling?
7. Can the spotting tool be used to clean a portion of the face around the spot?
8. How may a tool bit be ground to produce a satisfactory spot?
9. What is the effect in drilling if the tail center is off-set?
10. How do you feed the drill?
11. Should the cutting speed be the same as if the drill revolved?

Why?
12. What is the number of r.p.m. necessary to give 30 feet per minute cutting speed for a 1-inch diameter drill? For ½-inch diameter drill? For 1½-inch diameter drill?
13. How do you judge the proper feed?
14. As the drill breams through at the end of the hole the tendency is for it to draw in or “dig in.” This will pull the drill holder off the center and probably break the drill. How is this prevented?
15. If it is a fairly deep hole in steel, how is cutting compound applied?
16. What does a squeak indicate?
17. If a straight shank drill is used, how may it be held with a dog? What about the center in the drill? How do you keep the drill from becoming scored by the dog?
18. How is the dog arranged on the drill? How is it arranged to keep the drill from drawing in?

BORING IN A LATHE

Drilling a hole may be defined as one process of making a hole where none existed previously.

Boring may be differentiated from drilling in that it is the process of enlarging, by turning inside with some form of boring tool, a hole already existing, for example a hole already drilled, or a cored hole in a casting.

Reaming is the process of finishing a hole to the required size, by means of either a machine reamer or a hand reamer.

120. The Boring Tool.—The boring tool (the part that cuts), is a turning tool held in a bar or holder or forged on the end of the bar (Fig. 136). It is ground like a left-hand turning tool, a tool that cuts from left to right. It has side rake (see a, Fig. 137); a cutting angle of about 60°–70° (being a metal turning tool) and a rounded cutting point to give the tool a
longer life and the work a smoother finish. The cutting edge is not at right angles to the axis of the work but should be about 20° from this perpendicular as shown in the figure. This causes the chip to curl away from the finished cut and also reduces the tendency for the tool to spring into the work. The boring tool must have side clearance (b, Fig. 137) to permit of its biting into the work and thus peel off the chip; and also front clearance (c, Fig. 137) so as not to rub on the finished work.

The clearance must be sufficient to preclude any chance of rubbing but should not be excessive or the cutting edge, not being "backed up," will break away and dull quickly.

Have the boring tool bar as short as the length of the hole.
will permit and be very sure that the bar will clear as the tool works into the hole.

121. The Boring Tool Holder.—A boring tool holder in which the length of the bar, and the position of the cutting edge of the tool may be adjusted, will usually give better satisfaction than the forged tool. Figure 138 represents a boring tool holder having these advantages. The tool bit may be sharpened and re-set without disturbing the tool holder, which advantage is especially desirable when cutting an inside thread. The extra cap which holds the tool point at an angle is useful when boring to a shoulder or when squaring the bottom of a hole.

Figure 139 shows a "home made" boring tool holder which is very handy. The hole is filed V shape lengthwise opposite the screws to give a perfect seat for various sizes of tools. The tools may be made from pieces of drill rod.

122. Reasons for Boring.—It is often necessary, after drilling, to enlarge a hole with a boring tool, sometimes because a drill of the proper size to leave just enough for reaming
is not at hand; or because a machine reamer of the size to leave the correct amount for the hand reamer is not available; but usually for the reason of obtaining a hole which runs true.

If it is important to have a hole run true and central with the piece, as set up in the chuck or on the face plate, the hole should be bored. It is not safe to assume that a drill will run perfectly true through solid metal, even though it starts true; it might strike a blow hole, or a hard spot in the work, or it may become dull. These conditions will cause the drill to wobble. A reamer will follow the general direction of the hole as drilled, and the result, if the hole is not straight, will be most unsatisfactory.

A cored hole, if it is to be finished true, must invariably be bored. A three lipped drill is steadier and stronger, and is therefore better than a two lipped drill for drilling a cored hole, but the hole must be bored round and true after any drilling operation if accuracy is required.

Occasionally a core is not properly set in a mold and the hole in the casting is consequently out of center. In many such cases the casting may be adjusted in the chuck so that the eccentricity is divided and both surfaces be finished to size. (See Fig. 140.) In this event the hole should be trued up by boring, at least deep enough to give a fair start for the drill, possibly its full length.

123. The Operation of Boring a Hole.—Select a boring tool with a shank small enough to clear the hole and not unnecessarily long. When the shank is long, the spring of the tool is excessive, and the tendency to chatter is increased. Chattering, besides causing a poor appearance of the work, quickly dulls the cutting edge of the tool. It may be avoided, usually, by having a boring tool bar of sufficient cross section, not too long, and held rigidly.
Be sure that the boring tool is sharp, and that the clearance angles and rake angles are correct for the hole to be bored. It is never safe to assume that a boring tool or any other tool is right; examine it and be sure it is right for the job, and particularly that it is sharp. The best mechanic working on the best machine cannot do efficient work with dull tools; a real mechanic will not attempt it.

Set the cutting point of the boring tool as nearly on center as possible.

The speeds and feeds for boring are substantially the same as for turning a similar material. The chip, however, is usually less because of the spring of the tool.

The beginner should be warned against the tendency to bore bell-mouthed holes—holes larger at the beginning than further along. This is caused by taking several light cuts for a short distance to obtain the correct diameter, then throwing in the feed. When the tool gets to the heavier cut, it springs away somewhat, and of course bores a smaller dimension than at the start. In any event, an extra finishing cut with a sharp tool will probably be necessary if the hole must be exact.

Care must be taken to avoid springing or breaking the work when clamping in a chuck, especially work with a thin rim or wall, and it may be advisable to ease up on the jaws before taking the finishing cut.

124. Measuring a Hole.—The size of a hole is measured with an inside caliper, and the measurement may be read on a scale as shown (a, Fig. 141). This is all right for the roughing cut, but when accuracy is required it is better to use a micrometer to read the size. (See b, Fig. 141.)
Do not set the caliper to size and then try to judge how much of a finishing chip to take but measure the hole and then move the cross slide the desired amount. If several holes are to be bored, it is a good plan to have an extra caliper set to exact size to gauge the final cut.

Care must be exercised in measuring the hole. Hold the caliper straight across the diameter; do not cant it or tip it.

Questions on Boring Tools, Boring and Reaming

1. What is the difference between drilling and boring?
2. Is the boring tool a turning tool? Where does it cut? What part of the boring tool is the front? The side?
3. Will a boring tool properly ground for a hole 2 inches in diameter have the right clearance for a hole 1 inch in diameter? Give reason.
4. If a boring tool is properly ground and set, at what angle with the axis of the work is the cutting edge? Why? Why is it on center?
5. Give two reasons for rounding the cutting point of the tool.
6. Should the tool for boring cast iron and steel be given front rake and side rake? Give reasons.
7. In what respect should a tool for boring brass differ from a tool for boring steel? Give reason.
8. In what two directions does a boring tool have a tendency to spring? How may this spring be largely overcome?
9. What particular care must be taken regarding the shank or bar of a boring tool when setting up?
10. Using a carbon steel tool, what cutting speed is proper for boring machine steel? Cast iron? Tool steel? Brass? What cutting speed is proper for these materials if a high speed tool is used?
11. What r.p.m. of the work is necessary when boring a hole 2 inches in diameter in cast iron? In brass?
12. How is the proper feed determined in boring? The proper chip?
13. How is the hole measured? How is the measurement read if a scale is used? If an outside micrometer is used?
14. What is a bell-mouthed hole? Why will too many trial cuts at the beginning of a hole tend to make it bell-mouthed?
15. How is the graduated cross feed screw used in boring operations?
16. What is the object of boring a hole? When is it advisable to bore a hole that is afterwards to be reamed?
17. When is a machine reamer that is up to size used in lathe work? When is a machine reamer slightly undersize used? How much undersize should it be made?
18. How is a taper shank reamer held? How is a straight shank reamer held?
19. Why must the reamer be held back against the center during the whole cut?
20. How does the cutting speed of a reamer compare with that of a drill? How does the feed compare?
21. When is the hand reamer used in a lathe?
22. How much undersize should a hole be left for hand reaming?
23. How is the reamer started square? What must be the exact position of the dead center? Why?
24. How is the hand reamer turned when used in a lathe? How is it kept square?
25. Why should the hand reamer not be turned backwards?
26. When should a lubricant be used on a hand reamer?
27. Why must extreme care be taken when reaming pieces which have thin walls?
28. The body of a hand reamer is not cylindrical. What part is smaller? How much smaller? Why is it smaller? Is the rest of the body cylindrical?
29. How is a hand reamer given "clearance?"
30. What is meant by a reamer with "increment" cut teeth? What advantage has it?
31. What is an adjustable reamer? What are its advantages?
32. What are the advantages of a hand reamed hole?
33. As the chips do not fall out readily from a taper reamer what precaution must be taken?
CHAPTER IX

TAPERS AND ANGLES

125. Tapers.—One of the most important principles in machine shop practice is that involved in taper work, particularly the round taper shank and the round taper hole (Fig. 142).

![Fig. 142.](image)

There is hardly a revolving spindle in any machine that is not provided with a taper hole. This taper hole will receive and securely and rigidly hold the taper shanks of various tools such as centers, drills, reamers, etc. The correct position of the tool thus used is immediately obtained and indefinitely maintained, yet a slight blow serves to remove the shank from the hole.

Taper in round work may be defined as the difference in diameters, for any length, measured along the axis of the work. It is usually stated in tables, on drawings, etc., as the amount of taper per foot.

There are four parts to every taper, the amount it tapers per foot; the length of the taper; the large diameter; and the
small diameter (Fig. 143). There are various systems of
standard tapers in common commercial use, the most im-
portant being the Morse Standard, the Brown & Sharpe Stan-
ard, and the Taper Pin Standard. (See list of tables, p. 291.)

Twist drills are made with Morse Standard taper shanks,
up to and including $\frac{9}{16}$" with No. 1 taper, from $\frac{9}{16}$" up to
and including $2\frac{9}{32}$" with No. 2 taper, etc., etc.

The Brown & Sharpe Standard taper is used in milling
machines; the arbors, collets, end mills, etc. have shanks with
Brown & Sharpe tapers.

There is no standard taper for the shanks of lathe centers.
Each manufacturer seems to have established sizes of his own.
Do not attempt to use a center made for one lathe in another
kind of lathe, and do not use in a lathe a chuck with a shank
fitted to a drill press. The chances are the taper will not fit.

A disturbing fact in machine work which often results in
considerable confusion, is the number of mongrel sizes in the
various systems. For example, there are six different amounts
of taper per foot in the eight Morse standard tapers. No one
tries to remember the lengths, diameters, etc., a table of sizes
is necessary.

The Jarno system of tapers is the most sensible system.
In this series the number of the taper is the key by which
all the dimensions are immediately known. Thus, the number
of the taper is the number of eighths of an inch in diameter at
the large end, the number of tenths of an inch in diameter
at the small end, and the number of halves of an inch in
length. The taper is .6 per foot in each size. It is too late
to incorporate the Jarno system in drilling machines and milling
machines; there are too many million drills, reamers, end
mills, etc., in the shops to make a change feasible, but there
seems to be no real reason why the Jarno system cannot be
used in the new lathes.

To preserve the accuracy and efficiency of tapers (shanks
and holes) they must be free from dirt, chips, and nicks or
burrs. A most distressing sight is a taper, either a shank
or a hole, practically spoiled by being nicked and dented.
The most important single direction in regard to tapers is to *keep them clean*. The next important thing is to wipe them dry, because on oily taper will not hold.

**TAPER TURNING**

There are several methods of turning a taper in a lathe; by off-setting the tail stock slide; with a square-nose tool; by means of the compound rest; and with a taper attachment. When lathes are not provided with the taper attachment it is customary to obtain the taper by off-setting the tail stock slide. This will be the first method considered. The others will be explained presently.

126. Off-setting the Tail Stock Slide.—If the dead center is off-set, the center line of the work which is held on centers will not be parallel to the line of travel of the turning tool and the work will be turned taper. (See Fig. 144.) To find the proper amount to off-set the tail stock for a given taper on a piece of work requires a simple calculation, but there are a few things about off-setting the tail stock for turning tapers that must be thoroughly understood before calculations can be made intelligently.

*First.*—The more the given piece is off-set the greater will be the amount of taper.
Second.—The longer the piece of work the more the off-set required to obtain a given taper. For example: An off-set of \(\frac{1}{16}\)" for a piece 2" long would give a fairly steep taper but the same off-set for a piece 24" long would give a taper hardly noticeable, in fact, only one-twelfth as much as the first piece.

Third.—The length of the taper itself, that is the distance a taper is cut on a piece of work has nothing to do with the off-set. When making calculations do not get length of the taper confused with the length of the work.

Fourth.—Since the work revolves in the lathe only one half as much off-set is required to give the same amount of taper as if the work did not revolve.

Fifth.—The taper is proportional to the length—so much taper per foot. The off-set is proportional, but in the ratio of \(\frac{1}{2}\) the length because the work revolves.

Since most measurements in machine shop work, including lengths up to two feet or more, are expressed in the denomination of inches, when calculating the off-set, where the length is given in inches, the taper per foot is always reduced to taper per inch. (Divide taper per foot by 12.) Then the proportion will be

\[
\text{Off-set : taper per inch} = \frac{\text{Length of work in inches}}{2} : 1 \text{ inch.}
\]

that is,

\[
\text{Off-set} : T = \frac{L}{2} : 1
\]

or

\[
\text{Off-set} = \frac{TL}{2}
\]

This explains the derivation of the following:

127. Rule for Off-set when Turning Taper.—Multiply the length of the work in inches by the taper per inch and divide by two. The result will be the amount to off-set the tail stock.

128. Method of Gauging Off-set.—Assume that the lathe is set for straight turning and that a certain amount of off-set is required to turn the taper. Hold the tool post rigid by clamping a tool as in Fig. 145. Run the cross slide in until a piece of paper is lightly pinched between the tool post and the
tail stock spindle (a, Fig. 145). *Take up the lost motion in the cross feed screw*, and using the graduations on the collar, run the cross slide away from the spindle until the distance between the tool post and the spindle is equal to the amount of the required off-set (b, Fig. 145). Then adjust the *tail stock slide* until paper is pinched between the spindle and the tool post, thus obtaining the required off-set.

If necessary to off-set in a direction away from the operator, use a similar method. Arrange as above at a, Fig. 145, off-set the tail spindle a little further than necessary, run the cross slide in the proper amount, and adjust the tail stock back toward the operator until the piece of paper is pinched between the spindle and the tool post.

129. **Setting of Turning Tool.**—When calculating the required off-set, it may be readily proved that the three lines of the problem (the center line of the lathe, the center line of the work, and the off-set line) are in the same plane. Therefore in turning or boring taper it is absolutely necessary to have the cutting point of the tool on center.

130. **Methods of Measuring Tapers.**—For the reason that the centers enter the work a short distance, which fact is usually ignored in calculations, and also that there is a possibility of other errors, it is necessary always to *test the amount of taper before turning the work to size*.

The taper per inch of any turned piece may be easily obtained by dividing the difference in diameters by the length in
inches measured along the axis of the work between these diameters. For example: To find the taper of a sample piece, or to ascertain if a taper being turned is correct, the following method may be used to obtain an approximately accurate result. With pencil or scriber draw two lines on the surface of the taper parallel with the end, and if convenient, a whole number of inches apart. Measure the diameters at these lines and divide their difference by the number of inches between them. The result will be the taper per inch.

When obtaining the taper per inch it is the usual practice to consider the length as measured on the surface as near enough for practical purposes, because the difference between the length of an ordinary taper measured "along the axis of the work" and the length measured on the surface of the work is so small that it is, in most cases, not worth considering.

131. Fitting a Taper to a Gauge.—It is difficult to accurately measure the diameters of a taper with a spring caliper or a micrometer. A taper should be finally fitted to a gauge (Fig. 146), or to the spindle or sleeve for which it is intended. To try a taper draw three light chalk lines about equidistant along the length of the work and then wring the taper (to the left and it will not stick) a part of a turn in the gauge. If the chalk marks do not rub off evenly the taper is incorrect. When extreme accuracy is required, a very thin application of Prussian blue oil paint may be used instead of the chalk marks.

Standard taper gauges, external and internal (Fig. 146), are practically indispensable where accurate taper work is done.
132. Gauging the Size of a Taper.—A very quick, accurate method of gauging the size of a taper is to note the distance it goes into the gauge. If too large, it will not go in far enough; if too small it will go in too far. For example: a shank .6" per foot taper (.050 per inch) that does not enter the gauge within $\frac{1}{8}$" of correct depth is .025" too large; if it goes into the gauge $\frac{1}{8}$" too far it is .010" too small.

133. Duplicating a Taper Piece.—When a taper on a piece of work is to be duplicated, if it has centers it may be put in the lathe and the offset of tail spindle, or the adjustment of the taper attachment, or of the compound rest, be quickly obtained by means of an indicator placed in the tool post. When the setting is correct, the reading of the indicator will not change when moved along the length of the taper.

134. Turning a Taper with a Square Nose Tool.—It often happens that the easiest and quickest way to get an abrupt taper or angular cut on a piece is by means of a square nose tool. For example: The live center may be trued up with a square nose tool very efficiently. (This is illustrated in Fig. 82, p. 100.) It is only necessary to have a fairly broad square nose tool properly sharpened, set on center and to the desired angle.

135. Filing a Taper.—Most round work is either turned or ground to size. There are times, however, when a few strokes with a file will serve to fit a taper, that is nearly right and wanted in a hurry, much more quickly than it could be turned or ground.

136. The Taper Attachment.—If the centers of the lathe are in line but as the carriage is fed along the ways the tool moves toward (or away from) the operator gradually and uniformly, a taper will be cut. The object of the taper attachment (Fig. 147) is to control such a movement of the tool, that is, to cause the path of the tool when fed longitudinally, to move also, very slowly, in a lateral direction thus producing a taper. This is accomplished by means of a guide block connected to the cross slide (or to an extension of the cross slide) and moving in a guide bar set at the required angle to the center line of the lathe.
It is necessary when setting and using the taper attachment to loosen certain screws in order to permit of the cross slide being moved independently of the cross feed screw. The number, sometimes one, sometimes half a dozen, and the position of these screws, depend on the design of the attachment. When the screws are loose the cross slide may be moved freely back and forth by taking hold of the tool post. After the screws are loosened connect the cross slide to the guide block. The guide bar is pivoted at its center and may be rigidly secured in any desired position. Figure 147 shows one design of taper attachment. The principle is the same in all lathes.

The taper attachment has many features of especial value among which are the following: (1) The lathe centers are in line and the center holes in the work are not distorted. (2) The length of the work need not be considered, for once the taper is set, that particular taper will be turned on any length of piece. (3) The alignment of the lathe need not be disturbed, thus saving considerable time and trouble. (4) Taper boring is accomplished as easily as turning. (5) A much wider range
is possible than by the off-set method, for example to turn a taper ¾" per foot on the end of a bar 4' long would require a set over of 1½" which is outside the limit of a regular 14" or 16" lathe. Further, it is often convenient to use a combination of the off-set of the tail stock and taper attachment when turning tapers too steep for either method alone.

When using certain kinds of taper attachments, the machinist's enemy "lost motion" or "back lash" must be taken care of, or serious trouble will result. In every slide and every freely revolving screw there is a certain amount of lost motion. This is very noticeable if the parts are worn. In a taper attachment so designed that lost motion may occur anywhere between the tool and the guide; care must be taken that the lost motion is taken up in the right direction before proceeding to cut. Otherwise the piece will be turned or bored straight for a short distance before the taper attachment begins to work. To take up lost motion when turning taper, run the carriage back toward the dead center as far as possible without hitting the tool against the center, then feed forward by hand until the beginning of the cut, when the power feed may be thrown in. This operation must be repeated for every cut.

*Just as much judgment and care must be exercised in fitting a taper when using the taper attachment as when cutting a taper by any other method.*

137. Boring Tapers with Taper Attachment.—The best way to bore a taper in a lathe is by the use of the taper attachment. Extreme care must be exercised that the back lash or lost motion is taken care of when tapers are being bored with the taper attachment otherwise the hole will be bored straight for a certain distance, before the taper starts. Be sure that the boring tool is small enough to operate without rubbing at the small end of the hole.

138. Boring Tapers with Compound Rest.—Another method of boring a taper and one often used for very abrupt tapers (or angles) is by means of the compound rest. The compound rest is set around a certain amount to cut the desired taper or angle (see page 165), and the boring tool is fed by hand.
139. Fitting Taper Holes.—Taper holes are fitted to taper plug gauges similarly as taper shanks are fitted to taper ring gauges. (See paragraph 131.)

Taper holes are usually finished by reaming. For description of taper reamers, see page 140.

Questions on Tapers

1. Having found the taper per inch, how do you know the taper per foot?

2. How many parts, that is, dimensions, must be measured to determine the amount of taper?

3. When measuring a tapered piece with a caliper or micrometer, what care must be taken?

4. In measuring the taper why do you lay off a whole number of inches and not such a distance as 3 3/8 inches, or 4 3/2 inches?

5. What is the taper per inch of .6 taper per foot? Of 1/2 inch taper per foot? Of 3/4 taper per foot?

6. What is the taper per foot of .050 taper per inch? Of .042 taper per inch? Of .062 taper per inch?

7. How do you try a taper in a gauge? Why do you make three chalk marks? Why not one chalk mark? Why not cover the taper with chalk?

8. If the taper fits, but is too large and does not go in the gauge far enough, how do you determine from the amount it sticks out how much more to turn off?

9. Suppose the shank of a reamer is .6 per foot taper, and that it is required to leave .010 for grinding, how much further into the gauge will the shank go after it is ground?

10. Is there a taper hole in the spindle of the milling machine? Drilling machine? Grinding machine? Is there any revolving spindle in the shop which does not have a taper hole?

11. What are some of the cutting tools held by means of tapers? Are they held securely? Is the friction of the taper alone sufficient to hold them?

12. What are some of the advantages of the taper in machine shop work?

13. What is one of the most important considerations regarding tapers? What does a mechanic think of an ill-fitting or a damaged taper? What does he think of a taper that is nicked or burred?

14. Why are tenons or "tangs" milled on the ends of twist drills, machine reamers, and end mills? Why not on lathe centers?

15. What do you understand by the term "standard taper?"
two systems of standard tapers in commercial use for holding cutting
tools and state their chief difference.
16. Why will the B & S taper not fit in a drill press? Why will the
taper shank of a chuck fitted to a lathe spindle not fit in a milling machine
collet or in a drill socket?
17. What common cutting tools are provided with Morse standard
tapers?
18. In what machines are Brown & Sharpe tapers mostly used?
19. Do lathe centers have any standard taper?

Questions on Taper Turning

1. If the dead center is in line with the live center, a cylinder is turned.
   Why?
2. If the dead center is off-set, what shape is turned? Why? What
does the position of the dead center determine?
3. If the center is off-set toward the operator which end of the taper is
   smaller? If the off-set is in a direction away from the operator which
   end is smaller?
4. When off-setting the tail stock, how do you take care of the lost
   motion in the cross feed screw?
5. If a piece of work is one foot long and the dead center is off-set \( \frac{1}{4} \)
inch, the taper turned on this piece will be \( \frac{1}{2} \) inch per foot. Why?
6. If a piece of work is 2 feet long and the off-set of the tail stock is
   \( \frac{1}{4} \) inch, what will be the taper per foot? Why?
7. In the two preceding questions with the same off-set we have two
   different tapers? Give reason.
8. Suppose the machinist has two pieces of steel, one 12 inches long and
   the other 24 inches long, and it is required to turn the same taper on each
   piece. What will be the difference in the off-set? Which piece will
   require the more off-set? Why?
9. If the dead center is off-set \( \frac{1}{4} \) inch and a piece of work is turned
   taper a distance of 4 inches, and another piece, of the same size and the
   same off-set is turned taper a distance of 6 inches, will the taper per foot
   be the same in both cases? Give reason.
10. It may be stated that two factors, one of them being the off-set,
    determine the amount of taper that will be turned. What is the other
    factor?
11. Does the length of the taper have anything to do with the off-set?
    Give reason.
12. What is the rule for off-setting the tail stock when turning taper?
13. In the above rule, why divide by two?
14. How is the amount of taper given on drawings? How is it ex-
    pressed in charts of standard tapers?
15. A piece of work 7½ inches long is to have a taper 4 inches long ½ inch per foot. What off-set is required?

16. Calculate the amount of off-set for the following:
   (a) Length of work 8½ inches; taper 0.6 per foot.
   (b) Length of work 6½ inches; taper ½ inch per foot.
   (c) Length of work 9 inches; taper ¾ inch per foot.
   (d) Length of work 6¾ inches; taper .5 per foot.

17. Two pieces of the same length are to be turned taper; one has large centers, the other has small centers. Which will require the more off-set? Why?

18. If calculations are exactly right and set-over of tail stock is made accordingly, and to a thousandth of an inch, is it safe to assume that the taper will be correct and therefore turn to size before trying it? Give reason.

19. To make sure the correct taper is being turned, how may the piece be measured with a caliper or a micrometer?

20. If a micrometer is used, why is the measurement made with the edge of the spindle and anvil?

21. Why should the cutting tool be set on center when turning taper?

22. If a tapered piece is to be duplicated, how may the tail stock be adjusted without calculating the off-set? Can this be done if the new piece is longer or shorter than the sample? Give reason.

23. Under what circumstances only is it proper to fit a taper by filing?

24. If the centers of the lathe were in line, but as the tool was fed along it worked back gradually and uniformly, what shape piece would be turned?

25. In the taper attachment, how is the block guided? How is the guide bar pivoted? How adjusted? How is it tightened? Why is it tightened?


27. State at least four advantages of the taper attachment.

28. On what sort of a taper would the use of both the off-set and the taper attachment be advisable?

TURNING ANGLES

140. Angles.—An angle is the amount of the divergence between two straight lines that either meet in a common point, or would meet if sufficiently prolonged. The straight lines are called the sides of the angle and the meeting point is called the vertex of the angle. An angle is measured on the circumference of a circle drawn with the vertex as a center. The sides of the angle lay off a certain portion of the circumference.
The circumference is divided into 360 parts or degrees and the number of degrees between the sides of the angle is the measure of the angle. For example, if one-fourth of the circumference is intercepted between the sides, the angle is measured by 90 degrees (one-fourth of 360 degrees), and is commonly spoken of as an angle of 90 degrees, or a right angle (see Fig. 148). Also if one-sixth of the circumference is intercepted by the sides of the angle, the angle is measured by 60 degrees, or in other words, it is an angle of 60 degrees (Fig. 148). For fine angular measurements, the degree is subdivided into 60 parts called minutes and the minutes are subdivided into 60 parts, called seconds. The notations used are degrees (°), minutes (′), seconds (″). The subdivision of seconds are not used in ordinary machine work.

141. Classification of Angles.—

A right angle is equal to an angle of 90°.
An acute angle is less than 90°.
An obtuse angle is greater than 90°.

Two angles are called complementary angles when their sum is equal to a right angle and each is called the complement of the other. For example, 55° is the complement of 35°; 35° is the complement of 55°; 40° is the complement of 50°, etc.

Two angles are called supplementary angles when their sum is equal to two right angles (180°). For example, 55° is the supplement of 125°.

On drawings and blue prints, the angle is usually dimensioned as shown in a, Fig. 149. In formulas and calculations, it may be named by a small italic letter as shown in b, Fig. 149.

Tapers and Angles.¹—Tapering pieces up to an included angle of 8°² which is about 1 ¾ ″ taper per foot, are known as tapers

¹ For table of tapers and corresponding angles see list of tables, p. 291.
² A tapered piece over 8° included angle approximately will not hold in a taper hole.
and are measured as having "taper per foot." Pieces which are turned or bored to an included angle of greater than 8° are usually spoken of and are measured as having included angle or as having an angle with the center line (the angle with the center line is half the included angle).

To illustrate: On almost all lathe centers, the shank is turned taper .6" per foot, and the center is turned to an angle of 60°. (Sixty degrees is the included angle; the angle with the center line is 30 degrees.)

142. The Use of the Compound Rest for Turning Angles.
—The best method of turning an angle on a piece of work is to use a tool rest that may be swiveled on the cross slide to any desired angle. This tool rest is known as the compound rest (Fig. 150). The swivel plate of the rest is graduated in degrees, and the zero mark, which is usually on the side of the plate, is in line with a mark on the cross slide when the compound rest is at right angles to the center line of the lathe. The compound rest is provided with a hand feed which is independent of the lathe cross feed.

143. Setting the Compound Rest.—Setting the compound rest is comparatively easy if the operator realizes, first, that the compound rest is normally set at 90° from the center line of the lathe, and, second, that the travel of the tool is to be at a certain angle other than 90° with the center line of the lathe. In
order to cut at a certain angle with the center line of the lathe the compound rest must be swiveled either (1) the number of degrees which is the complement of the angle with the center line, or (2) 90° plus the angle with the center line.

To illustrate (1): Suppose that a bevel gear is to be turned to an angle of 67° 40', with its axis, that is 67° 40' (67\(\frac{2}{3}\)°) with the center line. The complement of 67\(\frac{2}{3}\)° is 22\(\frac{1}{3}\)°. Set the compound rest around 22\(\frac{1}{3}\)° from its normal position. (See Fig. 150.)

To illustrate (2): Suppose it is required to turn a lathe center 60° included angle, the angle with the center line is then 30°. Swivel the rest, to the right, 90° from its original position, then swivel it 30° more as shown (Fig. 151). If the compound rest were swiveled 60°, the complement of 30°, to the right it would be necessary to run the lathe backward and turn on the back side of the center. This is sometimes done. If the compound rest were set 60° to the left, the handle would probably interfere with the face plate.

There are different methods of dimensioning the degrees of the angles on a drawing. The dimension may be given as an included angle, a, Fig. 152; as the angle with the axis, b, Fig. 152; or as the angle with a line perpendicular to the axis, c, Fig. 152. It is always best to find the angle with the center line and then set the compound rest in one of the ways suggested in the preceding illustrations. A
sketch will help in determining the correct position at which to set the compound rest.

144. The Bevel Protractor.—The instrument used in machine shops for measuring angles is called the bevel protractor (Figs. 153 and 154).

The principle of construction of a bevel protractor is as follows: Two members, which may be called the beam A and the blade B with edges straight and parallel, are so arranged as to swivel on a pivot at the center of the dial C which is graduated in degrees. When the edges of the beam and blade are parallel, a small line on the swivel plate D coincides with the zero line on the dial, and when any measurement of an angle between the beam and the blade of 90° or under is desired, the reading may be obtained direct from the position of the line on the swivel plate with regard to the graduation numbers on the dial. But remember this: To obtain the measurement of the angle between the beam and the blade of over 90° subtract the number of degrees as indicated on the dial from 180°. This is because, as will be noted, the dial is graduated from opposite zero marks to 90° each way.

Figure 155 illustrates a variety of uses of the bevel protractor.

Note.—Vernier reading of Brown and Sharpe protractor is explained in appendix, page 287.
Fig. 155.
Questions on Angles

1. What is an angle and how is it measured?
2. What is an acute angle? An obtuse angle?
3. In what way is an angle usually dimensioned on a drawing?
4. What is meant by the term “included angle?” "Angle with the center line?"
5. What is meant by the complement of an angle? Complementary angles? Give an example of each.
6. How is the compound rest of a lathe graduated?
7. If the compound rest is set around 30° from normal position, what will be the included angle of the piece turned?
8. Why must extreme care be taken when setting a compound rest to turn a given angle?
9. Why is it best before setting the compound rest to determine at what angle with the center line the given cut is to be?
10. Explain the principle of the construction of the bevel protractor.
11. What caution must be taken when reading the measurement of an angle over 90° on the bevel protractor?
CHAPTER X

THREADS AND THREAD CUTTING

145. Threads.—A screw is a cylindrical bar into which a helical groove is cut or formed leaving a projection commonly called a thread. In all screws the profile of the groove is substantially the same as the thread.

A nut or any inside threaded piece is the reverse of the screw, having the thread formed on the hollow inside or hole, and fitting, more or less snugly, thread to groove over the screw of corresponding size.

The screw together with an inside threaded piece, may be used for obtaining and maintaining pressure; or to transmit motion; or as a fastening agent.

A thread on a screw may be formed by rolling in a special machine; by cutting with a revolving cutter as in a thread milling machine; by turning in a lathe; or by means of a master tool called a die.\(^1\) In machine shop practice the two methods last named are in common use.

Threads may be formed on the hollow inside as in a nut by boring in a lathe, but the usual practice is to form the threads by means of a master tool called a tap.\(^2\)

There are various common forms of threads used for different purposes and in different localities; for example, the lead screw of the lathe is probably an “Acme” thread; the cross feed screw is usually a “Square” thread, the thread on a pipe is a modified “V” thread; the threads on most of the bolts, screws, etc., made in the United States, are “U. S. Std.” and the commonly used thread in England is the “Whitworth Standard.” These threads will be explained presently but for the present

\(^1\) Threading Die: For description see page 182.
\(^2\) Tap: For description see page 177.

170
the following definitions apply to all threads and it is important to know them.

146. Definitions of Thread Parts (Fig. 156).—The Pitch of a thread is the distance from a part of one thread to the corresponding part of the next thread measured along the axis of the thread. (If a screw had eight threads per inch the pitch is $\frac{1}{8}''$.)

The Lead of a thread is the distance it will move into the nut in one revolution. (In a single thread the lead is equal to the pitch, in a double thread the lead is equal to twice the pitch, etc., etc.)

The Angle of a Thread is the included angle between the sides of the thread, measured on the axial line. The angle of the thread in Fig. 156 is 60 degrees.

The diameter of the piece on which the thread is formed is termed the Outside Diameter or nominal diameter. The perpendicular distance from the top of the thread to the bottom of the groove is called the Depth of thread. Twice this distance is called the Double Depth of thread. The outside diameter (O. D.) minus the double depth (D. D.) gives the effective diameter or Root Diameter (R. D.). The root diameter is the size of the hole to be bored in the nut or inside threaded piece to give a full thread.

The Pitch Diameter of a thread is equal to the nominal outside diameter less the depth of one thread. It is used when measuring threads with a screw thread micrometer (see Appendix, page 289).

Threads may be Right Hand (R. H.) or Left Hand (L. H.). Looking at the end of the screw the R. H. thread advances clock-wise and the L. H. thread advances anti-clockwise. The tail spindle screw, for example, has a left-hand thread. Most threads are right hand.
147. Thread Standards.—A generation or so ago the common bolts and screws were cut with a thread tool that was ground theoretically to a sharp V with an angle of 60 degrees and were cut deep enough to bring the thread to a more or less sharp V. Finding it impossible to keep or even make thread cutting tools, taps and dies with even a fairly sharp V point, the manufacturers adopted various modifications of the theoretical V form. They rounded or flatted the point of the tool, and cut the thread on the screw and on the tap a trifle less deep thereby leaving a small flat on top and bottom of the thread.

Since each manufacturer made screws or taps or dies to suit his own purposes, the screws made in one place might not fit tapped holes made in another shop or perhaps even in the same shop.

Further, there was no standard pitch; one man might like the looks of 10 threads per inch on a $\frac{3}{4}''$ bolt, another man might prefer 9, and another 12, etc., etc. Therefore, if a bolt or screw in a machine was broken, one to replace it could not be obtained from stock or bought in the open market, it had to be specially made and fitted.

As there was, then, no uniform modification of the $60^\circ$ V thread and no standard pitch for each diameter, interchangeability was practically unknown until about 1869. At this time William Sellers of Philadelphia suggested a uniform flat on top and bottom equal to $\frac{1}{8}$ of the pitch of the thread, and John Fritz of Bethlehem, Pa. who was having some machines made in the Sellers Shops, suggested a standard pitch for each diameter $\frac{1}{4}''$ and over. These ideas met with the approval finally of most engineers and have been adopted by the Army and Navy Departments, by the railway systems and by the great majority of the manufacturers. This shape of thread and size of thread for the given diameter are now known as the U. S. Std. form of thread and the U. S. Std. pitch respectively.

The automobile manufacturers have adopted a system, called Society Automotive Engineers Standard (S. A. E. Std.)
of finer pitches than U. S. Std. The American Society of Mechanical Engineers adopted in 1907 a system (A. S. M. E. screw threads), of standard pitches for screws under 1/4" diameter. Both systems use U. S. Std. form of thread.

The U. S. Std. form of thread has been adopted as the standard of the International System and the French System. These systems are metric threads and the pitches are therefore slightly different from U. S. Std. pitches.

In England the standard thread is known as the Whitworth Standard. It was devised in 1841 by Sir Joseph Whitworth, undoubtedly the foremost mechanic of his time. It is quite unlike the U. S. Std. having an angle of 55 degrees instead of 60 degrees and having a round top and bottom instead of the flat as in the U. S. Std.

The chart (Fig. 157) shows the various forms of threads. The tables of standard pitches etc. may be found in the list of tables, in this book.

148. The Dimensions of The U. S. Standard Thread.—It is very necessary that the machinist be able to calculate all the dimensions of the U. S. Standard form of thread, and if he understands thoroughly the development of the U. S. Standard, that is, the how and why of the calculations relating to this form of thread, he can calculate the dimensions of any form of thread from the formulas given in hand books and in most catalogs of drills, taps, etc., such as for example are given in the chart, Fig. 157.

The U. S. Standard Thread has an angle of 60 degrees. It is in the form of an equilateral triangle (a triangle having equal sides and angles) with the apex flatted. The top of the threads and the bottom of the grooves are not sharp as in the V thread or rounded as in the Whitworth, but are flat. The width of this flat is 1/6 of the pitch of the thread.

Many beginners make the mistake of thinking that they should first cut the thread to a sharp V and then file off or turn off the top to produce the flat. It must be understood that when the U. S. Std. thread is cut, a flat is left on top. In other words, the U. S. Std. thread is never cut as deep as a V thread.
Fig. 157.—Chart of threads.
Draw any two equilateral triangles, the base of one twice as large as the base of the other. (For example A 2" base and B 1" base, Fig. 158). Draw their altitudes. (The altitude of a triangle is the perpendicular distance from the apex to the base.)

Note that the altitude of B is just one-half the altitude of A. In any two equilateral triangles, the altitudes are proportional to the bases respectively (and of course the bases are proportional to the altitudes). Further to prove this graphically, draw triangle C with altitude equal to \( \frac{1}{8} \) of the altitude of triangle A and note that the base is equal to \( \frac{1}{8} \) of the base of A.

Note, also, that the length of each altitude is about \( \frac{1}{8} \) of the base. It is not exactly \( \frac{1}{8} \) (.875) of the base but is exactly .866 of the base. In every equilateral triangle the altitude is proportional to the base as .866 is to 1.000.

![Fig. 158.](image)

Draw a light line (Fig. 159), 2" long, divide it equally into AB and BC, and using AB and BC each as a base draw lightly two equiangular triangles. Connect the vertices by dotted line P. Draw the altitude and divide it into eight equal parts. Draw light lines through top and bottom dividing lines. Trace heavy line and shade as shown and the figure will represent the profile of two threads with the groove between.

P equals the pitch of the thread, being "the distance from a part of one thread to the corresponding part of the next").

D equals the depth of the thread being "the perpendicular distance from the top of the thread to the bottom of the thread."
The pitch of the thread is equal to the base of the original triangle in the figure but since $\frac{1}{8}$ of the altitude was cutoff to give $\frac{1}{8}$ of the pitch flat on top and another $\frac{1}{8}$ of the altitude was cut off to make the flat at the bottom of the thread (refer to proportional parts of triangles A and C, Fig. 158, to prove this) then the depth $D$ of the thread is $\frac{3}{4}$ of the altitude of the original triangle.

The altitude of the original triangle is equal to .866 of the base, therefore the depth of the thread is equal to $\frac{3}{4}$ of .866 of the base or .6495 of the base.

The base of the original triangle is equivalent to the pitch of the thread, therefore the depth of a U. S. Std. thread is equal to .6495 of the pitch or .6495 × $P$ or as usually stated $\frac{\pi}{3}$.

The double depth equals $2 \times .6495P$ or 1.299P.

The root diameter equals the outside diameter minus 1.299P.

To multiply by the pitch is equivalent to dividing by the number of threads per inch, therefore, to obtain the root diameter, subtract from the outside diameter the quotient obtained by dividing the constant 1.299 by the number of threads per inch, or expressed as a formula:

149. Formula for Finding Exact Root Diameter of U. S. Std. Thread.

\[
R.D. = O.D. - \left( \frac{1.299}{N} \right)
\]

Example: Find the root diameter of a screw $\frac{3}{4}''$ dia. 10 threads per inch. Solution: $750 - (1.299 \div 10) = .750 - .1299 = .620 = R.D.$

The above formula is one of the most important in lathe practice. Be sure to understand it.

Questions on Threads

1. In what ways may threads be formed on a screw? In a nut?
2. Name and describe three different forms of threads.
3. What is the difference between the pitch and the lead of a thread?
4. What is the difference between the U. S. Std. and the S. A. E. Std.? Between the square thread and the Acme thread?
5. Define “root diameter” and “double depth.”
6. What is meant by "modified V thread?"
7. What is meant by the term "standard" as applied to thread pitches? As applied to thread forms?
8. What are the differences between the U. S. Std. and the Whitworth Std. forms of thread?
10. State the double depth of a U. S. Std. thread in terms of pitch.
11. What is the formula for determining the root diameter of a U. S. Std. thread?
12. What is the exact root diameter of a ¾"-9 U. S. Std. thread?

150. Taps.—Most internal threads are cut with taps, usually in a tapping machine or with a tapping attachment in the drill press. Many threads, however, must be tapped by hand.

Figure 160 shows a set of machinist's hand taps squared on the shank end to receive the wrench (Fig. 161). Taps are manufactured in all of the standard sizes and pitches for the stand-

ard forms of threads. They are generally turned from the solid bar; accurately threaded; carefully and scientifically fluted, that is, grooved to form cutting edges; marked as to size and number of threads per inch; hardened and tempered.
151. Tap Sets.—Hand taps, except the sizes under $\frac{1}{4}''$, are made in sets of three taps, (see Fig. 160), called taper, plug and bottoming. The first tap or “taper tap” is tapered or “chamfered” back from the end at least six threads, the plug is chamfered about three or four threads, while the bottoming tap is merely backed off on the end teeth. Taps furnished in sets are of the same diameter unless otherwise specified, so that to tap a through hole it is only necessary to use the taper tap. Where the hole does not go through the piece (“blind hole”) it is customary to start with the taper, follow with the plug, and, occasionally, if the hole is fairly shallow, finish with the bottoming.

152. Relief of Taps.—The cutting edges of the chamfered portion of the tap are given clearance, that is, they are “backed off” or “relieved” the whole width of the land, otherwise the tap will not “bite.” Further, in order to reduce the friction between the teeth of the tap and the work being tapped, the teeth are relieved about two-thirds the width of the land (Fig. 162). The remaining third of the land back of the cutting edge remains the full cutting size so that the tap may be ground on the face of the teeth several times without changing the size.

153. Tap Size Drills.—The diameter of the hole to be drilled for the threads in a nut or any inside threaded piece is theoretically the root diameter of the corresponding screw size. This size of hole will give a full thread which is required in a die and occasionally in a special piece, but it is not practical or customary to tap a full thread, therefore the tap drill sizes are usually larger than the root diameter.

Two-thirds of the double depth of thread is enough to leave for tapping. An ordinary nut drilled out so that it has only half of a full depth of thread will break the bolt before it will strip. A $\frac{3}{8}$ depth of thread will give a margin of safety of
about 2 to 1 and only requires about \( \frac{3}{2} \) the power for tapping that is required to tap a full thread. As explained in paragraph 148, the full double depth of a U. S. Std. thread is obtained by dividing 1.299 by the number of threads per inch. Consequently, dividing \( \frac{3}{2} \) of 1.299 by the number of threads per inch will give \( \frac{3}{2} \) of the full depth of thread (\( \frac{3}{2} \) of 1.299 = .866 or .9 approximately).

Therefore in ordinary machine work, if a list\(^1\) of tap drill sizes is not at hand, the size of a hole to be drilled for tapping may be found by subtracting from the outside diameter of the tap, the quotient obtained by dividing .9 by the number of threads per inch. Select the size of drill required or the nearest 64th under for taps over \( \frac{1}{4} \)" diameter, and the nearest number size for taps \( \frac{1}{4} \)" and under.

Example: \( \frac{1}{2} \) tap — 13 threads. Solution: \( 0.500 - \frac{0.9}{13} = 0.500 - 0.069 = 0.431" \). The nearest drill under .431" is \( 2\frac{7}{64} " (0.421") \).

154. Length of Tapped Hole.—Except for the purpose of holding the screw fixed, as in a tool post for example, it is unnecessary to have the tapped portion of a hole longer than one and one-half times the diameter of the screw. In fact a well fitting screw entering a tapped hole a distance equal to its diameter will break about as soon as the threads will strip. This fact is often overlooked and deep tapped holes are called for, which makes for waste of time and breakage of taps.

155. The Operation of Tapping.—A certain pressure is needed to start the taper tap and care must be taken to make the tap "bite" or "catch the thread" and not ream the top of the hole taper. After the tap is well started it feeds itself and requires only to be turned. It is a good plan occasionally to turn the tap backward half a turn to break the chip, and in tapping soft material such as copper, babbitt, etc., it is necessary to remove the tap several times and clean away the chips.

Care must be taken to start the tap square and keep it square. A tapping bushing is a valuable aid. It consists merely of a tapped bushing faced square. If the diameter of

\(^1\) For list of tap drill sizes see tables 16, 17 and 18, pages 309 to 313.
the bushing is three or four times the diameter of the tap it will effectively prevent the tap from tipping.

Use lard oil as a lubricant when tapping steel, wrought iron or other metals, except cast iron. Tap cast iron dry (sometimes, however, a little oil or soap put on the teeth that are rubbing in that part of the thread already cut will ease the tap). Many mechanics prefer to use a mixture of turpentine and lard oil when tapping copper and a mixture of kerosene and lard oil when tapping aluminum.

A sharp V thread tap is an excellent roughing tap where a smooth tapped hole is required, since it leaves a small amount of stock on the sides of the thread for the standard tap to remove as a finishing cut.

Wrapping a suitable piece of waste over the end of the tap and turning it through, will serve to make the tapped hole slightly larger.

Taps are usually obtained from the tool room in sets, together with the body size and tap size drills, and often with the counterbore for the head of the screw.

Fig. 163.—‘‘Gun’’ tap (Greenfield Tap and Die Corp.)

The G. T. D. Gun Tap (Fig. 163) is a very efficient tap for the reasons that it is strong, shears the chip, and does not clog since the chip ‘‘shoots’’ out ahead of the tap.

Cautions.—Take note when re-sizing or ‘‘cleaning’’ a previously tapped pole that the piece has not been hardened, otherwise the tap may be ruined. For instance do not attempt to re-tap a case-hardened nut.

Certain manufacturers still use special pitches of threads. For example ½″−12 is frequently used for set screws instead of ½″−13. If a standard pitch screw does not fit easily, determine the pitch of the tapped thread before forcing in the screw.
LATHE—THREADS

Questions on Taps and Tapping

1. Name the three taps in a tap set. What is the purpose of each one?
2. How do you make sure that a tap is started square? Why is it necessary?
3. How hard is a tap? Is it brittle? Why does it break easily?
4. What is meant by the “feel” of a tap when cutting?
5. What is an adjustable tap wrench? Solid wrench? What are the advantages of each? What is a disadvantage of an adjustable wrench?
6. What is meant by “clearance” on a tap?
7. Why is it advisable when tapping to frequently turn backwards a quarter or a half turn?
8. When threads are tapped in tough metal, what should be done to keep them from tearing?
10. How far should a screw enter a tapped hole in order to give sufficient strength?
11. What is a “blind” hole?
12. If the effort to turn the tap is continued after it bottoms, what will be the result?
13. How do you use a scale when counting the number of threads per inch? What is a “pitch gauge”?
14. How may the pitch of the thread in a nut be determined with a whittled stick, if there is no tap or bolt to fit it?
15. What size is the largest drill that will pass through the nut?
16. What is a tap drill? What is meant by a “body size” drill?
17. Why is a tap drill smaller than the outside diameter of the bolt?
18. What form of thread have S. A. E. Standard screws?
19. How many threads per inch has the 3/4” U. S. Standard screw? The 3/4” S. A. E. Standard screw?
20. Why are the screws used in automobile work of finer pitch than those used in machine tool manufacture?
21. In common practice, is the tap drill size equal to the root diameter of the thread? Is it larger or smaller?
22. State three objections to the use of a tap drill that will give a full thread.
26. When two pieces are to be held tightly together why not tap both pieces?
27. If it is desired to tap a slightly larger hole in a nut so it will be a free fit on a thread, how can this be done with a standard size tap? 

28. Before attempting to re-tap an old nut, what precaution should be taken? Why?

29. It occasionally happens that a ½" screw will not fit a tapped hole which appears all right. What caution must be observed when using ½" set screws, ½" taps and ½" dies?

30. How is a tap sharpened? How is it "backed off?"

156. The Threading Die.—A die is a tool for cutting external threads. In general the threading die is so arranged as to permit the cutting edges of four cutters or chasers to do an equal share toward cutting their shape (the form of the thread desired) into a cylindrical rod, when turned or "screwed" on the end of the rod for a distance of the required length of the thread.

Some dies are made solid, some in two halves within a body or "head" and still others with the four separate chasers properly and securely held in the head. The last named (see Fig. 164) is perhaps the best type since the chasers can be easily removed and sharpened. It also permits of considerable adjustment which is a decided advantage when a screw slightly over-size or under-size is required, or when a roughing and finishing cut is desirable. The complete die when locked together seems to have all the advantages of a solid die.

Fig. 164.

Most threads used in manufacturing are cut with dies in screw machines and bolt machines. Very often, however, it is practicable to size a fairly long thread or even to cut the whole thread, "by hand" in which case the die is held in the die stock or screw plate (see Fig. 165).

Fig. 165.
When threading a piece by hand the end of the rod should be chamfered about 45° for at least the depth of the thread and care must be exercised in starting the die true. As in starting a tap, pressure must be exerted when starting the die, but after it is well started it will feed itself. Use lard oil when cutting a thread on steel and wrought metals, and turn backward part of a turn occasionally to break the chip.

Questions on the Use of Threading Dies

1. How can you chamfer the stock with a file? By what other means may it be chamfered?
2. Give two reasons for chamfering.
3. What care must be taken in starting a die? Why is it necessary?
4. Is it better to cut the entire thread in one cut or in two cuts? Why?
5. How do you adjust the die for this?
6. What lubricant is best to use when cutting a thread on steel or wrought iron?
7. Why is it advisable when cutting threads with a die to frequently turn backward a quarter or half turn?
8. When cutting the thread, what should be done to keep it from tearing?
9. How do you protect the thread already cut on one end of a stud when threading the other end?
10. When it is desired to cut the thread close to the head on a bolt or cap screw, how can it be done?
11. What is meant by a die? Die holder? Die stock?
12. What is meant by a solid die? Adjustable die?
13. What advantage has an adjustable die?
14. What special advantages has a die with removable cutters?
15. What precaution should be taken before attempting to recut threads on an old bolt or screw? Why?

GEARING A LATHE FOR CUTTING THREADS

To cut a thread in a lathe involves the use of gears. It is therefore essential that the beginner understand the first principles of spur gearing in order intelligently to set up his machine.

157. Definitions and Explanations of Terms Used.—A Spur Gear is a toothed wheel or cylinder with the teeth parallel to the axis. The smaller of the two gears in mesh is often called the Pinion.
A Train of Gears is a series of two or more gears with teeth in mesh. The motion of the first gear causes each gear in the train to move.

A Bank of Gears is a number of gears arranged together and revolving on or keyed to the same shaft or sleeve. When there is a bank of gears of different sizes arranged successively, the collection is often called a Cone of Gears.

In a simple gear train, the gear to which motion is first imparted is the Driving Gear and the gear of this train to which motion is finally transmitted is the Driven Gear, or Follower.

In a gear train the speeds of the gears are inversely proportional to the number of their teeth. For example: A driving gear has 20 teeth and a follower gear has 40 teeth; one revolution of the driving gear will engage 20 teeth of the follower gear and cause it to make one-half of a revolution, that is, the follower gear, which is twice as large as the driving gear will revolve half as fast. In the same way the follower gear half as large as the driving gear will revolve twice as fast. The speeds are not directly proportional to the number of the teeth of the gears, but are indirectly or inversely proportional.

A gear in mesh between the driving and follower gears is called the Intermediate. The purpose of an intermediate gear is to connect two gears that are too far apart to mesh with each other.

In a gear train each gear revolves in the opposite direction to that of the gear with which it meshes, therefore, adding an intermediate gear serves to change the direction of the follower gear. An intermediate of any number of teeth or any number of intermediates may be used, and not change the relative velocities of the driving and follower gears.

Example.—Driving gear 28 teeth, driven gear 28 teeth, one revolution of the driving gear will cause one revolution of the driven gear. Introduce one intermediate of any number of teeth, say 60 teeth, one revolution of the driving gear will engage 28 teeth of the intermediate and it in turn will engage 28 teeth of the follower gear causing it to make one revolution or the same as was obtained without the intermediate.
The direction of rotation of the follower gear, however, is changed.

In simple gearing then, the size of the intermediate may be disregarded. The velocity of the driving gear is to the velocity of the follower gear inversely as the numbers of their teeth.

Thread cutting in an engine lathe is accomplished by causing the lathe carriage to move, positively, a certain distance for each revolution of the main spindle.

The positive movement of the carriage is obtained by, first, connecting the main spindle to the lead screw by gears, thus transmitting any movement of the spindle, positively, to the lead screw; and, second, closing tightly the split nut upon the lead screw thereby ensuring a positive movement of the carriage for each revolution of the lead screw.

When cutting threads in a lathe the motion of the spindle is transmitted to the stud shaft by the tumbler gear train and from the stud shaft to the lead screw by the change gear train. (See Fig. 166.)

The tumbler gear train consists of a gear keyed to the spindle, two tumbler gears (or reverse gears), and the fixed stud gear which is keyed to the inside end of the stud shaft (Fig. 166).

The two tumbler gears are intermediate gears between the spindle gear and the fixed stud gear and are so mounted on a bracket as to make it possible for the operator to have one intermediate or two intermediates in mesh between the driving and driven gears or to throw them both out of mesh with the
driving gear. That is, with the three positions of the handle as shown (Fig. 167), the lathe-hand may have (1) forward movement, (2) reverse, (3) neutral or no motion of the stud shaft.

The function of the tumbler gears is to reverse the direction of rotation of the feed rod, if for any reason it is desired to feed toward the tail stock, or of the lead screw when cutting left-hand threads. With a forward motion of the spindle and of the lead screw, the carriage advances toward the head stock and a R. H. thread is cut. To cut a left-hand thread, the work turns forward just the same but the direction of the lead screw is reversed thus moving the carriage toward the tail stock.

The change gear train (so called because the driving and follower gears may be changed at the will of the operator) consists of the gear on stud, the intermediate, and the gear on screw.

![Fig. 167.—Illustrates operation of reverse gears or tumbler gears.](image)

(See Fig. 166.) A series of different gears called change gears are furnished with the lathe and by changing the sizes of the gears on stud and screw various velocity ratios between the two may be obtained.

158. Operation of the Gears.—When the spindle gear and the inside stud gear of the tumbler gear train (Fig. 166) are of the same size, and the gears on stud and screw are equal, one revolution of the stud shaft will cause one revolution of the lead screw. If the lead screw is $\frac{1}{4}''$ pitch the carriage will advance $\frac{1}{4}''$. The number of threads per inch which will be cut on the work will be the same as the number of threads per inch on the lead screw. However, a great many lathes are made with the inside stud
gear larger than the spindle gear. Suppose the spindle gear has 30 teeth and the inside stud gear has 40 teeth; one revolution of the spindle will cause \( \frac{3}{4} \) of a revolution of the stud shaft and with equal gears on stud and screw cause \( \frac{3}{4} \) of a revolution of the lead screw.

If the lead screw is \( \frac{1}{6}'' \) pitch, the carriage will advance \( \frac{3}{4} \) of \( \frac{1}{6}'' \) or \( \frac{1}{8}'' \) and eight threads per inch will be cut on the work.

If the inside stud gear is twice as large as the spindle gear and the lead screw has six threads per inch, twelve threads per inch will be cut on the work with equal gears on stud and screw.

**Lead Number.**—The number of threads per inch that are cut with equal gears on stud and screw is the basis of all calculations for change gears for thread cutting and is called the *lead number*. The lead number for any lathe may be found as follows: Find the ratio of the number of turns of the *spindle* to the number of turns of the *stud shaft* (whole numbers) and multiply the number of threads per inch on the lead screw by this ratio. For example: 4 turns of *spindle* to 3 turns of *stud shaft*, 6 threads per inch on lead screw, 

\[ 6 \times \frac{3}{4} = 8 = \text{lead number}. \]

Any lathe with a lead number of 8 will cut eight threads per inch with equal gears on stud and screw, will cut sixteen threads per inch if the gear on screw is twice as large as the gear on stud. Or it will cut four threads per inch if the gear on screw is half as large as the gear on stud.

169. Calculating the Sizes of Gears to Cut a Given Thread. —The *rule for change gears* may best be stated in the form of an equation:

\[ \frac{\text{Lead Number}}{\text{No. of threads per inch}} = \frac{\text{Gear on Stud (Driving Gear)}}{\text{Gear on Screw (Follower Gear)}} \]

or as a formula:

\[ \frac{L}{N} = \frac{D}{F} \]

By this equation the correct gears to cut any thread, whole or fractional, may be quickly ascertained.
Example.—Lead number 6; threads per inch 10; available gears 20 to 80, progression\(^1\) 4, \(\frac{3}{10}\) is the ratio of the driving gear to the follower gear (gear on stud to gear on screw).

Solution.—A 6-tooth gear on stud and a 10-tooth gear on screw would cut the thread, but no such gears are available. To multiply both numerator and denominator of a fraction by the same number does not alter the ratio, therefore, multiply by four and the result equals

\[
\frac{6 \times 4}{10 \times 4} = \frac{24}{40}
\]

Use a 24-tooth gear on stud and 40-tooth gear on screw.

Another example:

To cut 11\(\frac{1}{2}\) threads per inch

\[
\frac{6}{11\frac{1}{2}} \times \frac{6}{6} = \frac{36}{69}
\]

Use a 36-tooth gear on stud and 69-tooth gear on screw.

(Note.—A 69-tooth gear is usually furnished with a lathe.)

Index Plate.—An index plate is fastened to the side of the lathe to show which change gears to use when cutting threads. However, a specified gear may be mislaid or broken, or a thread not given on the plate may be required. The thinking mechanic is resourceful.

160. Compound Gearing.—Suppose the smallest gear available is at 24-tooth gear and it is necessary to have the follower gear revolve one-sixth as fast as the driving gear. For example, to cut 36 threads with a lead number of six. In a simple train this would require a 144-tooth follower gear \(\frac{6}{36} = \frac{24}{144}\). If such a gear is not available, or if the center distance between the shafts does not permit of its use, an arrangement known as compound gearing may be used to obtain the required result.

\(^1\) Change Gear Progression.—By "progression" in change gears is meant the regular increase in the number of teeth in each succeeding gear in a set of gears. The sizes of the gears increase by a certain number of teeth from the smallest to the largest gear. In the above case by 4 teeth from 20 teeth to 80 teeth.
In either arrangement (a or b, Fig. 168) is illustrated a compound of two simple gear trains, A the driving gear and B the follower gear of the first train, and C the driving gear and D the follower gear of the second train.

While the compounding gears B and C are arranged between the original driving gear A and the final follower gear D, it is evident that their sizes are not disregarded as is an intermediate in a simple train of gears. Suppose gear A to revolve 6 r.p.m., B would revolve three times, C keyed to the same shaft as B, would revolve three times and would cause D to revolve once because D has three times as many teeth as C. The gears B and C are together known as the compound. In many lathes a “compound” is furnished with the machine. It consists of two gears, one having twice as many teeth as the other, fastened together and arranged on a special bracket between the stud gear and the intermediate. This is illustrated in b, Fig. 168, the gears B and C forming the compound. In other lathes provision is made to substitute compounding gears for the intermediate as shown in a, Fig. 168.

In compound gearing the velocity of the original driving gear is to the velocity of the final follower gear, inversely as the product of the driving gears is to the product of the follower gears. The formula used for a compound train is the same as for a simple train if D equals the product of the
driving gears and \( F \) equals the product of the follower gears. Therefore to ascertain the proper gears to use for compound gearing the same rule is used to find the four gears as for finding the two gears in a simple train, namely:

\[
\frac{\text{Lead No.}}{\text{No. Thds. Req'd.}} = \frac{\text{Driving gears}}{\text{Follower gears}}
\]

For example: Required to cut 28 threads per inch, lead number 6, progression 4. Arrange the ratio of the lead number to the number of threads to be cut \((6 : 28)\) as a fraction \(\frac{6}{28}\) and factor. \(\frac{6}{28} = \frac{2 \times 3}{4 \times 7}\), now multiplying the numerator 2 and the denominator 4 by the same number does not change the value of the fraction and multiplying 3 and 7 by the same number does not change the value of this fraction. Multiplying 2 and 4 by 16 = \(\frac{32}{64}\) and multiplying 3 and 7 by 8 equals \(\frac{24}{56}\); that is, \(\frac{6}{28} = \frac{2 \times 3}{4 \times 7} = \frac{32 \times 24}{64 \times 56}\). Gears 32 and 24 are the driving gears and 64 and 56 are the follower gears. If a compound gear in a 1 : 2 ratio is furnished with the machine it may be used instead of the 32 and 64 gears. If either the 24 or the 56 gear is not available multiply 3 and 7 by any number which will give two gears that are available, for example, multiplying by 12 gives 36 driver and 84 follower and these gears may be at hand.

Note.—One of the best examples of compound gearing in the machine shop is to be found in the “back gears.”

**THREAD CUTTING**

161. Operation of Cutting the Thread.—The operation of cutting the thread is accomplished by feeding the thread tool in for a light cut, starting the lathe to take the cut for the required distance, backing the tool out of the groove and reversing the lathe to bring the thread tool back to the starting point; feeding in for another light cut (not over .003 or .005)
and repeating these operations until the thread is finished. If the thread tool is allowed to remain in the part of the groove already cut when the direction of the work is reversed, the point of the thread tool will be broken, owing to the back-lash in the lead screw and split nut and also in the gearing. It is essential therefore that the thread tool be pulled out of the groove before reversing the lathe. It is a knack to turn back the cross feed screw handle and reverse the lathe at the same time, and it might be well for the beginner to practise this for a little while before attempting to cut the thread. Grasp the controller handle (or the shipper) with the right hand and the cross feed screw handle with the left hand and practise working the two at the same instant.

Fig. 169.—Centre gauge.

Fig. 170.

162. Preliminary Hints on Thread Cutting.—Grind the thread tool accurately to gauge (Center Gauge Fig. 169). Do not grind the point flat for pitches under $\frac{1}{6}$", round it slightly with an oilstone.

The cutting point of a thread tool should be exactly on center, and the cutting edges set exactly to gauge. (See Fig. 170.) Do not jam the thread tool into the V of the gauge, but leave a little space between. With the gauge against the work move one side of the little V against one cutting edge of the tool and holding a piece of paper underneath see if the light is shut out. If one side is all right try the other side, and thus check the grinding of the tool.
The friction feed is never used when cutting a thread. Be sure the feed control knob is not tightened. Having the split nut and the feed both in will break the apron when the lathe is started.

Be sure the gears are right for the pitch required. Many jobs are spoiled by carelessness in this respect. Measure the pitch after the first light cut. This may be done by counting the number for one inch or for one-half inch (Fig. 171) but a screw pitch gauge (Fig. 172) is quicker and there is less chance of error.

![Fig. 171.—Counting threads with scale.](image)

![Fig. 172.—Screw pitch gauge. This gauge will measure the threads of nuts as well as screws. It contains a series of blades accurately notched and numbered according to the measure of the thread pitches in common use. The arrangement of the blades hinged in each end of the case enables any gauge to be quickly selected and placed in position for use.](image)

Be sure that the split nut is closed tight on the lead screw. If the threads on the nut do not go into the grooves of the screw, move the carriage a trifle by hand.
See if there is plenty of clearance for the dog when the thread tool is at the end of the cut.

If the thread being cut is of steel or wrought iron, apply good lard oil or cutting compound with a brush. Do not cut dry, and do not slush the lathe.

If there is more than one slot in the face plate, mark the one used with chalk. If necessary to remove the work from the lathe be sure to put the dog in the marked slot otherwise the thread will be ruined.

Even with a sharp thread tool, a burr will form on top of the thread. This burr should be "brushed off" with a file before the finish cut is taken.

If the outside diameter is the correct size the depth of the thread may be gauged fairly close by the amount of the flat left. When approaching the finish, work carefully. It is often advisable, especially when cutting several screws, to first rough the threads. Finish with a keen tool correctly ground to 60 degrees and take light cuts.

163. The Thread Stop.—It is usually advisable for the beginner to use the thread stop (Fig. 173). The Thread Stop A is arranged on the carriage in front of the cross slide and the screw S slides freely in a hole in the stop and screws into a hole in the cross slide. Turn the screw S until it enters the hole in the cross slide ¼" or more. Set the thread tool by the gauge, and on center, and run the cross slide in until the point of the tool nearly touches the work to be threaded. Then by clamping the stop, the cross slide (and tool) cannot be fed in except as the screw S is loosened.

In order to feed the thread tool in a certain definite amount for each cut, the screw S is loosened sufficiently to allow the tool to move this amount. The feed, of course, should be less as it approaches the full width of the cut, that is, as it ap-
approaches the finish of the thread. Many machinists prefer to use only the graduations of the cross feed screw for gauging the depth of cut. It will probably save time and trouble, especially for the beginner, to use both the thread stop and the graduations.

164. Three Wire Method of Measuring Threads (Fig. 174).—A method known as the Three Wire Method of measuring screw threads offers one of the easiest and most accurate ways of determining the exact size of a thread. The three wires are of the same diameter but a definite size for a given pitch is not required; any diameter, smaller than the pitch of the thread, that will not fall below the top of the thread will answer. (See \( W \) in the figure.) Select three wires about an inch longer than the diameter of the screw. Place two wires in adjacent grooves on one side of the screw and one wire directly opposite. Snap a rubber band over the three ends of the wires on each side of the screw to hold them while the measurement with the micrometer is being made. (Some machinists prefer to hold the three wires in a small block of wood as shown in Fig. 174.)

To tell whether the thread is cut deep enough, or for the
purpose of accurately gauging the thread at any time for depth of cut, the following rule may be used:

Rule: Measuring the U. S. Std. Form of Thread.—To the diameter of the screw add three times the diameter of the wire and from the sum subtract the quotient obtained by dividing 1.5155 by the number of threads per inch. The measurement over the wires should equal this result.

Formula:

\[ D + 3W - \frac{1.5155}{N} = M \]

Example.—It is required to measure a \( \frac{3}{8} \) - 9 thread.

Solution.—First: select the wires, say .066 "dia. which is between the minimum and maximum sizes that may be used. Second: Make the necessary calculation as follows:

\[ D = \frac{3}{8} = .875 \]
\[ 3W = 3 \times .066 = .198 \]
\[ \frac{1.5155}{N} = \frac{1.5155}{9} = .168 \]

Then .875 + .198 - .168 = .905.

Third: Placing the wires over the thread, make the measurement over the wires carefully. If this measurement is .905" the thread is right. If it is greater than .905, then it must be made just that much smaller. If it is less than .905, the thread has been cut undersize.

Rule 2. Measuring the "V" Thread.—In the V thread, instead of using the constant \( \frac{1.5155}{N} \) the constant \( \frac{1.732}{N} \) is used.

Rule 3. Measuring the Whitworth Thread.—In the Whitworth form of thread the constant \( \frac{1.6008}{N} \) is used, and for "three times the diameter of the wire" in the rule, substitute "3.1657 times the diameter of the wire."

165. The Use of the Compound Rest for Cutting Threads.—The great disadvantage of the thread tool cutting on both sides of the 60° angle is the fact that it cannot have rake and cut correctly. If a thread tool which is supposed to cut an equal amount on each side of the angle is given rake from one side, the other side will automatically be given a negative rake and
will not cut. The objection to front rake is shown in Fig. 175. It will be observed that any angle of front rake decreases the angle between the cutting faces, and this angle grows smaller as the rake is increased.

If, however, the lathe is provided with a compound rest, a tool having side rake as shown in a, Fig. 176 may be used.

Grind the tool to 60°, and set it with the center gauge b, Fig. 176. Set the compound rest so that the tool may be fed in at an angle of 30° to form one side of the thread. The set-up is shown at d.

The thread stop is used as a stop only and not to gauge the depth of the cut because the tool is fed into the work by the compound rest handle. One side of the tool does all of the cutting (c Fig. 176) and may be given the desired rake. The
adjacent side, if the tool is properly ground, will just clear the other side of the thread. This method of cutting threads is recommended as being two or three times as fast as with a tool having no rake.

166. Four Ways of Catching the Thread after Resetting the Tool.—If it is necessary for any reason to remove the tool before the thread is finished, reset the tool to gauge regardless of that part of the thread already cut. If the lathe is provided with a compound rest, adjust the tool to the desired position in the groove manipulating the cross feed handle and the compound rest handle.

If the lathe is not provided with a compound rest loosen the dog and turn the work until the tool enters the groove centrally.

If this is impractical put the reverse gear handle in neutral position and revolve the work forward by hand, until the tool is exactly opposite the groove then connect the reverse gears as before.

Another way is to disengage the intermediate gear from the screw gear and revolve the spindle forward by hand until the tool will enter the groove of the thread centrally, then re-engage the intermediate.

167. To Cut a Left Hand Thread.—To cut a left hand thread it is necessary to reverse the direction of rotation of the lead screw. This causes the carriage to move toward the tailstock with a forward motion of the spindle. When cutting a L. H. thread start the cut on the end of the thread nearest the dog, (usually in a groove already turned) and cut toward the tailstock.

168. To Cut a Thread on a Taper.—When cutting a thread on a tapering piece, for example, on a pipe, the thread tool should be set square with the center line of the piece to be threaded. The taper attachment, if available, should be used, if one is not available, and the piece is provided with centers the tail-stock may be off-set to give the desired amount of taper. If the work must be held in a chuck and the lathe is not provided with a taper attachment a fairly good job can be
done by slowly feeding the tool toward the operator as the work turns.

169. To Cut an Inside Thread.—Cutting an inside thread is no different than cutting an outside thread except that the tool is moved in opposite directions; toward the operator to cut, away from the operator when reversing the lathe.

The diameter of the hole is calculated for the exact root diameter of the thread or a little larger depending on the job. If the thread does not go through, a recess should be bored at the end of the thread to prevent trouble and make a better job. The thread tool is carefully ground with special attention to the clearance. To prevent unnecessary spring the shank of the tool should be as short as permissible and as large as convenient. The thread stop should be arranged, if possible, to act in a double capacity; in one direction to limit the size of the chip, and in the opposite direction to prevent the tool being run too far back, in which event it will rub off the top of the thread.

If no tap is available the thread must be cut to size. If the piece to be fitted is heavy or awkward to handle, it will save time to make a gauge. Take careful measurement of the thread to be fitted and cut a short screw of the same diameter and pitch to use as a gauge.

170. Cutting a Thread Without the Reverse Belt.—Sometimes when cutting a long thread the time wasted in the return of the tool from the end of the thread is a serious consideration, so serious, that some shop foremen forbid the use of the reverse belt\(^1\) when thread cutting.

To cut a thread without the reverse belt, the tool is withdrawn at the end of the thread, as is the usual practice, but, instead of reversing the lathe the split nut is opened and the carriage run back by hand a certain definite distance, at which point the split nut may be closed and the operator be assured that the tool will track in the original groove. This definite

---

\(^1\) *Reverse Belt.*—Sometimes called the "back belt" and often the "cross-belt" reverses the direction of rotation of the countershaft and consequently of the machine spindle.
distance depends of course on the length of the thread but it also depends on the pitch of the thread being cut, and on the pitch of the lead screw. It may be determined as follows:

First.—If the number of threads required is the same as the number of threads on the lead screw, close the split nut at any point.

Second.—If the number of threads on the lead screw is a factor of the number of threads required, close the split nut at any point.

Third.—For all other even threads, close the split nut at any \( \frac{1}{2}'' \) distance from the stopping point, if the number of threads on the screw is even; if lead screw is odd close the split nut on any inch distance from the stopping point.

Fourth.—For all odd threads close the split nut on any inch distance from the stopping point.

Fifth.—For half threads (for example 11\( \frac{1}{2} \) threads per inch), close the split nut any 2'' distance from the stopping point.

It is advisable to mark the definite distance from the stopping point of the carriage, perhaps with a lead pencil on the ways of the lathe. Then when the carriage is run back by hand to this mark the split nut will properly engage the lead screw and the thread tool will track. This operation is often spoken of as "catching the thread."

Many lathes are now equipped with a chasing dial. (See Fig. 32, page 50.) A brass plate giving directions for using the chasing dial on the particular lathe is screwed to the lathe. The dial (being connected with a small gear which meshes into the threads of the lead screw) is caused to revolve once for every 4 inches travel of the carriage. The dial is graduated into 8 divisions, consequently each division will indicate travel of the carriage of \( \frac{1}{2}'' \) and show at a glance the time when the lead screw and carriage bear exactly the same relative positions as before at which time the split nut is closed and the thread tool will track.

171. Cutting a Square Thread.—Although the square thread tool (a, Fig. 177) looks something like a short cutting-off tool it differs in one very important respect; the blade is not
square with the bottom as in a cutting-off tool but is canted to conform to the "slant"\(^1\) of the thread. This is illustrated in b, Fig. 177. Before attempting to make a tool for cutting a square thread one should know how to determine the correct amount of slant of the tool for the given thread.

The amount the tool slants or the slant angle changes for each different lead of thread on a given diameter. The greater the lead the greater the angle.

![Diagram of thread slant](image)

**Fig. 177.**

The slant changes for each different diameter of thread of any given lead. The larger the diameter the less the slant.

In addition to the slant which the tool blade must have it must be made thinner toward the bottom otherwise it could not enter the thread at all, let alone have clearance, because a piece with parallel sides cannot fit in a curved slot, and the groove of a square thread is a curved slot. The slant of the leading side of the tool therefore must be greater than the slant of the following side of the tool (c, Fig. 177).

The amount of the slant of either side for any thread may be represented by a right triangle (d, Fig. 177). One of the right angle sides equals the lead of the thread and the other the circumference, (1) of the root of the thread for the leading side, and (2) of the outside of the thread for the following side. The number of degrees of slant is measured between the hypo-

\(^1\) The reason that the thread slants is because it is a helix, and the amount the thread slants depends on the helical angle. The slant angle is the complement of the helical angle.
then use and the side representing the circumference \((d)\) Fig. 177. (The triangle may be drawn to larger scale if desired.)

**Example.**—Find the slant of the following side and of the leading side of the blade of a tool for cutting \(1\frac{1}{4}''\)–4 square threads.

**Solution:**

- Lead equals \(.250''\).
- Outside diameter = \(1.250''\).
- Root diameter = \(1.000''\).
  
  \[1.250'' \times 3.14 = 3.92'' = \text{circumference (outside of thread)}\]
  
  \[1.000'' \times 3.14 = 3.14'' = \text{circumference (root of thread)}\]

Draw a right triangle, one right angle side equal to \(.250''\) (Lead) and the other equal to \(3.92''\) (Circumference of outside); draw the hypothenuse and measure the angle between the circumference line and the hypothenuse. It will equal \(3\frac{1}{2}^\circ\) as nearly as can be measured with a bevel protractor. This is the angle of the following side. Draw another right triangle with \(L\) (Lead) equal to \(.250\) and \(C\) (Circumference) equal to \(3.14\) (circumference at root of thread) draw the hypothenuse and measure the angle. It will equal \(4\frac{1}{2}^\circ\) as nearly as can be measured with the protractor. This is the angle of the leading side.

**Note.**—In a right-angle triangle, side opposite divided by side adjacent equals tangent of angle. In example given above:

\[.250 + 3.14 = .0796 = \tan. \text{angle } 4^\circ 33'\]
\[.250 + 3.92 = .0637 = \tan. \text{angle } 3^\circ 21'\]

In addition to the theoretical angles of clearance, slightly more clearance is given in practice (for the same reason that theoretically a cutting-off tool need have no side clearance but for practical purposes it must have side clearance or it will rub).

The operation of cutting a square thread differs in no particular respect from that of cutting a U. S. Std. thread. If the
thread is half inch pitch or greater it is usually advisable to cut it somewhat narrower with a roughing tool, before finishing. Some mechanics prefer to finish the sides of the thread with the side tool, others prefer the regular square thread tool ground to full size. In any event the object is to secure a thread with smooth sides.

172. To Cut an Acme Thread.—One of the advantages of the Acme thread is that it is easier to cut than a square thread. The Acme thread tool is ground to gauge (Fig. 178) for the pitch required. The cant of the tool is similar to that of the square thread tool to give the necessary strength. The clearance angles are substantially the same as for the square thread tool. Be particularly careful that the tool is sharp.

173. Cutting Metric Screw Threads.—A meter, the standard of length in the metric system of measurement is equal to 39.37”.

For purposes of finer measurement the meter is divided into 100 equal parts called centimeters and the centimeter into 10 equal parts called millimeters.

Therefore a millimeter is equal to .03937” ($\frac{1}{25.4}$ of a meter or $\frac{1}{1000}$ of 39.37”). Also if 39.37” (one meter) equals 1000 millimeters one inch equals 25.4 millimeters (1000 mm. divided by 39.37).

In all threads except the metric it is customary to speak of the number of threads per inch, that is, the thread of $\frac{1}{13}$” pitch for example, is thought of and spoken of not as a “$\frac{1}{13}$ pitch” thread but as “13 threads per inch.” In the metric thread, however, it is the usual practice to think and speak in terms of pitch and the pitch of the thread is given in millimeters. (See list of tables: French (Metric) Standard Threads and International Standard Threads.)
For example the 26 mm. thread (1.024” dia.) has a pitch of 3 mm. (.118”).

In order to determine the number of threads per inch of a metric thread it is necessary to divide 25.4 by the pitch in millimeters. That is, \[
\frac{25.4}{\text{Pitch in mm.}}
\]
is the number of threads per inch.

The formula for gearing a lathe to cut any desired number of threads per inch is:

\[
\frac{\text{Lead Number}}{\text{No. thds. per in. required}} = \frac{\text{Driving Gears}}{\text{Follower Gears}}
\]

Substituting \[
\frac{25.4}{\text{Pitch (mm.)}}
\]
for the “number of threads per inch required” in the above formula and

\[
\frac{\text{Lead Number}}{25.4} = \frac{\text{Driving Gears}}{\text{Follower Gears}}
\]

Pitch mm.

Suppose the lead number is 8 and the pitch is 3 mm., then:

\[
\frac{8}{25.4} = \frac{8 \div 25.4}{3} = \frac{8 \times 3}{25.4} = \frac{24}{25.4}
\]

The denominator being fractional, both terms of the fraction are multiplied by 5 (in this case) to get a whole number.

\[
\frac{24 \times 5}{25.4 \times 5} = \frac{120}{127} = \frac{\text{Driving Gear (Stud)}}{\text{Follower Gear (Screw)}}
\]

Now 127 is a prime number, it cannot be factored, so it is impossible to cut a metric thread on a lathe with an English Measure Lead Screw without a 127-tooth gear, and, further, this gear is always a driven or follower gear. It is called a “translating” gear.

In the above example 120 stud gear and 127 screw gear will serve, but few lathes are equipped with a gear of 120 teeth.

It will be necessary to compound.

\[
\frac{120}{127} = \frac{60 \times 2}{127 \times 1} = \frac{60 \times 40}{127 \times 20} = \frac{60 \text{ and } 40 \text{ (Driving Gears)}}{127 \text{ and } 20 \text{ (Follower Gears)}}
\]

Some lathes are provided with compounding gears in the ratio of \[
\frac{50}{127}
\] especially for cutting metric threads. In such
lathes the gears, on stud and screw may be figured to conform to this compounding ratio. Thus in the above example:

Factoring \( \frac{120}{127} \) to obtain 50 in the numerator will give

\[
\frac{120}{127} = \frac{50 \times 2.4}{127 \times 1} = \frac{50 \times 96}{127 \times 40} \quad \text{(driving gear)}
\]

or

\[
\frac{50 \times 48}{127 \times 20} \quad \text{(follower gear)}
\]

or

\[
\frac{50 \times 72}{127 \times 30} \quad \text{(driving gear)}
\]

or

\[
\frac{50 \times 72}{127 \times 30} \quad \text{(follower gear)}
\]

174. Multiple threads.—In a single thread the lead is equal to the pitch.

Fig. 179.

A double thread is one having two thread pitches to one lead; a triple thread has three pitches to one lead; a quadruple thread has four pitches to one lead, etc., etc. These threads are known as multiple threads and are much used in machine
construction and may be of any recognized form of thread. (See a, Fig. 179, which represents a square thread.)

Suppose it is required to operate a part of a machine by screw action; that the movement must be ¼" per revolution of the screw, and that the diameter of the screw cannot be over 1". A single cut screw of 1" dia. ¼" pitch will look like b, Fig. 179, which is not a good looking screw and moreover the cross-section at the root of the thread is proportionally weak. If a thread of the same lead but half the depth were cut it would look like c, still faulty in appearance and the nut would be very weak, in fact only half as strong as it should be.

If a thread of the same depth as in c, is cut half way between the grooves of the first thread as shown in d, the double thread will be pleasing in appearance and of the required strength. It is, of course, understood that the nut also must have the double thread.

175. Cutting a Multiple Thread.—To cut a double thread, U. S. Std. for example, proceed as if cutting a single thread of the required lead until the thread is half the depth, and the groove is half the width of a single thread of the same lead. It is then necessary to give the work exactly half a turn without turning the lead screw. This may be accomplished by having a special face plate with the slot for the tail of the dog exactly opposite the one used for the thread groove already cut.

In the absence of an accurately slotted face plate, the method used is to disengage the intermediate gear from the screw gear and move the lathe spindle (and the work) one half turn. Before disconnecting these gears bring a tooth of the stud gear exactly between two teeth of the intermediate and mark this tooth with chalk. The distance this tooth moves shows how much the spindle has moved and here is where a serious mistake is liable to occur.

If the gear on spindle and the inside stud gear are of the same number of teeth, the marked stud gear will move one-half revolution when the work is turned half around, but most lathes are now constructed with the spindle gear smaller than the inside stud gear in a ratio of 3 : 4 or 2 : 3. This means
that the stud gear will not go half around; it will go $\frac{3}{4}$ or $\frac{3}{5}$ of half around as the case may be.

Suppose the spindle gear has thirty teeth and the inside stud gear has forty teeth, the ratio is 3 : 4. Instead of the marked stud gear revolving half around it will revolve $\frac{3}{4}$ of $\frac{1}{2}$ of a revolution or $\frac{3}{8}$ revolution. For this reason it will be necessary to select a stud gear with a number of teeth divisible by 8.

Beginning with the tooth next to the marked tooth count (in the proper direction) the number of teeth necessary to show the half revolution of the spindle and mark the last one. Turn the spindle to bring that tooth into exactly the proper position with respect to the intermediate gear and engage the screw gear.

The principle of cutting triple threads and quadruple threads is the same as for cutting double threads.

Questions on Cutting a Thread in a Lathe

1. How is thread cutting accomplished in an engine lathe?
2. What is the purpose of the tumbler gears?
3. Why are there a number of change gears furnished with the lathe?
4. With equal gears on stud and screw the lathe will cut a certain number of threads per inch. This is called the lead number. The lead number may or may not be the same as the number of threads per inch on the lead screw; give the reason for this.
5. State a rule or formula in the form of a proportion that will serve for calculating change gears.
6. If the lead number of a lathe is 8, what gears are suitable to cut 12 threads per inch? 10 threads? 6 threads? 11\(\frac{1}{2}\) threads?
7. In any lathe, does it make any particular difference how many teeth the intermediate gear has? Give reason.
8. Why is the intermediate gear adjustable on the bracket? Why is the bracket adjustable?
9. What is meant by compound gearing?
10. When is compound gearing used in thread cutting?
11. With a lead number of 6, gear progression 4, what gears may be used to cut 36 threads per inch?
12. What is the object of having a flat on the top of the thread? On the bottom?
13. What is the angle of a thread tool (U. S. Std.)?
14. What gauge is used when grinding the thread tool?
15. How much clearance has a thread tool? Which side? Why?
16. Why is the point rounded slightly instead of being flattened an exact amount? How is it rounded?
17. If a thread tool is to cut on both sides of the angle, can it be given rake? Give reason.
18. Why is it wrong to pinch the gauge between the tool and the work? How should the gauge be used?
19. What is the purpose of the thread stop? How is it arranged?
20. Why is it necessary to withdraw the thread tool at the end of the thread before reversing?
21. Why do you use a lubricant? How is it applied?
22. How much of a chip is advisable when cutting a thread? Why not more?
23. How do you judge when the thread is nearly cut?
24. How do you remove the burr from the thread?
25. Which is the better method of gauging a thread, with a nut or by the "Three Wire Method"?
26. When gauging with a nut, can you tell exactly how much more you have to cut? Can you tell with the "Three Wire Method"?
27. Why do you use three wires instead of two?
28. Why is a \( \frac{3}{4} \) inch wire too small for \( \frac{3}{10} \) pitch thread? Why is \( \frac{1}{4} \) inch wire too large? How do you judge the size?
29. State the rule for measuring U. S. Standard Threads by the "Three Wire Method."
30. Why is the "Three Wire Method" particularly valuable if a special tap is to be made?
31. If for any reason the tool is removed before the thread is finished, what care must be taken when resetting it?
32. State two ways of resetting the tool central with the part of the thread already cut.
33. When resetting the tool why must the lost motion of the lead screw be taken into consideration?
34. What is meant by change gear progression in a lathe?
35. Is it advisable to use the compound rest when cutting a thread? Give reasons.
36. Explain the set-up for using the compound-rest for thread cutting.
37. Explain the way in which a left hand thread is cut in a lathe.
38. What is the advantage of the chasing dial on the apron of the lathe?
39. Is the chasing dial necessary when cutting threads without the reverse belt? Explain.
40. What gear is necessary when it is desired to cut a metric thread with a lead screw of \( \frac{4}{6} \)" pitch? Why?
41. When is a double thread used?
42. How is a double thread cut in a lathe?
CHAPTER XI

FACE PLATE WORK

A large variety of jobs that cannot be machined on centers or held in a chuck may be fastened to the large face plate and turned or bored.

It is of course essential that the work is accurately fastened or "clamped" in position on the face plate and for this purpose certain accessories are necessary. Sketches and descriptions of some of these are here given (Fig. 180), and examples showing their use follow.

176. Definitions of Accessories Used (Fig. 180). Square Head Bolt.—May be used in any of the face plate slots for clamping pieces to the face plate.

Shouldered Stud.—Threaded on each end so designed that it may be fastened in any desired position on the face plate.

1Large Face Plate.—Screws on the threaded nose of the spindle: is as large as will conveniently swing over the ways of the lathe; faced flat and true; has, usually, four or more radial T-slots and several shorter slots which go through the plate.
"U" Clamp.—Used with either a bolt or a stud in clamping the work, is easily adjustable, and is light and strong.

Parallel Strip.—Made with at least two adjacent sides straight and square; is provided with slots and either straight or tapped holes as shown in the figure, so that it may be easily clamped by bolts or screws in any desired position on the face plate.

Angle Plate.—Made in various sizes usually with an angle of 90° between the finished faces, but may have any desired angle; provided with the necessary holes for clamping purposes; is used for a large variety of jobs in face plate work.

**Fig. 181.** (a) Center Tester.—"Wiggler." Used for locating centrally a prick punch mark or a hole in work held in the chuck or on a face plate. The amount and direction the work runs out of true is indicated by the long arm of the needle and the amount is magnified as many times as this arm is longer than the other. (b) Dial test indicator. For general description see page 106.

Stop Block.—A small piece of iron or steel, with a hole through it, so that it may be bolted securely to the face plate forming a positive stop in re-locating a piece, or locating several duplicate pieces in the same position on the face plate.

End Measuring Rod.—Sometimes called pin gauge, made from a piece of drill rod or similar material in any desired length; rounded sufficiently on the ends to offer a point contact and used for re-locating work at certain definite distances from the stop block.
Parallel.—A standard shop tool, opposite sides parallel, adjacent sides square, very useful in face plate work when used in connection with the parallel strip.

Indicator.—There are several different forms of universal indicators by means of which work may be accurately located either from a cylindrical projection on the work, or a hole in the work, or a prick punch mark in the work. (See Fig. 181, also Fig. 91, p. 106.)

Weights for Counter Balance.—Very necessary in face plate work. Any piece of iron or steel that may be picked up in the shop that is of sufficient weight to counter balance the work when fastened to the face plate may be used.

177. Typical Face Plate Set-ups.—Fig. 182 shows a flat piece of cast iron clamped to the face plate located by means of a parallel strip and stop block. This piece has been indicated to drill and bore a hole at one end. With the work in the position shown it would be out of balance were it not for the weight used as a counter balance.

Fig. 183 shows this same piece of iron moved along on the parallel strip and re-located by means of this strip and the gauge between it and the stop block. With the proper number of lengths of gauges a series of holes at any distance apart may be bored in such a piece, and by using a parallel strip for locating the edge, these holes will be in line.

Fig. 184 shows the use of a parallel by means of which two rows of holes may be drilled in parallel lines; the weights,
clamps, etc., are not shown. These operations are used in tool-making, that is, in making jigs, fixtures, gauges, etc. ¹

Sometimes when it is required to hold on the face plate and machine a number of duplicate pieces the face plate may be set up as a temporary fixture by using parallel strips, or angle irons, or stop blocks, or possibly all three, together of course with the necessary clamps. Fig. 185 shows a typical face plate job of this kind. The hole in the work must be bored at an angle of 65° with the finished flat surface. The special block A is planed at an angle of 25° (complement of 65°) to give the required seat for the work. After the block is once set it is not disturbed and duplicate pieces may be quickly and accurately located and machined.

Fig. 186 will give an idea of how, in a number of duplicate pieces, the hole may be bored accurately, a certain distance from a finished surface, by locating each piece with the finished surface against a parallel strip, and then clamping to the face plate as shown.

Fig. 187 shows the value of the angle plate in face plate work. The angle plate must be square and true. It is usually held against the face plate by screws coming through from the back of the face plate and screwing into tapped holes in the angle.

¹ Note.—Instead of using the parallel as shown “size blocks” may be used. Size blocks are rectangular pieces of hardened steel with two opposite sides ground and lapped to an exact dimension. Size blocks may be used also instead of the pin gauge for spacing the holes. It will be understood that these gauges are used only where extreme accuracy is necessary.
plate, though often straight holes are drilled through the angle plate and it may then be bolted fast to the face plate. It is often necessary in order to get the angle plate in the position desired on the face plate, to drill new holes. In such a case a certain amount of common sense should be exercised in regard to the position of these new holes.

![Diagram](image)

**Fig. 186.**

**Fig. 187.**

178. **Hints on Face Plate Work.**—Be sure that both the shoulder on the spindle and the face of the hub of the plate are free from burrs or nicks, and that both threads are clean.

The face plate should screw freely on the spindle and tight against the shoulder. It should not be forced too hard against the shoulder or it will jam. If a chuck or a face plate is run against the shoulder with a bang, it makes removal difficult.

The face plate should run perfectly true and if advisable may be tested with an indicator.

After the piece is clamped try every screw, and every nut, to make sure that each is sufficiently tight.

See that the face plate is free to turn, that no bolts or clamps project in any way that will come in contact with either the carriage, the ways, or the headstock. Turn once around by hand to make sure.

A piece of paper between two flat surfaces will reduce the tendency to slip. This is true in planer work, or shaper work, or boring mill work, and especially true in face plate work, where facilities for clamping are not always of the best.

Remember in clamping, that it is the work that is to be clamped and not the blocking under the other end of the clamp.
Use the dead center to hold the work against the face plate while clamping. If the work has a large hole in it a piece of flat stock slightly larger than the hole may be used between the work and the dead center.

The work and necessary counter-weights, etc., may often be more easily clamped to the plate when it is lying in a horizontal position on the bench. The clamps may be tightened sufficiently to hold the work, after which the face plate is mounted in its place, and then the work carefully adjusted to the desired position and made fast.

Use bolts long enough to obtain the full strength of the nut otherwise the thread of both will be strained and may be spoiled.

Avoid using bolts that are much too long; it is dangerous. If other bolts are not available put all excess of length possible back of the face plate.

Questions on Face Plate Work

1. What precautions are taken when the large face plate is put on the spindle?
2. What operations may be done on work which is fastened to the face plate?
3. What tools are used to fasten work to the face plate?
4. What is a test indicator?
5. What weight is often necessary when work is fastened to the face plate? Why?
6. Why should a piece of paper between the work and the face plate serve to help hold the work?
7. How is a clamp arranged so that the work is held most securely?
8. What is an angle plate? A parallel? A shouldered stud?
9. How is a pin gauge made? Why is it rounded on the ends?
10. For what kind of work is the face plate valuable?
BENCH WORK

AND

WORK AT THE FORGE
CHAPTER XII

HAMMERS, SCREW-DRIVERS, WRENCHES, HACK SAWS

179. The Use of Hand Tools.—There are many operations in machine shop work which involve the use of tools that are controlled by hand. The term bench work is used in the operations incident to the processes of laying out, fitting, assembling, etc. when the work is placed on the bench or in a bench vise, and the term "floor work" applies to the larger work which is erected on the floor of the shop. The same tools—hammers, wrenches, cutting tools, measuring tools, etc.—are used for either.

An expert machinist is skillful in the use of the hand tools of the craft. Being skillful is the opposite of being awkward or clumsy; it is the opposite of being ignorant or stupid; it is the opposite of being careless or indifferent. You would not use a tack hammer to drive a spike; why should you use a fine file for roughing stock? You would not use a sledge hammer to drive a tack, why should you use a 12" monkey wrench on a 1/4" bolt? You do not mistake a chisel for a screw-driver, why should you use a screw-driver with the end shaped like a dull chisel?

Skill means the knowledge of the proper tool to use and how to use it in the right way; it means more than this—it means a positive unwillingness to use a tool that is not right. The real machinist is proud of his kit of tools, proud of the work he can accomplish with them. Manual skill must be acquired through practice. Information regarding the proper use of hand tools can, however, be obtained by reading and by observation.

Bench work involves the use of hammers, screw-drivers, wrenches, hack saws, chisels, files, scrapers, taps, threading
dies, small drills (by means of "hand drills" or "breast drills"), hand reamers, taper reamers and taper pin reamers. In addition most of the machine shop measuring tools and gauges are used in bench work. Many of these tools and gauges have been described elsewhere in this book. Also the use of taps, dies and hand operated reamers while perhaps belonging particularly to bench or floor work have been previously described and descriptions of these tools will not be repeated here.

180. Hammers.—Machinists' hammers are made of steel, hardened and tempered. They are made in different sizes (weights) from 6 oz. to 2½ pounds; those weighing over 1½ pounds are not much used. The top of the hammer head is called the peen and the bottom is called the face. The illustration a, Fig. 188, shows a hammer with a "ball" peen which

![Fig. 188](image)

is the common form of machinist's hammer, although the straight peen b and the cross peen c are much used for swaging and riveting. The eye of a hammer head is somewhat smaller in the middle than at the ends. If the end of the handle is fitted to fill one end of the eye, and wedged with soft steel wedges to fill the other end, it will be tight and secure in the hammer head. A hammer with a loose head is dangerous. The handle is set about square with the head and should be of such a shape and length as to give the proper grip and balance when in use.

If one is using a sledge it is much more effective to take it easy when lifting the sledge and put the snak in the blow.
The same is true in all hammering; a solid snappy blow is more effective and less tiresome. Such a blow cannot be delivered if the handle is grasped too near the head or is grasped too tightly.

A hammer blow is a terrific force when applied carelessly. Remember that cast iron is brittle and breaks easily. Do not use a heavy blow on a weak section or on an unsupported projection. Use care when driving a taper pin or key. Remember that the work is softer than the hammer and if the hammer slips and strikes a finished surface the dent made cannot be removed.

_Soft Hammers._—Hammers made of lead or soft Babbitt are used instead of steel hammers to seat work in a machine vise, to drive a mandrel or arbor, or in any similar operation where the steel hammer might injure the work. A piece of lead more or less spherical in shape about the size of a base ball makes an excellent soft hammer.

181. _Screw-drivers._—The screw-driver blade is made of tool steel of the size and length desired. Those under ½" cross section are usually made of round stock while the larger sizes are better if made square so that a wrench may be applied if necessary. One end of the blade is forged to provide a suitable tang for the handle, and the other end is drawn out and flattened to fit the slot of the screw. This end of the screw-driver is hardened and the temper drawn to a purple verging into a blue (about 550°F.).

It is safe to say that most screw drivers are incorrectly shaped, Fig. 189, the flat sides taper more or less all the way to the end and consequently do not bear against the parallel sides of the screw slot. The reason for making screw drivers in this way is no doubt to produce a tool that will work in a variety of widths of screw slots. If this is true, it is too often carried to extremes with the result that most of the used screw
slots are mutilated. According to the principle of the action of an inclined plane a "wedge shaped" point on a screw-driver will tend to come out of the slot when turned. If considerable twisting force is exerted and not enough end pressure, the screw-driver will "jump" out of the slot and throw up an ugly burr. This spoils the appearance of the screw and irritates whoever is unlucky enough to rub his finger across it. A correctly made screw-driver is one of the little things that stamp a real mechanic.

Fig. 190.—Double-end offset screw-driver.

A double-end offset screw-driver, Fig. 190, is used for driving screws that cannot be reached with a straight screw-driver.

Questions on the Use of Screw-drivers

1. What happens if considerable force is exerted and the screw-driver jumps out of the slot?
2. Is the slot in a screw made with parallel sides or is it V-shaped?
3. Does a dull cold chisel make a good screw-driver? Give reason.
4. If the screw-driver blade is ground to look like a dull cold chisel will it work well?
5. How is a burr thrown up on the side of the screw slot? Does filing it off fix the slot? Is it fair to leave it on? What should be done?
6. Is there any real reason for applying two forces (holding down and turning) when using a screw-driver?
7. Give three reasons why a screw-driver blade should have parallel sides.
8. Of what kind of steel is a screw-driver made? Why?
9. How is the tang end shaped to hold fast in the handle? What other methods of holding may be used?
10. Why are the larger sizes of screw-drivers often made with square blades (or bodies)?
11. How is a screw-driver hardened?
12. What care must be taken when heating? Why?
13. How far back is the blade hardened?
14. What is the proper temper color for a screw-driver? Is it harder or softer than a file? Why?
15. What is a double-end offset screw-driver? When is it used?

182. Wrenches.—Wrenches are made in many different forms for turning (twisting) bolts, nuts, pipes, taps, etc. Any form of wrench consists of a handle (or handles) with jaws, lugs or an opening or socket to fit the object to be turned. They are named (1) from their shape; as “S” wrench, angle wrench, etc. (2) From the object on which they are used; as tap wrench, pipe wrench, etc. (3) From their construction; as spanner wrench, ratchet wrench, etc. Several of the wrenches commonly used in machine work are illustrated in Fig. 191.

183. A Few Suggestions Regarding the Use of Wrenches.—The wrench should fit, otherwise the corners of the bolt or nut or tap to be turned will be rounded. The wrench is a lever and the mechanical advantage is of course in proportion to the length of the handle. The handle of a solid wrench is usually made to give about all the leverage the part to be turned will stand without injury. In an adjustable wrench this cannot be the case. When a large monkey wrench is used on a small bolt, or when a large tap wrench is used on a small tap or reamer, considerable care and judgment must be used. With the extra leverage the workman is not as sensitive to the resistance of the bolt or tap and may turn too hard and break it.

Do not use a wrench with an opening too large. When using a monkey wrench adjust the sliding jaw until it is tight on the nut.

When using a monkey wrench, have the jaws pointing in the direction the force is to be applied. There are two reasons for this; the wrench is less apt to be sprung, and it is less liable to slip off the nut or bolt.

A quick jerk when tightening, and a blow with the ball of the hand when loosening a bolt or nut, is more effective than a sustained pull or push, because momentum is a factor.
Fig. 191.
It is distinctly not good practice to use a wrench for a hammer or to use a hammer on a wrench. It is good practice to oil the thread of a bolt or nut and occasionally to oil the screw of a monkey wrench.

Sometimes on heavy work when one is sure the wrench and also the part to be turned will stand the strain, it is allowable to extend the leverage and increase the mechanical advantage by putting a suitable piece of pipe over the wrench handle. Care and judgment must be used, however, or the wrench will be sprung and possibly the bolt will be twisted off, or the tap or the reamer, as the case may be, will be broken.

Oftentimes bolt heads or nuts are so arranged in a machine that a movement of the wrench through 90° (for a square nut) or through 60° (for a hex. nut) is impossible owing to obstructions. This difficulty is overcome by using for the square nut a 22½° angle wrench and for a hex. nut a 15° angle wrench. By turning the wrench over each time it is applied the nut may be turned completely around when the swing of the handle is limited to substantially half

that required for a straight wrench. This is illustrated in Fig. 192.

The ratchet wrench, Fig. 193, is especially useful when only a short swing of the handle is permissible; in fact the
multiple ratchet wrench may be used with a swing of only 10 degrees. An added advantage of the ratchet wrench lies in the fact that it is not necessary to remove it until the bolt or nut is tight.

184. Action of Check Nut.—The principle underlying the proper action of a check nut should be understood. The function of the check nut is to make more certain that the holding nut will not loosen, that it will stay where it is put. The check nut and the holding nut should be arranged so they bear on opposite sides of the screw thread and thus produce the effect of an extremely tight nut. It is not enough to screw the one nut down against the other. In order to make the two nuts bear on opposite sides of the thread it is necessary to use a wrench on each nut and to turn the first nut back a little as the second is turned down. If the first nut is a fairly close fit on the screw only a small fraction of a turn is necessary. It will be understood from the foregoing that the check nut is the first to be put on, and the second is the holding nut since it bears against the side of the thread that takes the reaction of the part being held. Therefore, if two nuts of unequal thickness are used the thicker one should be put on last in order to obtain the value of its greater strength.

Questions on the Use of Wrenches

1. What is the purpose of the 15° offset wrench? Of the 22½° offset wrench?
2. Explain the use of each of the following wrenches: single-end wrench, double-end wrench, closed-end wrench, spanner wrench, socket wrench, ratchet wrench, pipe wrench, monkey wrench, chuck key, hollow set-screw key, tap wrench.
3. What is the value of a socket wrench?
4. What is a lever? Is a wrench a lever?
5. Why are wrench handles made so short?
6. Should a 12” monkey wrench be used on a ½” cap screw? Give reasons. What do you mean by “sensitive”?
7. If too much force is applied to a wrench, what is liable to happen?
8. What are the two distinct disadvantages of a monkey wrench in the hands of an inexperienced person?
9. Which way should a monkey wrench be applied? Why?
10. What causes the corners of bolts and nuts to become rounded? What does it indicate on the part of the workman?
11. Is a push or a blow more effective in loosening a bolt or a nut? Why?
12. Explain the principle of the action of a chuck nut.
14. Why is a check nut sometimes used in conjunction with a set screw?
15. When should oil or grease be applied to a stud or a machine screw?

185. The Hack Saw.—The hack saw is a metal cutting saw. It is one of the most useful and probably one of the least understood and least appreciated and therefore most abused tools in the shop. The blades are thin and narrow and vary in length by inches from 6” to 16”. (Some makers will furnish them up to 40” long.) Those used for hand sawing are usually about ½” wide, .025” thick and 8”, 10”, or 12” long, as desired. The power saw blades are usually somewhat wider (¾”) and a trifle thicker (.032”).

Hack saw blades are made from a high grade of steel scientifically hardened and tempered. They must be very hard and are consequently very brittle. Some manufacturers harden the teeth only, leaving the back soft (flexible back blades) which renders them less liable to break.

186. Proper Number of Teeth.—If only a few teeth are broken the saw is ruined. One of the chief causes of breakage of saw blades is due to teeth unsuited to the work. Most shops buy “hack saws” without specifying the number of teeth per inch or the work on which the saws are to be used, and the dealer furnishes a saw with a medium number of teeth (18 or 20 teeth per inch). This saw is all right for solid pieces of steel and cast-iron, but is not right for cutting soft materials or tubing. Low carbon (machine) steel bars as small as 1” in diameter may be efficiently cut in a power saw with a heavy blade of 14 teeth per inch; if, however, it is attempted to cut pipe or thin stock or small bars with such a saw the teeth will catch and break. The following saws are recommended by manufacturers for the purposes noted.
Power blades for cutting solids in soft steel—14 teeth per inch.
Hand blades for cutting solids in soft steel—16 teeth per inch.
For general use in hand frames—18 teeth per inch.
High-carbon steel (tool steel) and cast-iron—20 teeth per inch.
Tubing, brass, copper, drill rod—24 teeth per inch.
Thin sheet metal and thin tubing—32 teeth per inch.

187. Hack-saw Frames are made in various patterns, either in fixed lengths or adjustable, to take 8", 10" or 12" blades. The blade is fastened in the frame to cut on the forward stroke and should be given considerable tension when in use; taut enough so it cannot buckle and thus be liable to bend or break under the pressure of the stroke. On the other hand, do not strain the blade so much that canting the frame ever so little will break it.

It is easier to cut down than it is to cut sideways or up, and the usual manner of holding the frame (and the blade) is illustrated in Fig. 194. Sometimes, however, it is practicable to hold the frame flat, as, for example when a long strip is to be cut from a sheet of metal. In such a case, the clips which hold the blade may be given a quarter turn thus setting the blade at right angles to its normal position in the frame. With the blade so arranged a strip of any desired length may be cut,
provided it is no wider than the distance from the blade to the back of the frame.

188. Hints on Hack Sawing.—The tendency is to cut too fast with a hack saw. Fifty or sixty strokes per minute is right for average work. The forward stroke is the cutting stroke, pressure should be relieved on the return stroke.

The amount of pressure necessary depends on the kind of material, the width of the cut, and the condition of the blade. If, for example, a fairly thick piece of machine steel is to be cut, considerable pressure will be necessary to make the teeth "bite;" if the same pressure is applied on a narrow piece, or on soft material such as copper, the teeth biting too deeply will catch and probably break. The same reasoning applies when starting a cut on a corner, or on a small rod, or any thin section. Judgment must be used.

Be careful as the finish of the cut is approached or the teeth will dig in the thin section and break.

If a saw-blade breaks when the cut is only partly finished, start the new blade in another place on the bar. This is especially important when using a power saw. The "set" of the teeth of an old blade is slightly worn and the cut is narrower than the new blade, consequently the new blade will bind if it is attempted to continue this cut.

Keep the saw cut straight; when the cut runs, the blade is cramped and will probably break. If it starts to run, give the bar about a quarter turn and begin a new cut; the first cut will help to keep the second one straight.

Hold the work securely, otherwise it may loosen under the pressure of the cut and the blade will be broken.

Never use oil as a hack-saw lubricant. A lubricant is unnecessary when hand sawing. In high-speed power sawing where the friction of the blade on the work will tend to heat the blade enough to spoil the temper, water is used to keep it cool. In such a machine provision is made for water to drip on the saw.

It is good practice, for the beginner at any rate, to make a small nick with the edge of a file to start the saw cut.

Fairly thin sheet metal may be neatly sawed if clamped be-
tween two pieces of board. Saw the boards and metal together; this will serve to steady the blade and keep it from digging so easily. If the teeth are too coarse, a blade somewhat worn is better for sawing thin metal than a new saw.

Hack-saw blades of four different thicknesses (about .049", .065", .083" and .109", see Fig. 195) are made by several manufacturers for slotting screws and similar work. Such a blade is very handy indeed when a few screws for a special job are needed in a hurry since it saves the time of setting up a machine even if one is available.

Questions on Hack Saws

1. If you are to cut out a strip say 6" long and 1" wide, from a sheet of brass, how will you hold the blade in the frame? Why?
2. When using a hack saw, which stroke is the cutting stroke? Which way should the saw be placed in the frame? Does the saw cut on the return stroke? Give reason.
3. What is a fair cutting speed for a hack saw?
4. When are blades with 16 teeth to the inch used? With 20 teeth to the inch? With 32 teeth per inch?
5. What is the effect if one or two teeth are broken out of a hack saw blade? Why?
6. How tightly should a hack saw blade be strung in a frame? Why?
7. Why will a hack saw break if the frame is tipped sideways? If it is not pushed straight? If pushed too hard?
8. When cutting off a piece of flat stock say \( \frac{1}{4} \times 1" \), how should it be held in the vise? Why?
9. Why is a hack-saw blade harder than a wood saw blade?
10. What makes the hack saw blade more brittle than a wood saw?
11. State three common faults in the use of hack saws any one of which may result in breaking the blade.
CHAPTER XIII

LAYING-OUT

189. Laying-out is the shop term used to include the marking or “scribing” of center points, circles, arcs, or straight lines upon metal surfaces, either curved or flat, for the guidance of the workman. It is much used in drill-press work and in shaper and planer work. The layout to be worth anything must be right for the job at hand, therefore the dimensions on the blue print must be carefully followed and the layout lines made sharp and distinct. The degree of accuracy necessary depends on the job; a great amount of layout work is done on rough castings and it is not to be expected that the dimensions will be in thousandths of an inch. On the other hand it often requires more calculation and judgment—head work—to lay out a casting so that it will machine to size (especially if it is scant here or there) than it does to lay out accurately two or three holes on a finished surface.

190. Tools Used for Laying-out Work. Fig. 196.

1. Bench Plate, or Surface Plate.—A cast iron plate of convenient size, ribbed to give strength and staying qualities, machined on top and bottom, and usually on sides and ends. Used when laying-out work as a base upon which to rest the work, gages and other tools. (A surface plate may be differentiated from a bench plate in that its top surface is scraped flat. It is an expensive tool, especially in the larger sizes, and is used for testing work, gages, etc.) Laying-out plates (floor work) are often 4 or 5 feet wide and 8 or 10 feet long.

2. Hammer.

3. Center Punch.

4. Prick Punch. Similar to center punch except point is much sharper, angle being about 30° while the center punch is ground about 90°.
5. **Scriber.** A slender piece of tool steel 8" or 10" long, sharp pointed both ends, one end bent to approximately a right angle, points hardened and tempered, used for marking or scribing on metal.

6. **Divider.** A tool with hardened steel points used for scribing circles or laying off distances. It is adjustable and is classified as to size by the maximum opening between the points.

To set the divider place one point in a convenient graduation line on a scale (for example the 1" line) and adjust the other point until it exactly splits the graduation line the correct distance away. Since the graduation lines are V shaped a divider may often be set more nearly exact by feeling than by seeing. Adjust it until no "give" can be felt either side of the V.

7. **Hermaphrodite caliper (morphy).**
8. **Scale.**
9. **Bevel protractor.**
11. Surface gauge.
12. Square.

Height Gauge. Fig. 197.—Used for obtaining the height of projections from a plane surface and for locating and scribing dimensional lines. The upright bar is graduated to read, by means of a vernier\(^1\) scale on the movable jaw, to thousandths of an inch. The fixed jaw forms a base. The sharp point of the extension of the movable jaw may be used to scribe lines on surfaces that have had the scale removed.

Trammel Points. Fig. 198.—Practically a beam compass or divider used for drawing arcs or circles having a large radius.

191. Scribing the Lines.—The process of laying-out work calls for intelligent reasoning on the part of the workman.

\(^1\)For directions for reading vernier see page 284.
It is impossible to give but few rules or directions since each job is practically a problem in itself. Laying-out is line work. The surface to be machined is indicated by a line either straight or curved, and the centers of holes are located by the intersections of lines. A double line, or a blurred line is worse than useless; have the point of the scribing tool sharp and draw one line. Chalk well rubbed into the surface of a casting will help to make the lines more distinct. White lead mixed with turpentine is quicker and better for the larger work. To make the lines more distinct on a finished surface apply blue vitrol solution sometimes called "copper solution" with a piece of waste. To make blue vitrol solution dissolve a small handful of copper sulphate (blue vitrol crystals) in a half pint of water and add a few drops of nitric acid. A cream jar or a pickle bottle having a good sized neck makes a good container. The die-maker frequently heats the steel for the die or punch until it is blue. In this oxidized surface the finest line is clearly visible.

192. The Operation of Laying-out.—It is usually advisable to work from a given surface or "seat," and if several lines are to be scribed a base line, to which the other lines may be referred, should be drawn. The base line may be valuable later to re-level or square up by if it is necessary to move the piece during the layout. If a bench plate is used as a seat for the work and the tools, the surface gauge may be adjusted to a scale held vertically. By carefully adjusting the point of the surface gauge scriber to the dimensions given on the drawing, parallel lines of the required distances apart may be easily scribed on the work. If greater accuracy is required a height gauge may be used.

The intersections on these lines may then be laid off by scale measurement from a finished surface, from a previously located square or angle plate, or by means of a divider from a given point. Angular lines may be laid off by using a bevel protractor.

If it is not convenient to seat the work directly on the bench plate it may be supported on parallels placed on the plate or
it may be clamped to an angle plate and the lines scribed as suggested above.

In drill press work the centers of the holes are located by the intersections of lines. A light indentation is made at the point of intersection with a prick punch, and using this mark as a center, a circle the size of the hole required is scribed with a divider. After the circles are scribed, make a deeper indentation with the center punch to make it easier to start the drill central.

In layout work if the intersection of lines comes in an opening, as for example in a cored hole or between projections of a casting, it is customary to bridge across the opening with soft wood. A small piece of sheet copper or tin with corners bent down to drive into the wood is used as a surface upon which to scribe the lines and make the slight indentation for the divider point.

Oftentimes in cored work, the core may move a little in the mold which will cause the hole to be out of center. Such a casting may often be saved by compensating on some other surface for the error. In such a case a preliminary layout may be advisable to determine if this surface will clean.

If there is a likelihood of the lines on the rough surface of a casting becoming obliterated, it is good practice to make a series of light center punch marks $\frac{1}{4}$" or so apart along the line. This is not done on finished surfaces.

It will be well for the beginner to check his layout to be sure it is right.

Oftentimes it is necessary to lay off a number of equally spaced holes in a given circumference. The following table of chords (distances between centers of holes) for a circle having a diameter of 1" may be used by multiplying the constant for the required number of spaces by the diameter of the given circle.

<table>
<thead>
<tr>
<th>Number of divisions in circle</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
<th>10</th>
<th>11</th>
<th>12</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length of chord...</td>
<td>.866</td>
<td>.707</td>
<td>.588</td>
<td>.500</td>
<td>.434</td>
<td>.383</td>
<td>.343</td>
<td>.309</td>
<td>.282</td>
<td>.259</td>
</tr>
<tr>
<td>diam. of circle 1&quot;...</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Questions on Laying-out

1. What is chalk used for in layout work? When is "whiting" used?
2. What solution is used on surfaces which have been machined, to make the lines show more distinctly? How is it made?
3. What is the difference between a prick punch and a center punch?
4. What do you mean by a light indentation with a prick punch? For what purpose is a light indentation used?
5. Why is a divider set by feel more accurate than when set by sight?
6. What tool is used for a heavy indentation? Why? When is a heavy indentation made? Give reasons.
7. Why is a series of prick punch marks sometimes made to show the location of the circle? When are they used in other line work?
8. On finished work is this necessary? Is it advisable? Give reasons.
9. How are lines parallel to each other or parallel to a base usually scribed?
10. How are lines at an angle to a given line or base usually scribed?
11. How are intersections accurately laid off from a finished surface? From an established point?
12. What do you mean by checking the layout? When is it advisable?
13. Explain the use of the following tools used in layout work: Scale, Scriber, Hermaphrodite Caliper, Divider, Surface Gauge, Prick Punch, Center Punch.
14. Explain the use of the following tools used in layout work: Bench Plate, Angle Plate, Parallels, Parallel Clamps, C Clamps, Square, Bevel Protractor, Height Gauge.
15. What is the distance across the flats of the largest square that can be filed on the end of a cylinder 1" in diameter? 2" in diameter?
16. To what dimension should a divider be set to space equally ten holes in a 5" circle? Five holes in a 10" circle?
CHAPTER XIV

CHIPPING, FILING, SCRAPING

193. Chipping.—There is a considerable satisfaction in being able to hold a cold chisel and strike it with a hammer in such a manner as to produce a surface that indicates real workmanship. Most anyone thinks he knows how to use a hammer, and yet, excepting the screw-driver, there is probably no tool in the shop that is more ignorantly or carelessly used.

The shaper, the planer, or the milling machine may usually be regarded as more efficient than the hammer and chisel for the removal of metal, but there are many times when the hammer and chisel are invaluable. There never has been a real machine shop without a cold chisel or a real machinist who did not know how to grind and use one.

194. Cold Chisels.—Cold chisels are made in several shapes. (See Fig. 199.) The flat chisel is used for chipping flat surfaces, and often for cutting thin sheet metal. The cape chisel is forged to give the greatest strength possible with a narrow cutting edge. It is used for chipping grooves and often holes or slots where a sturdy narrow chisel is needed. The diamond point chisel is used for cutting V-shaped grooves or for chipping in square corners. The round nose chisel and the "gouge chisel" are used for such work as roughing out small convex surfaces or filleted corners and for chipping oil grooves, etc. The gouge chisel is used also for drawing a drill to center.

Chisels are made of octagonal shaped carbon steel (chisel steel) and are classified as to size according to the cross section of the steel. They are carefully forged and should be annealed before being hardened and tempered. Annealing serves to improve the grain of the steel and makes the chisel

235
stronger. The temper of a cold chisel is drawn somewhat lower than a lathe tool or a drill because it must withstand more of a shock. Only the cutting end is hardened, and this end only half or three quarters of an inch back from the edge. The proper temper color is purple merging into blue (about 530°–540°F.)

The cutting angle of a cold chisel is about 70°, and in flat chisels and cape chisels this angle is included symmetrically between two “facets.” Back of the cutting edge the facets should be straight, but across the width of the chisel they may be a trifle curved to give the slightly convex cutting edge which many machinists prefer. The other chisels have only one facet.

195. Grinding a Cold Chisel.—Much better control of the chisel is obtained if it is held as shown in Fig. 200 with the left hand resting on the tool rest. Do not hold the chisel too hard against the wheel or the temper will be lost. A flatter and better facet will result if the chisel is held slightly canted and moved slowly back and forth across the face of the wheel, especially if the wheel is at all worn. As the tendency is to
tip the chisel and thus grind more off one end of the facet, the
beginner should frequently examine the facet in order to
correct this fault. Efficient chipping cannot be accomplished
with a chisel on which the facet curves from the cutting edge
back; grind it straight. Most beginners will grind the chisel
with too sharp a cutting angle; remember that 70° is nearer a
right angle than it is half a right angle.

Fig. 200.—Proper way to hold a cold chisel when grinding.

196. The Operation of Chipping.—The first direction is to
avoid gripping the chisel or hammer too tightly. The boy
on the New England farm acquired the art of holding a plow
after his hands became so tired he could no longer grip the
plow handles, and the same principle applies to chipping.
Grasp the hammer handle well back toward the end; don’t
"choke the hammer." Swing the hammer with an easy
forearm movement vertically over the shoulder, but hit the
chisel with a solid snappy blow. Depending upon conditions,
and often upon choice, the chisel may be held with the hand
over or under as desired. The surface of the facet bearing
on the work is the guide and should be kept parallel to the
surface desired. If the chisel is held too high it will "dig," if
held too low the cut will "run away from the line." With
close attention to the line and to the chisel edge (never look
at the head of the chisel when chipping) one will very soon auto-
matically raise or lower the chisel as needed, and the knack of chipping is acquired.

197. Hints on Chipping.—Instead of keeping the edge of the chisel constantly against the chip, most mechanics prefer occasionally to draw it back \( \frac{3}{8} \) or so, say every two or three blows. This eases the hand, gives better control and a better cut. Of course the chisel edge is again in place before the next blow falls.

Do not take too deep a cut, \( \frac{1}{16} \) or \( \frac{3}{32} \) is enough. The chisel is less liable to break, the cutting edge will stand up longer, and more metal will be removed in a given time. Leave at least \( \frac{1}{32} \) for the finishing cut to give the chisel a chance to "bite." When finishing be sure the cutting edge is sharp, ease up on the force of the blows but keep them snappy.

If the surface to be chipped is fairly wide, use a cape chisel first to cut shallow grooves, and then the flat chisel to remove the stock between the grooves.

When chipping wrought iron or copper, occasionally lubricate the chisel with oil or soapy water.

When chipping cast metal it is best to begin at the ends and chip toward the middle because the force of the blow against the corner is liable to break the corner off below the finish line. And also as a second cut approaches the first cut it is good practice to ease up on the force of the blow in order not to break out a chunk of metal.

Always chip toward the solid vise jaw, if possible.

When holding the work in a vise put a packing block of wood or metal under it to keep it from working down in the vise.

Do not hammer the vise handle.

Look at the guiding line on the surface to be chipped otherwise you will pound your thumb.

Use a light hammer for a small chisel and a heavier hammer (about one pound) for ordinary chipping with a \( \frac{3}{4} \) chisel.

Use protecting pieces of brass or copper or similar soft material between the work and the vise jaws when the work has finished surfaces.
Questions on Chipping

1. Examine the cutting edge of a properly ground cold chisel. Is the cutting edge straight? Are the facets ground flat?

2. Test the hardness of the chisel near the cutting edge with a file. Test it 3/4 of an inch or so back from the cutting edge. Which part is harder? Is it as hard as the file?

3. How far back is a cold chisel hardened?

4. What is the proper temper color for a cold chisel? How does the temper of a cold chisel differ from the temper of a lathe tool?

5. If the cutting edge of a chisel is heated until it is red hot, or even blue, has the temper been destroyed?

6. Why will not grinding off the color restore the temper?

7. After a chisel has been sharpened several times, is it still as hard as ever on the cutting edge? Give reason.

8. How is the chisel held against the grinding wheel? Why not rest the chisel on the tool rest?

9. When the face of the grinding wheel is grooved, how may the facets of the chisel be ground flat?

10. Why is a wet grinder better for grinding tools than a dry grinder?

11. What is the best cutting angle for a cold chisel? Why not 90 degrees? Why not 50 degrees?

12. When grinding a cold chisel, or any other cutting tool, what precaution must be taken regarding the temper?

13. There is always a tendency for the beginner to grind one end of the facet wider than the other end. How do you account for this? How do you correct it?

14. If no grinder is available, how may a cold chisel be sharpened?

15. How dull should a cold chisel become before it is proper to sharpen it?

16. What is a flat chisel? What is a cape chisel? What is a gouge chisel?

17. About what weight of hammer should be used for chipping?

18. Where and how should the hammer handle be grasped? Why?

19. What is the proper position at the vise for chipping? Should the workman raise the hammer over his shoulder or toward his side?

20. How should a chisel be grasped? How tightly?

21. Should the workman look at the cutting edge or the head of the chisel when chipping? Why?

22. At about what angle with the surface being chipped should the chisel be held? Why not a greater angle? Why not a less angle?

23. How do you prevent marring the work when holding it tightly in the vise?

24. How may the piece being chipped be kept from working down in the vise?
25. What kind of a chisel is used for chipping a keyway?
26. Why is it best, if possible, to drill a hole at the end of the keyway to be chipped?
27. Is it proper to lubricate a cold chisel when chipping steel?
28. Why is it advisable to anneal a forged tool before hardening and tempering it?

FILING

198. Use of Files.—When fitting machine parts together there are occasions when a slight reduction in size is required, and the use of a machine tool is impracticable. In such cases the file is most useful. Further, in many classes of work such as die making, experimental work and model work, surfaces must be finished and parts fashioned by filing. Therefore, it is important that a machinist shall be able to use a file skillfully.

Nothing looks much worse in the estimation of a mechanic than a poor job of filing. The art of filing, especially at the bench, is an acquired "knack." The beginner should learn all he can by reading, by observation, and by asking questions; then he may practice intelligently, and in no other line of machine shop work is it more true that practice makes for skill.

199. Machine Shop Files.—A file (Fig. 201) is a piece of high carbon crucible steel having teeth cut upon its body by parallel rows of chisel cuts. These cuts are made more or less diagonally across the face depending on the material on which the file is to be used and whether for roughing or finishing. When a file has a single series of cuts across its face, it is known as "single cut." The "double cut" file had two courses of cuts crossing each other. (See Fig. 202.) The terms "rough," "coarse," "bastard," "second cut," "smooth," and "dead smooth" refer to the distance apart of the parallel cuts on the larger files.
(10" or over) see Fig. 203, and on smaller files the Nos. 00, 0, 1, 2, 3, 4, 5, 6, 7, 8, refer to the same thing, No. 00 being the coarsest.

![Diagram showing different cuts and angles](image)

**Fig. 202.**

- **Single Cut**
- **Double Cut for General Work**
- **Double Cut for Finishing Work (Finer Files)**

These terms are relative and depend on the length of the file; a 16" "second cut" file is much coarser than a 10" "second cut."
cut" and a No. 00 8" file is coarser than a No. 00 4" file. The length of a file is always measured exclusive of the tang.

In general it may be stated that a 10" or 12" bastard file is used for rough filing at the bench, the second cut for bringing the work fairly close to a finish, and as fine a file as desired for the finish. The "rough" and "coarse" files and the "dead smooth" files are not much used in machine shop work, and of the smaller files the Nos. 00 and 2 are used much more than the finer cuts.

Files may be obtained in almost any desired shape or length and are commonly known either by their cross section as "square," "round," "three-square" (triangular), "half-round," etc.; by their general shape as "flat," "hand," "pillar," etc.; or by their particular use as "mill file," "warding file," etc. Files are used in all of the metal working trades; the shapes most commonly used in machine work are illustrated in Fig. 204 and a brief description follows.

The Mill File.—Nearly all of the files used in machine shops are double cut, the most notable exception being the mill file. The mill file is single cut, of substantially the shape of the flat file (Fig. 201) and most commonly 10" or 12" in length. It may be obtained with flat or rounded edges or one flat and one round as desired. Bastard or second cut are mostly used. The chisel teeth (single cut) give a smoother finish than the pointed teeth (double cut) but do not remove the metal
as fast. It is usually regarded as better than the double cut file for lathe work. It is much used for draw filing (paragraph 207), and in the bastard cut is fairly efficient for filing brass or bronze. The mill file derives its name by reason of its extensive use in wood-working mills for sharpening saws and planer knives. In machine shop work it is often called "float" file or "lathe" file, but "mill" file is the correct name.

The flat file is rectangular in shape and tapers slightly narrower and thinner toward the point and toward the heel (Fig. 201). It is the most commonly used file in the shop for general work. Usually 12" long and bastard cut but may be obtained in lengths from 6" to 16" in any cut.

The hand file is parallel in width with faces slightly convex. The hand second cut file is an excellent file for removing feed marks and for bringing flat surfaces fairly close to the finish. The hand smooth is the favorite finishing file for flat surfaces and is often used for finishing surfaces of round work revolving in a lathe.

The pillar file is similar to the hand file except that it is narrower. An 8" pillar file is light and "handy." It is adaptable for a great variety of filing operations and is one of the most popular files in the shop. Used usually in Nos. 00, 2, or 4 cut.

The square file is used for filing the smaller square on rectangular holes, for finishing the bottoms of narrow slots, etc.

The round file is used for enlarging round holes and for finishing round corners. It is generally tapered and the small sizes are often termed "rat-tail" files.

The three-square files are double cut on all three sides and the edges are very sharp. These files are especially valuable for finishing surfaces that meet at less than a right angle, and for backing-off special taps counterbores, etc., that frequently must be "home-made."

The half-round file is one of the most useful files in general machine shop work. It may be obtained in any length and cut.

The crochet file has both edges rounded. It is useful when filing against a filleted shoulder or a rounded corner of a hole.

The crossing file, sometimes called the "shad belly," is often
used in place of the half-round. Each side of the file has a different curve which feature frequently is of great convenience.

The warding file is a very thin file. It is essentially a locksmith's file for making the ward notches in keys. It is however very useful in the machine shop for filing slots and notches, and for finishing the sides of narrow grooves.

The barrette file has a flat triangular shape with teeth on the wide face only (safe back). It is a most useful file for finishing the sharp corners of many sorts of slots and grooves.

The knife file is used instead of the barrette file in similar work where the thinner cross section and the safe back of the latter is not necessary.

The files mentioned above are only a few of the shapes manufactured. By referring to a catalog of files it will be observed that there is manufactured a size, shape, and cut of file for practically every purpose and any material.

200. The Safe Edge.—The mill file and the flat file have single cut teeth on both edges; the hand file usually has teeth on one edge only, the other edge being termed the "safe" edge; the pillar file has two safe edges. If a safe edge is desired on any file it is easy to grind off the teeth. As a matter of fact a sharper corner may be obtained with a file so ground.

201. Convexity of Files.—Most files are made with the faces slightly convex lengthwise or "bellied." There are good reasons for this. If when filing a broad surface all the teeth were in contact, it would require too much pressure downward to make the file "bite" as well as forward to make it cut; this would mean practically double work and also make it more difficult to control the file. If the face of the file were straight, to produce a flat surface every part of the stroke would have to be perfectly straight. This is impossible. If a file were cut with flat faces and warped ever so little in hardening (and this is impossible to avoid) then one side would be concave and useless for flat work.

202. Taper of Files.—The convexity of a file should not be confused with the taper of a file. A flat file has faces which are convex, and it also tapers slightly in width. Certain files, for
example the square, round, and triangular files are more adaptable for a variety of work if they taper from near the middle to a very small cross section at the point, and are generally so made. There are occasions, however, when it is preferable to use a file of uniform cross section; such files are termed "blunt."

203. File Handles.—One important thing concerning files is too often disregarded; the file should be provided with a suitable handle properly fitted. The size of the handle depends on the size of the file, and the nature of the job. On the larger files the handle should be of a size that may be easily grasped; if too large or too small it will tire the hand in heavy filing. On the smaller files the handle should be of a size that will give balance to the file. When using a 4" or 5" file a piece of leather belting cut to a convenient shape makes an excellent handle. A wooden handle is fitted as follows: Drill a hole in the handle, of a size equal to the average thickness of the tang, and to a depth about equal to the length of the tang. Heat the tang of an old file to a dull red and force it about \( \frac{3}{2} \) its length into the hole and quickly withdraw it. Plunge the handle in water to stop the burning and then drive it on the new file, being careful not to split it. Practically the whole of the tang should be fitted to the handle.

![Diagram of file and handle](image)

Fig. 205.

204. Care of Files—Pinning.—There is always a tendency, especially when filing narrow surfaces or corners to have the file "pin," that is, small particles of the material being filed get wedged in front of the teeth of the file and scratch the work. Keep the file clean either with a "file card" \( a \), Fig. 205, or, by pushing the dirt from between the teeth with a
piece of soft steel, brass, or copper flatted thin on the end (b, Fig. 205).

Pinning is caused often by bearing too hard on the file, especially on the finer cut files. The worst kind of pinning is caused by hard usage of a new file. The new file should be used with great care until the small burrs on the ends of the teeth are worn away. Files are expensive cutting tools and it is a sure sign of ignorance or carelessness to throw them in a drawer or on the bench. Be orderly and careful; it pays.

**205. Cross Filing.**—Pushing the file endways, under more or less pressure against the work, is called cross-filing, or merely "filing." No pressure is applied to the file on the return stroke; it should not be removed from the work but may rub lightly.

**Holding the File.**—Most filing is done by holding the file in both hands. The file should be held in one hand only in especially delicate work where the suitable file is too small to be held in both hands. The proper way to hold the file for heavy filing is shown in Fig. 206. Grasp the file handle in the right hand, with the palm of the hand against the end of the handle and the *thumb on top*. Cover the other end of the file with the base of the thumb of the left hand and curl the fingers under.

For the lighter finishing cuts the position of the right hand remains the same, but the left hand may be changed to the
position shown in Fig. 207. This gives better control of the file.

The beginner will usually grasp the file too firmly. He will generally acquire about the proper grip after his hands get tired and cramped.

Position of the Body when Filing.—Skillful filing is a knack and to acquire this knack it is essential that conditions must be right. It is easier to do a thing well by the right method than it is by the wrong method, but having once learned by the wrong method it is difficult to acquire the right. The

![Fig. 207.](image)

height of the work to be filed in the vise should not be above the level of the workman's elbow as he stands erect, therefore, the shorter boys should be provided with platforms to stand on. Filing at the bench, especially the heavier filing, calls for a certain harmonious action of the arms, body and legs. Stand with the left foot pointing toward the bench, the hollow of the right foot 8'' to 12'' from the left heel and bend the body slightly forward at the hips. Hold the file as shown in Fig. 206 with the right arm bent to about 90° and the left arm somewhat nearer straight. Lean forward slowly for about two-thirds of the stroke, bending the knees slightly, and at the same time, push with the arms. During the last
third of the stroke, keep pushing with the arms but bring the body back slowly to nearly the original position. Then bring back the file lightly on the work to position for another stroke. Keep the file level or the work will be rounded instead of flat. The great fault is too much speed. Bear on hard but take slow strokes.

206. Operation of Filing.—If the work is cast-iron the scale should be removed, possibly by chipping, before an attempt is made to file the surface. A few strokes on cast iron scale will ruin a file. When rough filing it will be found that "crossing the stroke" (Fig. 208) at short intervals will rest the arms. It also serves to show the beginner where he tends to file the hardest, thus helping him to keep the surface straight, and practice in keeping the surface flat and straight is practice in learning to file. The work should be tested occasionally with a scale or with the blade of a square (Fig. 209). Test it crosswise, lengthwise and diagonally. A common fault is to rock the file. Either an over arm or an under arm rocking action will produce a convex surface; push the file as straight as possible.

If the surface being filed is to finish square with another surface, care must be taken to keep it fairly square when roughing because if one corner is filed \( \frac{1}{16}'' \) low, it means either that the work is spoiled or that the sixteenth thickness must be filed off the whole surface. Test it frequently with a steel square as shown in Fig. 210. When finishing the surface
the defects will show more plainly if the work and the square are held between the light and the eye. Remember that a machinist’s steel square is a tool that will not stand rough handling. If it is the least bit “out” it is worthless. Another advantage of the convexity of the file will by this time have become apparent. The skillful workman can control the file to make the few teeth that touch the work at one time cut just about where he desires. If the middle, or
one corner, or one edge is a little high he files off the high spot and this is the art of filing a true surface. Filing is a knack, to acquire which one needs to pay careful attention and to have patience; "All of a sudden" you have it and filing is easy. If one has attained a reasonable amount of proficiency in two hand filing—the sense of file balance and control, he will find no particular trouble in learning to file with one hand. A stool of a height that will bring the workman's shoulder to about the level of the work should be provided. The file may be grasped in several ways; the position of the hand as illustrated in Fig. 211 is suggested as one giving ease in operation and good control.

![Fig. 211.](image)

207. **Draw-filing.**—When it is desired to "line" or grain a piece of work (either flat or round) lengthwise it may be done by draw-filing. Hold the file as shown in Fig. 212. Keep it flat, and bear on as hard as necessary both directions of the stroke. Removing the handle in certain cases may give better balance of the file. The single cut file is usually regarded as best for draw-filing.

208. **Filing Soft Metals.**—Filing brass, solder, lead, etc., with the ordinary double cut file is very unsatisfactory for the reason that the teeth quickly become clogged with chips which are difficult to remove. The brass file (Disston) has been
designed to overcome this difficulty and also to produce a better finish with a coarser file. It is made with deep teeth and open bottoms; the up cut is on a longer angle than usual and the over cut is almost straight across. If such a file is not available a single cut file is usually more satisfactory than a regular double cut file.

The curved cut file (Vixen) Fig. 213 has proved very efficient for filing soft metals. The shape of the grooves between the teeth reduce to a considerable extent the clogging of the teeth when filing aluminum, brass, copper, etc. The curved cut file with its chisel edges gives an excellent finish when filing round work in a lathe. Further, it has very free cutting action which makes it desirable when considerable stock, iron
or steel as well as the softer metals, is to be removed. It is especially economical for the reason that unlike other files it may be advantageously re-cut or sharpened.

209. Needle Handle Files.—The very smallest files are made in the form of "needle handle files" (Fig. 214). They are much used in fine die work and are also very useful in delicate finishing touches in a variety of filing jobs. They are made in 4" to 6" lengths, only a third to a half of which is file shaped and cut, the remainder forming a slender handle.

![Diagram of file shapes](image)

**Fig. 214.—Needle handle files.**

210. Don't's in Filing

Don't use a worn out file.
Don't use a dirty file.
Don't use a new file on narrow edges.
Don't use a good file on cast-iron scale.
Don't use a fine file for filing soft metal.
Don't use a bastard file for finishing.
Don't use a smooth file for roughing.
Don't let the file hit the vise jaws.
Don't allow files to scrape together.
Don't put your fingers on the cast-iron surface being filed.
Don't push the file too fast.
Don't file too much before testing the work.

Questions on Files and Filing

1. What is one reason for having the file slightly convex or "bellied?"
2. If the file were not bellied, and warped in hardening, would its usefulness be impaired? Explain.
3. If the file were not bellied, would it be easier to "take hold" or harder? Why?
4. What is the effect when filing if the right hand tends to go down and the left hand rises slightly?
5. What is the effect when filing if the right hand tends to rise and the left to go down?
6. Is it easy to file the edges and produce a convex surface? Give reason.
7. It may be stated that in order to produce a flat surface with a 10" or a 12" file, a harmonic movement of the arms, body and legs is necessary. What does this mean?
8. What do you mean by a "knack?" Is filing a flat surface a knack? How is a knack acquired?
9. What is meant by crossing the cut in filing?
10. Should the file be lifted from the work on the return stroke? What is the reason?
11. What are the differences between a flat file and a hand file?
12. What is the difference between a bastard file and a second cut file?
13. In what way does the mill file differ from the other files?
15. How is the length of a file measured?
16. What is the difference between a "double cut" file and a "single cut" file?
17. What is the difference between a "double cut" file and a "second cut" file?
18. Which is the easier metal to cut with a file, cast iron or wrought iron?
19. How should the scale on castings be removed before filing the surfaces?
20. What is the reason a file should not be used to remove the scale from cast iron?
21. When should the coarser files be used? When should the finer files be used?
22. On narrow work, should an old file or a new file be used? Why?
23. What can be done to keep cast-iron filings from clogging the file?
24. How is the handle properly fitted on the tang of a file?
25. What is meant by a “safe edge” on a file? When is it advisable to use a file with a safe edge? If necessary could you grind a safe edge on a file?
26. What commonly used file has two safe edges?
27. Is a half-round file half round?
28. On what kind of surfaces is a half-round file used? What is the purpose of having teeth cut on the flat side?
29. How is the cut of a file designated in the smaller sizes?
30. Are the terms coarse and fine as applied to files relative for all kinds of files?
31. What 12” file would you use to remove stock rapidly? What file would you use for finishing?
32. What number 8” pillar file would you use for roughing? For finishing?
33. What is a file card? How may a piece of brass or copper rod be made into a most efficient file cleaner?
34. What causes “pinning”? How may the pin be removed?
35. How many different shapes (cross section) of files are you able to find?
36. In your judgment, why should a single cut file be best for filing in a lathe?
37. What is a needle handle file?

SCRAPING

211. Reasons for Scraping.—(1) It is practically impossible to produce a true flat surface in a machine or with a file. (2) Most of the flat bearing surfaces are iron surfaces, and the condition of an iron surface as it comes from a planer or shaper or milling machine is not suitable for a first class bearing surface; first, because it is not exactly flat and true, and second, because even the sharpest cutting tool does not produce the close grained and smooth surface that is necessary for such a bearing. (3) There are many curved bearing surfaces, for example in the bearing boxes and caps for shafts, spindles, etc. These surfaces whether of cast iron, Babbitt, or bronze must be scraped to obtain the alignment and fit necessary in high grade work.
212. Tools Used for Scraping a Flat Surface.—To produce an accurate flat bearing surface the "high spots," although they are only two or three thousandths high, must be located and scraped off. The tools used for locating the high spots and for otherwise gauging the shape and accuracy, are special gauge plates as for the ways of a lathe or a dovetail slide; surface plates, \(a\), Fig. 215 which are made in a variety of shapes and sizes; and iron straight edges, \(b\), Fig. 212 which are practically long narrow surface plates. These plates are scientifically designed to retain their shape. They are very expensive and should be handled and used with the greatest care.

The tools used to remove the high spots are called scrapers. Fig. 216 shows the commonly used forms of scrapers for flat work either of which may be made of a size convenient for the job at hand. The flat scraper \(a\), Fig. 216 for general machine shop use is usually about the size of a 10" or 12" hand file. It is drawn down on the end to about \(\frac{3}{16}\)" thick, hardened and the "snap" taken out, that is, heated just enough so that a drop of water will bubble on it. In other words, a scraper should be as hard as it is possible to make it. Scrapers are often made from old files but they are not as good as those made of special scraper steel. The hook scraper, \(b\), Fig. 216 is used for flowering or frosting, which are the terms used for the more or less regular scroll or patch work design which is sometimes used to "finish" a scraped surface. They are also used for scraping surfaces where it is inconvenient to use a
flat scraper, as for instance in the angle of a dovetail bearing surface.

Fig. 216.

213. Sharpening the Flat Scraper.—The taper sides of the cutting edge are ground flat and smooth, and the end is ground square edgewise but slightly convex lengthwise. The object
of the slight curve of the cutting edge is to enable the user to cant it ever so little to one side or the other to scrape exactly the spot desired without danger of scratching under the surface of the work with the corner of the scraper. After grinding, the scraper is oil stoned, the flat sides first and then the end. When stoning the end hold as shown in Fig. 217 and move the scraper for a distance of about 3” back and forth on the oilstone. Tip the handle ever so little forward and bear down on the push stroke easing up on the return. When one cutting edge is sharp, turn the scraper half around and oilstone the other edge. Do not hold the edge square to the direction of the stroke but about 45° as shown in the figure. This will tend to give a better edge if the oilstone is uneven, and also give the slight convexity desired.

214. The Scraping Operation.—To mark the high spots a thin application of Venetian red or Prussian blue paint is rubbed on the surface plate, and the work rubbed on the plate or the plate on the work, whichever is the more convenient. The high spots on the work will then show plainly, being marked with the paint. There will show the first time only a few spots and these will be more or less isolated. These are scraped off and the operation of marking repeated. As the
scraping continues, the number of the spots will increase, and strange as it may seem, the greater number of spots evenly distributed the more perfect the scraped surface.

Fig. 218 shows the correct position for holding the scraper. With the flat scraper the forward stroke is the cutting stroke and the usual stroke is seldom over half an inch in length. The effect should be clean cut and smooth—no scratches. Keep the scraper sharp; when one edge becomes dull turn it over and use the other edge. Oilstone it when necessary and grind it a little every three or four times it is oilstoned. Not more than two or three thousandths should be scraped; if the work is more uneven it probably should be machined again although it may perhaps be more economical to file it. As the spots increase in number and decrease in size, considerable judgment must be used as to just where and how much to scrape. Turpentine may be used instead of the paint when close to the finish to show the high spots which will appear bright. The turpentine acts also as a lubricant between the work and the plate, which is often an advantage, and further it serves to make the scraper cut better.

215. Hints on Scraping.—Do not allow any oil, not even your fingers which are naturally oily, to touch the surface being scraped.

It is usually quicker and easier to remove the "feed marks" by filing. Be careful, however, not to over-do it.

Until the surface is substantially true it is necessary, to accomplish anything, to scrape hard. As the spots begin to show evenly over the surface ease up on the chip.

Dipping the scraper occasionally in turpentine (or water) will help it to cut easier and better.

When roughing especially, try to keep the cuts about square in shape and cross them in succeeding courses. This will help to make the marking more easily distinguishable.

The best way to apply the paint is with a rag swab or by hand. Apply it more generously when roughing than when finishing.

Be extremely careful that no grit or dirt gets on the surface
plate and that none remains on the work when being tested. Keep the paint box covered.

Use the whole plate—not one spot.

Keep the scraper sharp or it will scratch. Scratches spoil a scraped surface.

![Fig. 219.](image)

216. Scraping Curved Surfaces.—Round bearings and other curved surfaces have often to be scraped to fit running or sliding parts. The scrapers used are shown in Fig. 219. The half-round bent scraper (a) is mostly used. It has two cutting edges and the cutting stroke may be either toward or away from the user as desired. The proper way to hold this scraper is shown in Fig. 220. For the larger curves the scraper shown
at b (Fig. 219) is preferred by many; it is ground to conform to the curve of the work and sharpened to cut on the pull stroke. Scraping a curved surface to fit a shaft or a gauge is not particularly different in principle or operation from scraping a plane surface.

The three-cornered scraper c, Fig. 219, is used to "break" (remove the sharp edge), or round as desired, the corners of holes or other curved edges. It is usually made from an old file by grinding off the teeth. The three corners of the body back of the cutting edges should be well rounded in order not to cut the hands of the user.

**Questions on Scraping**

1. Is there a difference between the cutting angle of a flat scraper and a scraper for curved work?
2. Is the end of a flat scraper straight? What shape is it lengthwise? Crosswise?
3. What is the difference between a scraper for a round bearing and a three-cornered scraper? Could a three-cornered scraper be used for scraping a round bearing?
4. Why is a scraper for round bearings curved slightly?
5. When is a three-cornered scraper used? How are they usually made in the shop?
6. Of what kind of steel is a scraper made? Why?
7. How is a scraper hardened? How is it tempered?
8. What care must be taken when grinding a scraper?
9. Why is a scraper oil-stoned after grinding?
10. How is a scraper for a round bearing oil-stoned?
11. How is a flat scraper held when being oil-stoned?
12. What are the advantages of a scraped surface?
13. What is a surface plate? Why must great care be taken of a surface plate?
14. How is a flat bearing surface tested?
15. How is a round bearing surface tested?
16. Why is a thin coating of red lead or prussian blue applied to a surface plate? Which is better? Why?
17. How do you determine when a surface is sufficiently scraped?
CHAPTER XV

WORK AT THE FORGE

SOLDERING, BRAZING, BABBITTING, HARDENING AND TEMPERING

217. Soldering is the operation of joining metal surfaces by means of a surface fusion to these surfaces of another metal (solder) of a lower melting point. Common soft solder is a compound or an alloy of equal parts of tin and lead and melts at about 430°F.

218. The Principle of Soldering.—Metals have an affinity or attraction for each other, that is, under certain conditions they will combine forming a new substance called an alloy. Many conditions govern the alloying of metals. One of these conditions, usually but not always, is heat. Mercury will combine with gold, silver, copper and zinc at ordinary temperatures. This can be observed by rubbing mercury on a piece of copper. On the other hand lead and copper and tin do not combine at ordinary temperatures, but they do combine readily at higher temperatures.

Soldering is a process of forming an alloy of the various metals present on the surfaces of the pieces to be soldered together, and involves the fusing into a common mass of all the metals present at the point of connection. This fusion of metals takes place at a temperature below the melting point of some of them. As stated above soft solder is an alloy of tin and lead in equal parts; tin melts at a temperature of 450°F. and lead melts at a temperature of 618°F.; nevertheless, if a piece of lead be placed in molten tin the two will quickly combine. If equal amounts of lead and tin are thus combined the melting point of the alloy formed will be about 430°F. which is lower than that of either metal.
When two pieces of metal are soldered tightly together they are held together because of the fact that a very thin layer of alloy (an alloy of the solder and the metal) has been formed on the surface of each piece through the application of heat and they have fused together. If it is not a tight joint the solder between the surfaces will fill the space as well as hold the pieces together. The underlying principle of soldering is that the comparatively low temperature of the soldering copper will serve to melt the solder and form an alloy on the surface of the materials being soldered, and fuse these alloys together, or to the solder between them as the case may be.

219. Cleaning the Surfaces. Fluxes.—One of the essentials of the soldered joint is that there be contact between absolutely clean surfaces. The preliminary cleaning may be done with a scraper, file, or piece of emery cloth. However, a metal surface that has been thus cleaned immediately oxidizes through exposure to the air. That is, the oxygen in the air combines with the metal and forms a thin coating on the surface of the metal. The layer of oxide no matter how thin, will not form an alloy with another metal; soldering cannot be accomplished between surfaces which are oxidized. Scale, grease and dirt must be removed, and further, the oxides of the metals must be removed before a union of the metals can take place. Since oxidation cannot be entirely avoided the layer of oxide must be removed at the instant of soldering. This is accomplished by the use of fluxes. The action of the flux as it vaporizes is to carry off with it the oxide. There are several kinds of fluxes used in soldering. The flux most commonly used in machine shop work for soldering brass, galvanized iron, and steel is "killed" muriatic (hydrochloric) acid. Muriatic acid is killed by adding small scraps of zinc a little at a time until the acid has eaten all it will and ceases to boil. As the acid eats the zinc, hydrogen in the acid is liberated. The killed or cut acid is no longer muriatic acid, but is known as chloride of zinc. While it is being killed it should be moved away from any machines or tools that may be rusted by the escaping fumes. For soldering brass, copper, tin and
steel it is inadvisable to use a flux which is too strong. It should be diluted with about 25% water.

Muriatic acid, or raw acid as it is called in soldering work, is used as a flux when soldering pieces to cast iron or black sheet iron or zinc. When raw acid is used for a flux the work should be rinsed in water, after soldering, to stop the action of the acid.

For soldering tin or copper powdered rosin makes the best flux. It is sprinkled along the joint and when melted has a tendency to flow into the seam. Borax is used as a flux when using hard solder (spelter) that is, when brazing (see paragraph 224). The borax acts similarly to rosin in that it works its way into the seam or joint by capillary action, and the solder follows.

220. Soldering Coppers.—Soldering coppers, Fig. 221, are made in different shapes and of varying weights. The square pointed copper is used more for general work than such shapes as the blunt roofing copper or the chisel shaped bottom copper. A small copper should not be used on heavy work because it will not keep its heat long enough. On the other hand a copper that is too heavy is unnecessarily clumsy for light work. It is very necessary that the copper be properly "tinned" and that in heating care be exercised not to overheat. If it is overheated a scale of copper oxide is formed on the point and
this scale being practically a non-conductor of heat renders the copper almost useless until it is re-tinned.

221. Tinning a Copper.—To tin a soldering copper it is necessary to file or otherwise smooth and clean the end for about \( \frac{3}{4} \) of an inch back. The copper is then heated enough to melt solder, after which the flux and the solder are applied. There are several ways of applying the flux and solder when tinning the copper, the method employed depending almost altogether on the material at hand.

1. The soldering iron may be dipped in the acid and then the solder applied.

2. The point of the copper may be rubbed with a piece of sal-ammoniac and then the solder applied. This is the quickest and best way.

3. A piece of solder and some powdered rosin may be placed on a brick and the heated copper rubbed thereon until it is tinned.

222. Dipping Solution.—If the copper while being heated becomes discolored by reason of the kind of fuel used (charcoal, gas, or gasoline) it may be cleaned by dipping it quickly in a solution made by dissolving a teaspoonful of powdered sal-ammoniac in a quart of water.

223. Soldering Operation.—When soldering it is often convenient to "tack" one piece to the other, that is, a few drops of solder are put here and there to hold the piece in position. When finishing the operation care must be taken to let one portion cool before proceeding to the next. The copper should be so applied that as much of the available heat as possible may be utilized and further it must be placed in such a position on the work over the joint that the solder will flow into the seam and not merely along the outer edge.

Oftentimes it is good practice to solder two or more thin pieces together and machine them as one, afterwards melting them apart. It is usually best when soldering steel to steel to be sure both are tinned, because steel does not alloy with solder as readily as does copper or brass. If the steel pieces are properly cleaned and heated, flux applied generously and
solder rubbed on with the heated copper and the excess rubbed off with a piece of waste, the surface will show bright with a thin coating of solder. Like the soldering copper they are "tinned." If a little flux is applied and the tinned surfaces are held together and heated they will be perfectly soldered or as sometimes called, "sweated" together.

**BRAZING**

224. Brazing.—A much stronger joint can be made by brazing than by soldering. Brazing is a process of joining metal parts which is similar to soldering except that "spelter" is used instead of solder. Spelter is a compound of copper and zinc and is often called hard solder. It is usually about half and half copper and zinc; adding more copper up to \( \frac{2}{3} \) copper and \( \frac{1}{3} \) zinc serves to produce a stronger joint, but makes it more difficult to work. A spelter made of half copper, \( \frac{3}{8} \) zinc and \( \frac{1}{8} \) tin makes an excellent spelter. Spelter is used in either granular or wire form. Brass rod is a combination of substantially \( \frac{2}{3} \) copper and \( \frac{1}{3} \) zinc (with a small amount of lead added to make it machine easier) and if prepared spelter is not available brass filings will make an excellent substitute.

The flux used for brazing is powdered borax. A small amount is mixed with water to form a paste and applied to the surfaces to be brazed before they are heated. During the process of brazing the dry borax is sprinkled on the joint where it melts and flows between the surfaces to be joined. The use of too much flux should be avoided because it hardens the surfaces of the joint and makes the filing and finishing difficult. When the flux begins to flow the spelter is placed on the joint and the heat continued until the spelter flows into the joint and no longer. A spatula for placing the flux and also the spelter on the joint may be made by flattening the end of a steel rod of suitable diameter and length.

To produce a strong joint it is necessary to have the surfaces fitted and held together tightly. Copper, brass, wrought iron, malleable iron, or steel may be brazed. Care must be taken when brazing copper or brass not to overheat and melt the work.
Questions on Soldering and Brazing

1. Of what materials is soft solder composed?
2. Steel is an alloy. Brass is an alloy. What is an alloy? Is solder an alloy?
3. When a piece of solder is melted on a piece of copper is a new alloy formed?
4. What is meant by the term "fuse"?
5. When a piece of solder is melted on a piece of copper, why does the solder cling to the copper?
6. What is rust? What is meant by oxidation?
7. If a piece of steel is polished, it will retain its brightness in ordinary temperature for several days or maybe weeks, but if heated sufficiently it will quickly become almost black. Why is this?
8. Why will not solder cling to an oxidized surface?
9. What is the action of a flux?
10. Why is a flux used in soldering?
11. What flux is commonly used in machine shops? How is it made?
12. How is the soldering copper tinned?
13. What care must be taken when heating a copper? Why?
14. How are pieces of metal sweated together?
15. What is the difference between soldering and brazing?
16. Of what materials is spelter composed?
17. What flux is used for brazing?
18. How is the flux applied to the surfaces to be joined?

BABBITTING

225. Babbitt.—Babbitt metal is an alloy of copper (4 parts), tin (88 parts), and antimony (8 parts). It is widely used as a lining for bearings for the following reasons: The bearings (boxes and caps) do not have to be bored; the shaft or spindle of the machine may be aligned and the Babbitt melted and poured around it. It is practically an anti-friction metal. It is strong, tough and durable. It may be readily machined if necessary, and scraptes much easier than cast iron or bronze. When the bearing becomes worn the Babbitt is easily broken out with a chisel, remelted and pured to form a new bearing surface.

Lead is added in the cheaper grades of Babbitt. This is all right for shafts having light duties to perform but to produce good Babbitt bearings requires a high grade of Babbitt metal, of which there are several brands in the market.
226. Babbitting a Bearing.—There are several methods of pouring Babbitt bearings:

1. The bearing box or housing is cylindrical in shape and the metal is poured around the shaft, or around a rod or "babbitting mandrel" the exact size of the shaft to be used.

2. The metal to form the bearing is poured around a mandrel somewhat smaller than the exact size of the shaft. The bearing is afterward bored to the size required.

3. The bearing is split on a center line horizontally, separating the upper and lower halves. Both the lower and upper parts of this bearing are poured at the same time. Pieces of card-board called liners are placed between the two, and against the shaft, holes being made in the liners near the shaft for the passage of the metal from the upper to the lower part. (The narrow portions of metal may be easily broken when the cap is removed.)

4. The bearing metal is poured in the upper and lower parts separately, the lower part, being poured first, after which the upper part or cap is adjusted and the Babbitt metal poured in to form the lining for the cap.

It should be understood that in either of the above methods (3) or (4), the metal may be poured around a shaft or mandrel to the exact size and merely scraped to form a suitable bearing surface, or it may be poured around a shaft or mandrel or a suitable rod somewhat smaller than the exact size of the bearing required and afterward machined to size. If before machining, light blows are struck with a ball peen hammer over the entire surface, it will serve to harden the surface and make the grain of the alloy closer when machined and a more durable bearing is produced. This method is considered the best.

The bearing boxes are usually cored to the required size. On the smaller sizes the cored holes are straight and cylindrical. To prevent the Babbitt lining from working loose small holes may be drilled at right angles to the axis of the bearing and the metal running into these holes when poured, serves to tie the lining securely to the box or cap. To serve the same
purpose in the larger bearings the box or cap may have two or more dove-tail recesses cored parallel to the axis. Where a thrust load is taken on the bearings, the casting is either cored or counterbored somewhat larger in the end taking the thrust in order to provide against any tendency for the load to loosen the lining.

If the shaft or babbitting mandrel is painted with a mixture of graphite and gasoline it will make removal easier. This is especially true in a solid bearing, that is, one in which the metal forms a solid sleeve around the shaft. A coating of lamp black applied by holding a candle flame against the shaft will serve the same purpose.

It is advisable to preheat the mandrel and also the casting of the bearing. Otherwise the cold casting may chill the Babbitt enough to prevent its filling the space, and in any event the bearing surface will not be as smooth as if the box and shaft were heated. The heating may be done with a blow torch or by any convenient means.

If the bearing is to be poured "to size" considerable care should be taken to align the shaft. If many boxes are to be Babbitted it will save time to make a suitable fixture for holding the shaft in position.

Fire clay mixed with water to the consistency of putty may be used to close the openings between the shaft and the box and thus keep the metal from running out. It may be well to cut card-board washers and back them up with the clay.

The Babbitt metal should be slowly heated until it will quickly burn a pine stick to a dark brown. Too much heat will injure the metal and when insufficiently heated it does not pour well. If a small amount of crushed rosin is put in the Babbitt just before pouring a smoother bearing surface will result.

Care must be taken that no water comes in contact with the melted Babbitt. Even a few drops of water will cause the metal to spatter.

The pouring must be continuous. If it is interrupted the additional metal will not fuse with that already poured and
the lining will be cracked. It is sometimes necessary when pouring large bearings to use two ladles pouring from both at the same time.

Questions on Babbitting

1. Of what materials is Babbit made?
2. What causes the difference in the grades of Babbit?
3. What is Babbit used for?
4. Name three advantages of Babbit for the purpose for which it is used.
5. Explain how the cheapest type of Babbitted bearing is produced.
6. Explain the principal features of the best type of Babbitted bearing.
7. Would a bearing such as suggested in question 5 be suitable for shafts having heavy duty to perform? Why?
8. Would a bearing such as suggested in question 6 be practicable for a 1" diameter shaft? Why?
9. How is a small Babbit bearing kept from working loose?
10. How is a large Babbit bearing kept from working loose?
11. When a thrust load is to be taken on the bearing, what provision is made to keep the Babbit in place?
12. When proceeding to Babbit a bearing, how do you get the boxes ready?
13. How do you get the shaft or mandrel ready?
14. How do you separate the cap from the box? Why?
15. Why do you heat the box and the shaft?
16. How do you prevent the Babbit from sticking to the journal?
17. How do you prevent it from running out at the ends of the box?
19. Why is a bearing spoiled if the pouring is interrupted?
20. How slowly should the Babbit be melted? Why?
21. How may you tell when the Babbit is hot enough to pour?
22. What does it matter if the Babbit is too hot?
23. How does a little resin affect the Babbit?
24. When do you put it in?
25. What will cause the melted Babbit to "spatter"?

BRIEF OBSERVATIONS REGARDING THE HARDENING AND TEMPERING OF STEEL

227. Steel is an alloy of iron and carbon. The carbon content is usually not over 1.40% (140 point)—high carbon "razor temper" steel, and may be as low as .10% (10 point)—"dead soft" steel. The nearer pure iron the steel is,
the softer it is, the more easily it forges and welds, and the more readily it machines. Consequently 10 point to 20 point carbon steel (dead soft) is used for making chains, rivets, etc., and 20 point to 35 point, being stronger, yet machining easily, is used for bolts, shafting, etc.

Steel between 30 and 70 point is used for heavy forgings in engine work and for car axles, rails, etc.

Steel over 75 point (75%) carbon content if heated to a dull red and plunged into water will become very hard and very brittle. This is called hardening. If it is slowly and carefully re-heated the brittleness is gradually reduced but it also becomes softer as the temperature rises. This is called tempering. The degree of heat for tempering depends on the combination of hardness and toughness desired.

228. Machine Steel and Carbon Tool Steel.—Steel under 50 point carbon does not harden perceptibly, and is called "low carbon" or "mild" or "machine" steel. Steel over 75 point will harden and is called "high carbon" or "tool" steel and to differentiate it from the "high speed" steels, which will presently be referred to, it is often called "carbon tool steel." While the 75 point steel will harden and is suitable for hammers, crowbars, etc., it is not suitable for the cutting tools used in machine shops. Steel for drills, taps, reamers, etc., requires more carbon and the best carbon tool steel for cutting tools has a carbon content of from 90 to 100 point. Steel of substantially 100 point carbon is used for drill rod and spring steel; for punches and dies; for hack saws and files, etc., etc. Steel when heated is changed in its structure and in its properties. Steel of around 100 point carbon when heated to a temperature ranging from 1375° to 1400°F. attains the finest structure (grain size) which it can have, and if quenched in cold water the moment it has attained this heat, it is trapped in this perfect structure and in a condition of extreme hardness.

When re-heated sufficiently (about 400°F.) a noticeable change begins to take place in the quality of hardness, and
when heated some 600°F. the steel has lost all of its brittleness and most of its hardness. That is to say, in carbon tool steel the changes from glassy hard to practically soft—the whole range of temper—takes place between the temperatures of 400° and 600°.

229. Hardening.—If steel is not heated enough it will not harden; if overheated it is injured; if very much overheated it is ruined. The proper temperature is determined largely by the carbon content, the less carbon the greater heat. For instance, a hammer made of 75 point carbon steel will require 1425°-1450°F. while a reamer made of 100 point carbon steel requires less heat (1350°-1400°F.) and will be injured if heated to 1450°F.

Metcalf's Experiment.—To show the effect of the different degrees of heat on the strength and appearance of the hardened steel, take a piece of steel about \( \frac{1}{2}'' \) diameter 7'' or 8'' long, and with a pointed tool, cut shallow grooves \( \frac{1}{2}'' \) or so apart and number the sections, Fig. 222. Put one end of the piece in the fire and allow it to get white hot (very much overheated) before the other end is hot enough. Plunge into water and after thoroughly drying break off as many sections as are necessary to show: (1) That the overheated end is weak—breaks easily—and the grain is very coarse. (2) That as succeeding pieces are broken the blow necessary to break them is increasingly greater and the grain size is finer. (3) That the end insufficiently heated is not hard—it may be easily filed. (4) That in the hardened section the greatest strength is accompanied by an uneven fracture and the finest grain.

230. Hints on Hardening.—Do not let the air blast come in contact with the steel more than is necessary because it oxidizes the surface—that is, burns out the carbon and consequently the surface will not harden. If such a surface can-
not afterward be ground off—for example in such a tool as a tap, the tool will be worthless.

Heat the piece slowly, thoroughly, and evenly, otherwise it is liable when quenched to be soft in spots, or warped or possibly cracked.

Do not overheat the steel or it will not hold an edge. If by any chance it is overheated, do not hold it until it looks about right and then quench it, this will do no good. It will be much better to allow the steel to cool, then re-heat it to the proper temperature and quench. If a piece is much overheated the surface is blistered. Such a surface is permanently ruined.

Do not use the fire hotter than necessary, else the corners or thin sections will heat too quickly—before the rest of the piece is hot enough.

Do not lay a long slender piece so that it can bend of its own weight when heating, or it will be bent when hardened.

Quenching should be done on the ascending heat. Remove the work from the fire to the quenching bath as quickly as possible.

Quench the longer work lengthwise and the flat work edgewise to avoid warping. If the sections of the work vary greatly in size, quench the large section first to avoid cracking.

To avoid soft spots keep the piece moving in the quenching bath, thus keeping it in constant contact with the cold bath and away from the steam produced.

Keep the work in the bath until it may be touched without burning the hand, then remove it and re-heat until a drop of water will bubble on the surface. This is called “taking the snap out.” The work as it comes from the bath is under enormous hardening strains. Taking the snap out means reducing these strains to a considerable extent and has saved hundreds of pieces.

For the smaller cutting tools or work of a frail cross section or larger pieces that need not be especially hard, a bath of “quenching oil” is most satisfactory. For average work, however, cold running water is the most efficient quenching bath.
231. Tempering Experiment—Temper Colors.—Take a rod \( \frac{1}{4}'' \) or more in diameter and 5'' or 6'' long and harden it. Polish it bright and wipe it clean. Hold it in a pair of tongs and pass it lengthwise slowly back and forth over the fire allowing the heat to act most on the further end and gradually less toward the end in the tongs. Examine occasionally and observe that soon the silver brightness will disappear on the hottest end giving way to a faint yellow or light straw color. As the rod is further subjected to the heat the light straw will creep up the rod and be followed successively on the end and along the rod by a dark straw—brown—brown with purple spots—purple—bright blue—dark blue.

These are called temper colors and are very useful many times to indicate the degree to which the steel has been re-heated or as the hardener says, how much the temper has been "drawn." With the corner of an old file test the hardness of various sections of the work. It will be noticed that the end which has been drawn to a blue files fairly easy, and that it is increasingly difficult to make the file bite as the portion of the rod showing light straw is approached.

The temperatures corresponding to the various colors are as follows:

<table>
<thead>
<tr>
<th>Color</th>
<th>Degrees F</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pale Yellow</td>
<td>430</td>
</tr>
<tr>
<td>Light Straw</td>
<td>450</td>
</tr>
<tr>
<td>Dark Straw</td>
<td>470</td>
</tr>
<tr>
<td>Brown</td>
<td>490</td>
</tr>
<tr>
<td>Brown with Purple Spots</td>
<td>510</td>
</tr>
<tr>
<td>Purple</td>
<td>530</td>
</tr>
<tr>
<td>Bright Blue</td>
<td>560</td>
</tr>
<tr>
<td>Dark Blue</td>
<td>600</td>
</tr>
</tbody>
</table>

The colors themselves are merely different thicknesses of films of oxide which are caused by heating the steel to different degrees. A soft piece will color the same as a hard piece. The fact that a piece is temper colored does not indicate that it has been hardened, and rubbing off the color certainly does not make the piece soft.
232. Hints on Tempering.—Machine shop tools will give good results if tempered about as follows:

<table>
<thead>
<tr>
<th>Color</th>
<th>Tools</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pale Yellow</td>
<td>Cutting tools for Lathe, Planer, Shaper.</td>
</tr>
<tr>
<td>Light Straw</td>
<td>Milling Cutters, Drills, Reamers.</td>
</tr>
<tr>
<td>Dark Straw</td>
<td>Taps and Dies.</td>
</tr>
<tr>
<td>Purple</td>
<td>Center Punches, Cold Chisels.</td>
</tr>
<tr>
<td>Purple Verging into Blue</td>
<td>Screw-drivers.</td>
</tr>
</tbody>
</table>

Common soft solder melts at approximately the heat required for tempering a milling cutter, reamer, or similar tool. Heat the piece carefully until it will melt the solder rubbed on it.

In tempering, the piece should be heated slowly, carefully and thoroughly at some little distance from the fire, otherwise the thinner sections will be drawn lower than necessary before the heavier part is drawn enough.

The piece being tempered should be moved constantly, to bring each portion, where an even temper is desired, to the same degree of heat. Such a tool as a tap or a reamer or an end mill should be passed slowly back and forth over the flame and turned at the same time. The distance from the fire will depend on the intensity of the heat; do not attempt to draw the temper too quickly.

Small pieces may be tempered by moving them about on a heated plate until the temper is sufficiently drawn. The most satisfactory method of tempering is by means of oil (tempering oil, cotton-seed oil, or fish oil) heated to the desired temperature. The pieces are allowed to remain in the oil until they are thoroughly tempered. It does not injure them to remain longer than necessary in the oil provided the temperature does not rise.

When hardening and tempering cold chisels, screw-drivers, or forged lathe, shaper and planer tools, etc., it is customary to proceed as follows: Heat the piece somewhat farther back than is necessary, harden as much of it as desired, brighten the hardened part with a piece of emery cloth, or a piece of a broken abrasive wheel, and then allow the heat in the unhardened portion to draw the temper. When the proper temper color is observed dip the whole piece. One precaution should
be emphasized; when hardening a piece as above, keep it moving up and down in the bath through a distance of half an inch or so, in order not to cause too sharp a line between the hardened and unhardened portions. If the piece is not moved it is liable to crack at the water line.

233. High Speed Steel.—In the two experiments made (paragraphs 229 and 231) it was demonstrated (1) that heating a piece of carbon steel above 1400° produced a coarse grain, and (2) that the range of temperature for tempers from glassy hard to practically soft was from about 400° to 600°. This is true for carbon steel but not true for high speed steel.

Mushet, the famous English steel expert, discovered many years ago that a steel containing 1.5% carbon, from 5 to 8% tungsten, and about \( \frac{1}{2} \)% manganese would harden in an air blast, and would hold its temper until it was practically red hot. Mushet steel, or air hardening steel as it was sometimes called, was therefore capable of taking a faster and heavier cut since the friction did not burn out the temper and break down the cutting edge. But Mushet steel could not be readily machined and did not come into popular use.

It was not until the American investigators Messrs. Taylor and White discovered the special properties of steel with substantially .68% carbon, 18% tungsten and 5 to 6% chromium and developed a special process of heat treatment of this steel that the possibilities of high speed steel were recognized. Since then (1898) dozens of different brands of high speed steels have been manufactured. They are probably all or nearly all alloys of tungsten and chromium, or molybdenum and chromium with carbon steel.

There are two great advantages in the use of high speed steel for cutting tools. (1) Greater lee-way may be given in the heat treating temperature—it is fairly difficult to spoil a piece of high speed steel unless it is melted. (2) The temper is not ruined and the cutting edge does not break down even when the tool is red hot consequently more than double the production can be obtained with a high speed cutting tool than with a carbon steel tool.

High speed steel costs several times as much as carbon steel
and for such tools as taps, dies, hand reamers, gauges, etc., is no better than carbon steel. For milling cutters, lathe tools, and much used sizes of drills and machine reamers, it is worth much more than the extra cost.

The Taylor-White process is used for the treatment of tungsten-chromium steel.

Taylor-White Process

1. *The high heat treatment.*
   (a) Heat slowly to 1500°F.
   (b) Heat rapidly from 1500°F. to a white heat (2200°F.).
   (c) Cool rapidly (Kerosene bath) to below 1550°.
   (d) And cool either rapidly (Kerosene) or slowly (air blast) to the temperature of the air.

2. *The low heat treatment.*
   (e) Heat to a low red (1150°F.) for about 5 minutes.
   (f) Cool either rapidly or slowly to the temperature of the air.

For molybdenum-chromium high speed steel the treatment is substantially as above except that in b it is unnecessary to heat the steel to a white heat, 1850°F. is sufficient.

Questions on Hardening and Tempering

1. What is the difference between mild steel and carbon tool steel?
2. What is the difference between hardening and tempering?
3. What is the meaning of the term "point" when speaking of steel?
4. Explain how "Metcalf's experiment" is made.
5. Explain the value to the machinist of Metcalf's experiment.
6. When hardening steel why should it be heated thoroughly and evenly?
7. When tempering steel why should it be heated slowly and evenly?
8. What effect does overheating have when hardening?
9. What effect does overheating have when tempering?
10. What is the difference between "burning" a piece of steel and "burning" the temper?
11. What is meant by "taking the snap out" of a piece of steel?
12. What are temper colors? What do they indicate?
13. Does a cutter drawn in oil have temper colors? Give reason.
14. Explain the processes of hardening and tempering a cold chisel.
15. What may happen if the chisel is not kept in motion when quenching? Why?
16. What are some of the advantages of high speed steel?
17. How is high speed steel heat treated?
APPENDIX

RULES FOR FINDING THE DIAMETERS AND SPEEDS OF PULEYS

The speeds (r.p.m.) of driving and driven pulleys are to each other inversely as their diameters. That is, the Speed of driving pulley is to the speed of the driven pulley as the diameter of the driven pulley is the Diameter of the driving pulley.

\[ S : s = d : D \]

This is usually written \( DS = ds \) and is the formula for

The Fundamental Rule for Speeds of Pulleys.—The diameter of the driving pulley multiplied by its speed is equal to the diameter of the driven pulley multiplied by its speed.

Knowing any three of the quantities the fourth can be found by substituting values in the proper one of the following equations:

\[ DS = ds \]

Then

\[ (1) \quad D = \frac{ds}{S} \]

\[ (2) \quad S = \frac{ds}{D} \]

\[ (3) \quad d = \frac{DS}{s} \]

\[ (4) \quad s = \frac{DS}{d} \]

Example: A pulley 12'' in diameter is running 220 r.p.m., is connected by a belt to a pulley 7'' in diameter. How many revolutions per minute will the smaller pulley make?

277
Solution.—Use (3) \( d = \frac{DS}{s} = \frac{12 \times 220}{7} = \frac{2640}{7} = 377 \) r.p.m.

Pulley Train.—The principal driving shafts in a shop are called "main lines" and the smaller shafts that carry the pulleys over the machine are called "counter-shafts." Often the speed must be reduced between the engine and the main line in which case a "jack-shaft" carries the speed reducing pulleys. When motion of one shaft is transmitted to another and from that to a third and so on to any number of shafts, the pulleys that carry the belts which transmit the motion, make up what is called a pulley train.

Suppose power is transmitted from a pulley on the motor to a pulley on the line shaft. The motor pulley is the driving pulley and the line shaft pulley is the driven. The power is further transmitted from another pulley (a driving pulley) on the line to a driven pulley on the countern shaft and from another (driving) pulley on the countern shaft to the (driven) pulley on the machine.

The problem of calculating the speeds, etc. in a pulley train is the same in principle as for two pulleys. Instead of calculating for each pair of pulleys in the train separately, a combination of the different proportions will give the same result. Hence the following:

Rule for Pulley Speeds.—The continued product of the diameters of the driving pulleys and the speed of the first driver is equal to the continued product of the diameter of the driven pulleys and the speed of the last driven pulley.

Example.—A certain line shaft runs at 250 r.p.m. A 15" pulley on this shaft is connected by a belt to a 10-inch pulley on the countern shaft. From a 12-inch pulley on the countern shaft motion is transmitted to the machine. What diameter must the pulley on the machine be to give a spindle speed of 600 r.p.m.

Solution.—Let \( x = \) dia. of pulley on machine.

\[
\frac{\frac{15}{2}}{10} \times 15 \times 12 = x \times \frac{600}{2}
\]

\[15 = 2x \text{ or } x = 7\frac{1}{2}" \text{ Ans.}\]
RULES FOR FINDING THE NUMBER OF TEETH AND VELOCITY OF GEARS

The velocity (r.p.m.) of the driving gear and the follower gear are to each other inversely as the numbers of their teeth. That is, the velocity of the driving gear is to the velocity of the follower gear as the number of teeth in the follower gear is to the number of teeth of the driving gear.

or

\[ V : v = n : N \]

This is usually written \( NV = nv \) and is the formula for the Fundamental Rule for Velocities of Gears.—The number of teeth in the driving gear multiplied by its velocity is equal to the number of teeth of the driven gear multiplied by its velocity.

Knowing any three of the quantities, the fourth can be found by substituting values in the proper one of the following equations.

\[ NV = nv \]

Then

\[ (1) \quad N = \frac{nv}{V} \]

\[ (2) \quad V = \frac{nv}{N} \]

\[ (3) \quad n = \frac{NV}{v} \]

\[ (4) \quad v = \frac{NV}{n} \]

Example.—A gear with 40 teeth meshes with a gear having 96 teeth. If the small gear makes 120 revolutions per minute, what will be the velocity of the larger gear?

\[ \frac{NV}{n} = \frac{5}{40} \times \frac{10}{120} = 50 \text{ r.p.m. Ans.} \]

Solution.—Use (4) \[ v = \frac{NV}{n} \]

A compound gear train is a train of gears composed of two or more pairs, or simple trains, of gears. The problem of calculating velocities, etc. in compound gearing is the same
in principle as for two gears. Instead of calculating for each pair of gears in the train separately a combination of the different proportions will give the same result. Hence the following:

**Rule for Compound Gear Velocities.**—The continued product of the *numbers* of teeth in the driving gears and the velocity of the first driving gear is equal to the continued product of the *numbers* of teeth in the follower gears and the velocity of the final follower gear.

**MACHINE FITS**

There are four different kinds of cylindrical fits used in machine work, namely: the running (and sliding) fit, the drive fit, the force fit and the shrink fit.

In most cases, excepting the shrink fit, the hole should be finished to a standard size, the shaft or stem or other part then fitted to it. Babbited bearings are an exception.

**Sliding and Running Fit.**—For a sliding or running fit the diameter of the shaft should be enough smaller to allow for a film of oil for lubrication. This allowance, as it is called, depends on (1) the purpose of the bearing, (2) the diameter of the shaft, (3) the length of the bearing, (4) the kind of metal used for each.

For an average length of bearing an allowance of .001" per inch of diameter of bearing is sufficient. A longer bearing requires usually a trifle more allowance.

The speed of the shaft is a factor in running fits and the necessity of high speeds and close running fits has developed bearing metals such as Babbitt, bronze and hardened steel which are much used in machine construction. Hardened and ground spindles in hardened and ground bearings require very little allowance, likewise hardened and ground shafts in bronze bearings. Unlike metals work best in running fits with the exception of hardened steel. Like metals may be used for sliding fits. The bar of a sliding fit is usually finished lengthwise by drawfiling to give it a longer life and a better appearance.
Drive Fit.—When two pieces are to retain indefinitely a fixed relative position, they may be so held by driving the one in the other as a key in a shaft, or a straight dowell pin in a hole. The allowance depends on the length and cross-section or diameter of the bearing surfaces, and the smoothness of the surface and the form of the surrounding part. The longer bearing surfaces and the larger diameter require less allowance; a carefully ground surface will hold better than a turned or filed surface and therefore does not require as much allowance; and a thin or weak pulley hub for example will not stand the driving allowance for a key that might be used in a heavy solid hub.

Force Fit.—When two pieces are to retain permanently a fixed relation a greater allowance than a drive fit may be used and the one part is forced into the other in a screw press or in a hydraulic press. For example, in forcing the axles into locomotive driving wheels, a pressure of 150 tons is not uncommon.

Shrink Fit.—When two pieces are to remain permanently together but the shape of one or both would make it impracticable or impossible to force one within the other, the enveloping piece is heated and thereby expanded sufficiently to slide over the other and then cooled slowly with water. Care must be taken when a piece is to be shrunk on another against a shoulder to prevent the piece shrinking away from the shoulder leaving a space between. Cool the part against the shoulder first and then gradually away from the shoulder.

FASTENING A BELT

There are many kinds of patented metal belt fasteners in the market. Some of them are quickly applied and very serviceable but none, except perhaps the wire lacing, or the Clipper lacing, is as flexible and smooth running as the old fashioned rawhide belt lacing, which is still very generally used. It is desirable that every boy in the shop should know how to lace a belt.

The tools used comprise a belt punch, a belt awl, a pair of pliers, a try square, and a sharp knife.
General Directions

1. Cut the ends of the belt square.
2. The "grain" side or hair side of the belt should run against the pulleys. (The grain side is the smooth side.)
3. The lacing should be crossed on the flesh side (outside) of the belt and not on the grain side (side toward pulley).
4. The holes in both ends should be exactly opposite.
5. The belt should not be too tight or it will injure both the belt and the bearings.
6. A belt is made of sections of leather lapped and cemented together. The belt should be put on so that the points of the laps will run against the pulley.

The following directions apply particularly to belts from 2" to 5" wide.

1. Put the belt around the pulleys and pull tight letting one end lap over the other and note the amount. A good strong pull will indicate the amount the belt must be shortened to give it the proper tension.
2. Lay off the amount to be trimmed with the point of the knife using the try-square as a guide, and then cut straight on the line.
3. Punch holes approximately $\frac{3}{16}$ diameter about $\frac{3}{4}''$ from center to center and not nearer than $\frac{1}{2}$ from the end or from the sides.
4. Select a lace, a trifle longer than is required, that will pull fairly easy through the holes. Butt the ends of the belt together with edges flush. Put lace up through holes 3 and 6 from the flesh side (see Fig. 223) pulling ends of lace even. Put lace down through 7, up through 4 (and pull tight using the pliers if necessary), down through 8, up through 4 once more, down through 8 again, up through 3, down through 7, up through 2. Punch a hole $\frac{1}{2}''$ back of hole 2 and pull the lace through this hole. Cut off lace leaving tab end of about $\frac{3}{8}''$. Cut lace nearly half through at the surface of the belt and twist tab end one-half turn, thereby fastening it.
Put lace b down through 2, up through 5, down through 1, up through 5, down through 1, up through 6, down through 2, and up through 7. Fasten lace directly back of 7.

Cementing Belts.—Cementing together the lapped ends of a belt, Fig. 224, is considered the best method of fastening. The ends are shaved down as shown so that the lapped portion is not noticeably thicker than the rest of the belt, and the shaved parts should be flat and fit snugly together.

Get two pieces of board about 6 and 12 inches long respectively (for a belt under 4" wide) and a little wider than the belt, and arranging the lap about the middle (lengthwise) of the longer board lightly clamp at each end. Then fix the lap exactly right, with the edges of the belt flush and straight and set the clamps tight.

![Diagram](image)

**Fig. 224.**

Put a piece of paper between the belt and the board and lifting the upper lap apply the cement to both shaved surfaces rubbing it in well and being careful not to put on too much. Make the joint tight by rubbing out the air pockets, perhaps with the face of a hammer, paying particular attention to the ends and edges. Put a piece of paper over the joint, then the short piece of board and clamp tightly the two boards with the belt between them. Allow to set for several hours at least, and overnight if convenient. The object of the paper is to keep the boards from sticking to the belt; what little paper cannot be torn off will soon wear off.

**GEOMETRICAL PROGRESSION**

A series of numbers is said to be in geometrical progression when any number in the series equals the preceding number multiplied by a given constant. For example 2, 4, 8, 16, 32 is
such a series and the constant is equal to 2. Any one of these numbers is equal to the preceding number multiplied by 2. Also, in other words, this constant equals the ratio of any number of the series to the next lower number. For example in the series given 16 divided by 8 is equal to 2.

The speed changes and usually the feed changes of most machine tools advance from the slowest to the fastest in Geometrical Progression. The reason for this is, that in this arrangement speeds for the larger diameters of work or cutters (milling cutters, drills, etc.) increase slowly giving a greater number of available speeds, and the speeds for the smaller diameters increase more rapidly which is as it should be as a comparatively small difference in diameter requires a considerable change in speeds. The ratio used is usually around 1.2.

THE VERNIER

The principle of vernier graduations was devised in 1631 by Pierre Vernier. There are two kinds of verniers, direct and retrograde. They are alike in principle but are arranged to read in opposite directions. The direct vernier is used in various machine shop measuring instruments and is here described.

A short scale is employed in connection with the regular graduated divisions of the measuring instrument for indicating parts of these divisions. It is so graduated that the space occupied by a certain convenient number of divisions is equal
to the space occupied by one less than that number of divisions on the true scale of the instrument. To illustrate the vernier principle: Lay off a distance of 9" on each of two strips of paper or card board and on one divide this distance into nine equal parts and on the other into ten equal parts to represent respectively the true scale and the vernier scale. (See Fig. 226.) It will be observed that the difference in length between these divisions is equal to $\frac{1}{10}$ of the division on the true scale or, $\frac{1}{10}$ of an inch. (9" divided into 9 equal parts; each part equals one inch. 9" divided into 10 equal parts; each part equals .9 inches. The difference is equal to 1" minus .9" = .1 of an inch.)

The difference between one division on the true scale and one division on the vernier scale is always equal to a division of the true scale divided by the number of divisions on the vernier scale. (In the above case one inch divided by ten equal $\frac{1}{10}$ of an inch). As a further example: suppose twenty-five divisions on the vernier scale occupy the same space as twenty-four divisions on a true scale, the difference between these two divisions is equal to one twenty-fifth ($\frac{1}{25}$) of the true scale division, whatever its length may be.

It will be observed that if the zero mark on the vernier scale (Fig. 226) coincides with the zero mark on the true scale, and the vernier scale is moved so that "1" coincides with a line above, it has moved $\frac{1}{10}$ of an inch. If "2" is exactly opposite a line above, it has moved $\frac{3}{10}$ of an inch. If it is moved so that 5 coincides with a line above it has moved $\frac{5}{10}$ of an inch; and so on. The distance that the zero on the vernier has been moved beyond the preceding line on the true scale will be indicated by the number of the line on the vernier scale that coincides with the line above.
The Vernier Caliper (Fig. 227), Depth Gauge, Height Gauge, and Gear Tooth Vernier

All of these instruments read in exactly the same way. The bar of the instrument is graduated into fortieths of an inch. Each inch is numbered and each tenth of an inch is numbered. On the sliding jaw is a vernier scale graduated into twenty-five parts numbered 0, 5, 10, 15, 20, 25. The twenty-five parts on the vernier correspond, in extreme length, with twenty-four fortieths of an inch on the bar, consequently each division on the vernier is smaller than each division on the bar by \( \frac{1}{25} \) of \( \frac{1}{40} \)" or one-thousandth part of an inch. To read the distance the caliper is open, commence by noticing how many inches, tenths and fortieths, the zero point on the vernier has been moved from the zero point on the bar. (The best way of expressing the value of the divisions on the bar is to call the tenths one hundred thousandths \( .100 \) and the fortieths
twenty-five thousandths (.025). Now find on the vernier scale the number of the divisions which is exactly opposite to a line on the bar and this will be the number of thousandths added to the distance read off on the bar.

Referring to Fig. 228 it will be observed, first, that the preliminary reading is one inch, four-tenths and one-fortieth which is equal to one inch and four hundred twenty-five thousandths (1.425), next that the eleventh division on the vernier scale coincides with the line on the regular scale above, and therefore eleven thousandths more must be added to the one inch and four hundred twenty-five thousandths which will make the reading one inch four hundred thirty-six thousandths total.

In making inside measurements with the 6" vernier two hundred and fifty thousandths of an inch, and with the twelve inch and larger verniers, three hundred thousandths of an inch, should be added to the apparent reading of the vernier side for the space occupied by the caliper points.

**Vernier For Universal Bevel Protractor**

The vernier indicates every five minutes (5') or one-twelfth of a degree. Twenty-four spaces on the vernier scale equal in extreme length twenty-three double degrees. Thus, the difference between the space occupied by two degrees on a regular scale and the space of the vernier scale is equal to one-twenty-fourth of two degrees or one-twelfth of one degree (or five minutes).

**To Read the Protractor Setting.**—Read off directly from the true scale the number of whole degrees between 0 on this scale and the 0 of the vernier scale. Then count, in the same direction, the number of spaces from the zero on the vernier scale to a line that coincides with a line on the regular scale; multiply this number by 5 and the product will be the number of minutes to be added to the whole number of degrees.

For example: In Fig. 229 the 0 on the vernier has moved 12 whole degrees to the right of the 0 on the regular scale and the 8th line on the vernier coincides with a line upon the regular scale as indicated by *. Multiplying 8 by 5, the product, 40, is the number of minutes to be added to the whole
number of degrees, thus indicating a setting of 12 degrees and 40 minutes (12° 40').

Fig. 229.

MICROMETER WITH VERNIER FOR TEN-THOUSANDTH GRADUATIONS

The reading in ten-thousandths of an inch is obtained by means of vernier graduations on the hub of the micrometer.

Fig. 230.—Ten-thousandths micrometer. (a) Development of vernier scale and beveled edge of thimble—ten spaces on vernier equals nine spaces on bevel. (b) Shows micrometer reading .2123 (.212 + .0003) since the "3" line on the vernier coincides with a line on the bevel.

These divisions are ten in number and occupy the same space as nine divisions on the thimble. (For convenience in reading, the graduations are numbered 0, 1, 2, 3, 4, 5, 6, 7, 8, 9, 0.) Accordingly when a line on the barrel coincides with the first line of the Vernier, the next two lines to the right differ from each other one-tenth of the length of a division on the thimble and indicate one-tenth of a thousandth, the next lines differ by two-tenths, etc.
To Read the Caliper.—Note the thousandths as usual and the tenths of a thousandth to add will be indicated by the number of the line in the vernier scale which coincides with a graduation on the barrel. In b, Fig. 230 the reading is .2123. It will be noted that without looking at the vernier reading it may be called a scant .2123½, but with the 3 line in the vernier scale coinciding with a line on the thimble the exact reading is obtained.

**SCREW THREAD MICROMETER CALIPER FIG. 231**

The distinctive feature in the construction of this Caliper is that the end of the movable spindle is pointed and the fixed end or “anvil” is V-shaped. Enough is taken from the end of the point and the bottom of the V is carried down low enough, so that they will not rest on the bottom or top of the thread to be measured but on the cu surface. As the thread itself is measured, it will be seen that the actual outside diameter of the piece does not enter into consideration.

As one-half of the depth of the thread from the top is measured on each side, the diameter of the thread as indicated by the caliper, or the pitch diameter, is the full size of the screw less the depth of one thread. When the spindle point and the anvil points are in contact the 0 represents a line drawn through the plane \( AB \) and if the caliper is opened, for example to .463” it represents the distance of the two planes .463” apart.

The pitch diameters for the various sizes of machine screws and standard screws are given in Tables 16, 17, and 18.

1 *Pitch diameter* of a thread is equal to the nominal outside diameter less the depth of one thread and may be found as follows:
- Depth of V thread equals \( .866 \div \text{No. of threads per inch} \).
- Depth of U. S. Std. thread equals \( .6495 \div \text{No. of threads per inch} \).
- Depth of Whitworth thread equals \( .640 \div \text{No. of threads per inch} \).
LIST OF TABLES

Table 1. Decimal Equivalents of Parts of an Inch.
Table 2. Cutting Speeds, Lathe Work, Drills, Milling Cutters.
Table 3. Morse Tapers.
Table 4. Brown and Sharpe Tapers.
Table 5. Taper Pins and Reamers.
Table 6. Tapers per Foot and Corresponding Angles.
Table 7. U. S. Standard Screw Threads.
Table 8. Sharp "V" Screw Threads.
Table 9. Whitworth Standard Screw Threads.
Table 10. British Association Screw Threads.
Table 11. French (Metric) Standard Screw Threads.
Table 12. International Standard Screw Threads.
Table 13. Acme 29° Screw Threads.
Table 14. Acme 29° Tap Threads.
Table 15. Brown and Sharpe 29° Worm Thread.
Table 16. Machine Screw Sizes.
Table 17. U. S. Standard Screw Sizes.
Table 18. S. A. E. Standard Screw Sizes.
Table 19. U. S. Standard Bolts and Nuts.
Table 20. Wrought Iron Pipe Threads.
Table 21. Metric System of Measurement.
Table 22. Metric Conversion Table.
Table 23. Decimal Equivalents of Number and Letter Sizes of Drills.
Table 1.—Decimal Equivalents of Parts of an Inch

<table>
<thead>
<tr>
<th>Fraction</th>
<th>Decimal</th>
<th>Fraction</th>
<th>Decimal</th>
<th>Fraction</th>
<th>Decimal</th>
</tr>
</thead>
<tbody>
<tr>
<td>1/64</td>
<td>.01563</td>
<td>21/64</td>
<td>.32813</td>
<td>45/64</td>
<td>.70313</td>
</tr>
<tr>
<td>3/32</td>
<td>.03125</td>
<td>11/32</td>
<td>.34375</td>
<td>23/32</td>
<td>.71875</td>
</tr>
<tr>
<td>7/64</td>
<td>.04688</td>
<td>23/64</td>
<td>.35938</td>
<td>47/64</td>
<td>.73438</td>
</tr>
<tr>
<td>23/64</td>
<td>.07813</td>
<td>25/64</td>
<td>.39063</td>
<td>49/64</td>
<td>.76563</td>
</tr>
<tr>
<td>7/32</td>
<td>.09375</td>
<td>13/32</td>
<td>.40625</td>
<td>29/32</td>
<td>.78125</td>
</tr>
<tr>
<td>5/16</td>
<td>.10938</td>
<td>27/64</td>
<td>.42188</td>
<td>51/64</td>
<td>.79688</td>
</tr>
<tr>
<td>9/64</td>
<td>.14063</td>
<td>29/64</td>
<td>.45313</td>
<td>53/64</td>
<td>.82813</td>
</tr>
<tr>
<td>15/64</td>
<td>.15625</td>
<td>15/8</td>
<td>.46875</td>
<td>27/8</td>
<td>.84375</td>
</tr>
<tr>
<td>11/64</td>
<td>.17188</td>
<td>31/64</td>
<td>.48438</td>
<td>55/64</td>
<td>.85938</td>
</tr>
<tr>
<td>1/4</td>
<td>.1875</td>
<td>1/2</td>
<td>.5</td>
<td>5/8</td>
<td>.875</td>
</tr>
<tr>
<td>13/64</td>
<td>.20313</td>
<td>33/64</td>
<td>.51563</td>
<td>57/64</td>
<td>.89063</td>
</tr>
<tr>
<td>5/16</td>
<td>.23438</td>
<td>35/64</td>
<td>.54688</td>
<td>59/8</td>
<td>.92188</td>
</tr>
<tr>
<td>1/2</td>
<td>.25</td>
<td>7/16</td>
<td>.5625</td>
<td>15/16</td>
<td>.9375</td>
</tr>
<tr>
<td>17/64</td>
<td>.26563</td>
<td>37/64</td>
<td>.57813</td>
<td>61/64</td>
<td>.95313</td>
</tr>
<tr>
<td>19/64</td>
<td>.29688</td>
<td>39/64</td>
<td>.60938</td>
<td>63/64</td>
<td>.98438</td>
</tr>
<tr>
<td>5/16</td>
<td>.3125</td>
<td>5/8</td>
<td>.625</td>
<td>1</td>
<td>1.00000</td>
</tr>
<tr>
<td>41/64</td>
<td></td>
<td>21/8</td>
<td>.64063</td>
<td></td>
<td></td>
</tr>
<tr>
<td>43/64</td>
<td></td>
<td>43/8</td>
<td>.65625</td>
<td></td>
<td></td>
</tr>
<tr>
<td>11/16</td>
<td></td>
<td></td>
<td>.67188</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>.6875</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Dia.</td>
<td>1/4</td>
<td>3/8</td>
<td>1/2</td>
<td>5/8</td>
<td>3/4</td>
</tr>
<tr>
<td>------</td>
<td>-----</td>
<td>-----</td>
<td>-----</td>
<td>-----</td>
<td>-----</td>
</tr>
<tr>
<td>20</td>
<td>306</td>
<td>204</td>
<td>153</td>
<td>122</td>
<td>102</td>
</tr>
<tr>
<td>30</td>
<td>458</td>
<td>306</td>
<td>229</td>
<td>183</td>
<td>153</td>
</tr>
<tr>
<td>40</td>
<td>611</td>
<td>407</td>
<td>306</td>
<td>244</td>
<td>204</td>
</tr>
<tr>
<td>50</td>
<td>764</td>
<td>509</td>
<td>382</td>
<td>306</td>
<td>255</td>
</tr>
<tr>
<td>60</td>
<td>916</td>
<td>612</td>
<td>458</td>
<td>366</td>
<td>306</td>
</tr>
<tr>
<td>70</td>
<td>1070</td>
<td>712</td>
<td>534</td>
<td>428</td>
<td>356</td>
</tr>
<tr>
<td>80</td>
<td>1222</td>
<td>814</td>
<td>612</td>
<td>488</td>
<td>408</td>
</tr>
<tr>
<td>90</td>
<td>1376</td>
<td>916</td>
<td>704</td>
<td>550</td>
<td>458</td>
</tr>
<tr>
<td>100</td>
<td>1528</td>
<td>1019</td>
<td>704</td>
<td>612</td>
<td>509</td>
</tr>
<tr>
<td>110</td>
<td>1681</td>
<td>1120</td>
<td>704</td>
<td>612</td>
<td>509</td>
</tr>
<tr>
<td>120</td>
<td>1833</td>
<td>1222</td>
<td>704</td>
<td>612</td>
<td>509</td>
</tr>
<tr>
<td>130</td>
<td>1986</td>
<td>1324</td>
<td>704</td>
<td>612</td>
<td>509</td>
</tr>
<tr>
<td>140</td>
<td>2139</td>
<td>1426</td>
<td>704</td>
<td>612</td>
<td>509</td>
</tr>
<tr>
<td>150</td>
<td>2292</td>
<td>1528</td>
<td>704</td>
<td>612</td>
<td>509</td>
</tr>
<tr>
<td>160</td>
<td>2462</td>
<td>1632</td>
<td>704</td>
<td>612</td>
<td>509</td>
</tr>
<tr>
<td>170</td>
<td>2615</td>
<td>1735</td>
<td>704</td>
<td>612</td>
<td>509</td>
</tr>
<tr>
<td>180</td>
<td>2780</td>
<td>1836</td>
<td>704</td>
<td>612</td>
<td>509</td>
</tr>
</tbody>
</table>

**Table 2. Cutting Speeds Lathe Work, Drills, Milling Cutters.**

Formulas: $C.S. = .26D \times \text{r.p.m.}$ and $\text{r.p.m.} = \frac{C.S.}{.26D}$

<table>
<thead>
<tr>
<th>Cutting speeds in feet per minute</th>
</tr>
</thead>
<tbody>
<tr>
<td>20</td>
</tr>
<tr>
<td>-----</td>
</tr>
<tr>
<td>Dia.</td>
</tr>
<tr>
<td>------</td>
</tr>
<tr>
<td>20</td>
</tr>
<tr>
<td>30</td>
</tr>
<tr>
<td>40</td>
</tr>
<tr>
<td>50</td>
</tr>
<tr>
<td>60</td>
</tr>
<tr>
<td>70</td>
</tr>
<tr>
<td>80</td>
</tr>
<tr>
<td>90</td>
</tr>
<tr>
<td>100</td>
</tr>
<tr>
<td>110</td>
</tr>
<tr>
<td>120</td>
</tr>
<tr>
<td>130</td>
</tr>
<tr>
<td>140</td>
</tr>
<tr>
<td>150</td>
</tr>
<tr>
<td>160</td>
</tr>
<tr>
<td>170</td>
</tr>
<tr>
<td>180</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Revolutions per minute</th>
</tr>
</thead>
<tbody>
<tr>
<td>1/4</td>
</tr>
<tr>
<td>-----</td>
</tr>
<tr>
<td>20</td>
</tr>
<tr>
<td>30</td>
</tr>
<tr>
<td>40</td>
</tr>
<tr>
<td>50</td>
</tr>
<tr>
<td>60</td>
</tr>
<tr>
<td>70</td>
</tr>
<tr>
<td>80</td>
</tr>
<tr>
<td>90</td>
</tr>
<tr>
<td>100</td>
</tr>
<tr>
<td>110</td>
</tr>
<tr>
<td>120</td>
</tr>
<tr>
<td>130</td>
</tr>
<tr>
<td>140</td>
</tr>
<tr>
<td>150</td>
</tr>
<tr>
<td>160</td>
</tr>
<tr>
<td>170</td>
</tr>
<tr>
<td>180</td>
</tr>
</tbody>
</table>

**Digitized by Google**
<table>
<thead>
<tr>
<th>Number of taper</th>
<th>Diam. of plug at small end</th>
<th>Diam. at end of socket</th>
<th>Shank</th>
<th>Whole length of shank</th>
<th>Shank depth</th>
<th>Depth of hole</th>
<th>Standard plug depth</th>
<th>Tongue</th>
<th>Thickness of tongue</th>
<th>Length of tongue</th>
<th>Radius of mill for tongue</th>
<th>Diameter of tongue</th>
<th>Radius of tongue</th>
<th>Keyway</th>
<th>Width of keyway</th>
<th>Length of keyway</th>
<th>End of socket to keyway</th>
<th>Taper per foot</th>
<th>Taper per inch</th>
<th>Number of key</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0.252</td>
<td>0.3561</td>
<td>2(\frac{1}{4})</td>
<td>2(\frac{1}{4})</td>
<td>2(\frac{1}{4})</td>
<td>2(\frac{1}{4})</td>
<td>5(\frac{3}{8})</td>
<td>3(\frac{1}{4})</td>
<td>2(\frac{1}{2})</td>
<td>3(\frac{1}{8})</td>
<td>0.235</td>
<td>0.04</td>
<td>0.160</td>
<td>9(\frac{3}{8})</td>
<td>1(\frac{1}{4})</td>
<td>0.625</td>
<td>0.05208</td>
<td>0</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>0.369</td>
<td>0.475</td>
<td>2(\frac{1}{4})</td>
<td>3(\frac{1}{4})</td>
<td>3(\frac{1}{4})</td>
<td>2(\frac{1}{4})</td>
<td>2(\frac{1}{2})</td>
<td>1(\frac{1}{4})</td>
<td>2(\frac{1}{2})</td>
<td>2(\frac{1}{2})</td>
<td>0.343</td>
<td>0.06</td>
<td>0.213</td>
<td>8(\frac{3}{8})</td>
<td>2(\frac{1}{4})</td>
<td>0.600</td>
<td>0.05</td>
<td>1</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>0.572</td>
<td>0.700</td>
<td>3(\frac{1}{4})</td>
<td>3(\frac{1}{4})</td>
<td>3(\frac{1}{4})</td>
<td>3(\frac{1}{4})</td>
<td>3(\frac{1}{4})</td>
<td>2(\frac{1}{2})</td>
<td>3(\frac{1}{4})</td>
<td>3(\frac{1}{4})</td>
<td>0.260</td>
<td>0.06</td>
<td>0.260</td>
<td>7(\frac{3}{8})</td>
<td>2(\frac{1}{4})</td>
<td>0.602</td>
<td>0.05016</td>
<td>2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>0.778</td>
<td>0.938</td>
<td>3(\frac{1}{4})</td>
<td>3(\frac{1}{4})</td>
<td>3(\frac{1}{4})</td>
<td>3(\frac{1}{4})</td>
<td>3(\frac{1}{4})</td>
<td>2(\frac{1}{2})</td>
<td>3(\frac{1}{4})</td>
<td>3(\frac{1}{4})</td>
<td>0.322</td>
<td>0.08</td>
<td>0.322</td>
<td>7(\frac{3}{8})</td>
<td>3(\frac{1}{4})</td>
<td>0.602</td>
<td>0.05016</td>
<td>3</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>1.020</td>
<td>1.231</td>
<td>4(\frac{1}{8})</td>
<td>4(\frac{1}{8})</td>
<td>4(\frac{1}{8})</td>
<td>4(\frac{1}{8})</td>
<td>4(\frac{1}{8})</td>
<td>3(\frac{1}{8})</td>
<td>3(\frac{1}{8})</td>
<td>3(\frac{1}{8})</td>
<td>0.478</td>
<td>0.10</td>
<td>0.478</td>
<td>6(\frac{1}{8})</td>
<td>3(\frac{1}{8})</td>
<td>0.623</td>
<td>0.05191</td>
<td>4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>1.475</td>
<td>1.748</td>
<td>6(\frac{1}{8})</td>
<td>5(\frac{1}{8})</td>
<td>5(\frac{1}{8})</td>
<td>5(\frac{1}{8})</td>
<td>5(\frac{1}{8})</td>
<td>4(\frac{1}{8})</td>
<td>4(\frac{1}{8})</td>
<td>4(\frac{1}{8})</td>
<td>0.635</td>
<td>0.12</td>
<td>0.635</td>
<td>5(\frac{1}{8})</td>
<td>4(\frac{1}{8})</td>
<td>0.630</td>
<td>0.0525</td>
<td>5</td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>2.116</td>
<td>2.494</td>
<td>8(\frac{1}{4})</td>
<td>7(\frac{1}{4})</td>
<td>7(\frac{1}{4})</td>
<td>7(\frac{1}{4})</td>
<td>7(\frac{1}{4})</td>
<td>5(\frac{1}{4})</td>
<td>5(\frac{1}{4})</td>
<td>5(\frac{1}{4})</td>
<td>0.760</td>
<td>0.15</td>
<td>0.760</td>
<td>6(\frac{1}{4})</td>
<td>5(\frac{1}{4})</td>
<td>0.626</td>
<td>0.05216</td>
<td>6</td>
<td></td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>2.750</td>
<td>3.270</td>
<td>11(\frac{3}{4})</td>
<td>10(\frac{3}{4})</td>
<td>10(\frac{3}{4})</td>
<td>10(\frac{3}{4})</td>
<td>10(\frac{3}{4})</td>
<td>6(\frac{1}{4})</td>
<td>6(\frac{1}{4})</td>
<td>6(\frac{1}{4})</td>
<td>1.135</td>
<td>0.18</td>
<td>1.135</td>
<td>9(\frac{3}{4})</td>
<td>6(\frac{1}{4})</td>
<td>0.625</td>
<td>0.05208</td>
<td>7</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Taper Per Foot</td>
<td>0.416</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>---------------</td>
<td>-------</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Shank Depth</th>
<th>S</th>
<th>1.77</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Radius of Round</th>
<th>R</th>
<th>1.23</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Radius of Mill</th>
<th>R</th>
<th>1.23</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Round Length</th>
<th>T</th>
<th>6.00</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Width of Key-Eye</th>
<th>W</th>
<th>1.35</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Key-Eye Length</th>
<th>L</th>
<th>1.35</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Depth of Hole</th>
<th>D</th>
<th>1.00</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Standard Plain Depth</th>
<th>D</th>
<th>1.00</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Number of Taps</th>
<th>D</th>
<th>1.00</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>5.00</td>
<td>5.00</td>
<td></td>
</tr>
</tbody>
</table>

Table 4: Brown and Shapere Tapers

- Width of Key-Eye: W
- Key-Eye Length: L
- Depth of Hole: D
- Standard Plain Depth: D
- Number of Taps: D

Digitalized by Google
Table 5.—Taper Pins and Reamers

(PRATT & WHITNEY CO.)

Taper = ¾ inch per foot or .0208 inch per inch

<table>
<thead>
<tr>
<th>Size No.</th>
<th>Dia. of Small End of Reamer</th>
<th>Dia. of Large End of Reamer</th>
<th>Length of Flute</th>
<th>Total Length of Reamer</th>
<th>Size Drill for Reamer</th>
<th>Longest Limit Length of Pin</th>
<th>Dia. of Large End of Pin</th>
<th>Approx. Fractional Size at Large End of Pin</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0.135&quot;</td>
<td>.162&quot;</td>
<td>2&quot;</td>
<td>28&quot;</td>
<td>25&quot;</td>
<td>1&quot;</td>
<td>.156&quot;</td>
<td>.146&quot;</td>
</tr>
<tr>
<td>1</td>
<td>.146&quot;</td>
<td>.170&quot;</td>
<td>2 1/2&quot;</td>
<td>28 1/2&quot;</td>
<td>25 1/2&quot;</td>
<td>1 1/2&quot;</td>
<td>.172&quot;</td>
<td>.166&quot;</td>
</tr>
<tr>
<td>2</td>
<td>.162&quot;</td>
<td>.200&quot;</td>
<td>3&quot;</td>
<td>21 1/2&quot;</td>
<td>19&quot;</td>
<td>1 1/2&quot;</td>
<td>.193&quot;</td>
<td>.183&quot;</td>
</tr>
<tr>
<td>3</td>
<td>.183&quot;</td>
<td>.226&quot;</td>
<td>3 1/2&quot;</td>
<td>31 1/2&quot;</td>
<td>12&quot;</td>
<td>1 1/2&quot;</td>
<td>.219&quot;</td>
<td>.210&quot;</td>
</tr>
<tr>
<td>4</td>
<td>.208&quot;</td>
<td>.257&quot;</td>
<td>4&quot;</td>
<td>31 1/2&quot;</td>
<td>3&quot;</td>
<td>2&quot;</td>
<td>.256&quot;</td>
<td>.245&quot;</td>
</tr>
<tr>
<td>5</td>
<td>.240&quot;</td>
<td>.300&quot;</td>
<td>5&quot;</td>
<td>48&quot;</td>
<td>21&quot;</td>
<td>3&quot;</td>
<td>.285&quot;</td>
<td>.274&quot;</td>
</tr>
<tr>
<td>6</td>
<td>.279&quot;</td>
<td>.354&quot;</td>
<td>6&quot;</td>
<td>61 1/2&quot;</td>
<td>33&quot;</td>
<td>4&quot;</td>
<td>.341&quot;</td>
<td>.330&quot;</td>
</tr>
<tr>
<td>7</td>
<td>.331&quot;</td>
<td>.423&quot;</td>
<td>7 1/2&quot;</td>
<td>81 1/2&quot;</td>
<td>43&quot;</td>
<td>6&quot;</td>
<td>.409&quot;</td>
<td>.404&quot;</td>
</tr>
<tr>
<td>8</td>
<td>.398&quot;</td>
<td>.507&quot;</td>
<td>9&quot;</td>
<td>101&quot;</td>
<td>53&quot;</td>
<td>7&quot;</td>
<td>.492&quot;</td>
<td>.487&quot;</td>
</tr>
<tr>
<td>9</td>
<td>.482&quot;</td>
<td>.600&quot;</td>
<td>10 1/2&quot;</td>
<td>121 1/2&quot;</td>
<td>73&quot;</td>
<td>8&quot;</td>
<td>.591&quot;</td>
<td>.586&quot;</td>
</tr>
<tr>
<td>10</td>
<td>.581&quot;</td>
<td>.727&quot;</td>
<td>12&quot;</td>
<td>141 1/2&quot;</td>
<td>93&quot;</td>
<td>9&quot;</td>
<td>.706&quot;</td>
<td>.702&quot;</td>
</tr>
<tr>
<td>11</td>
<td>.706&quot;</td>
<td>.878&quot;</td>
<td>13 1/2&quot;</td>
<td>161 1/2&quot;</td>
<td>113&quot;</td>
<td>10&quot;</td>
<td>.857&quot;</td>
<td>.854&quot;</td>
</tr>
<tr>
<td>12</td>
<td>.842&quot;</td>
<td>1.050&quot;</td>
<td>15&quot;</td>
<td>181 1/2&quot;</td>
<td>133&quot;</td>
<td>11&quot;</td>
<td>1.013&quot;</td>
<td>1.010&quot;</td>
</tr>
<tr>
<td>13</td>
<td>1.009&quot;</td>
<td>1.259&quot;</td>
<td>16&quot;</td>
<td>201 1/2&quot;</td>
<td>153&quot;</td>
<td>12&quot;</td>
<td>1.233&quot;</td>
<td>1.230&quot;</td>
</tr>
</tbody>
</table>

These reamer sizes are so proportioned that each overlaps the size smaller about ½ inch.
### Table 6.—Tapers per Foot and Corresponding Angles

<table>
<thead>
<tr>
<th>Taper per foot, in.</th>
<th>Included angle</th>
<th>Angle with center line</th>
<th>Taper per foot, in.</th>
<th>Included angle</th>
<th>Angle with center line</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\frac{1}{16}$</td>
<td>$0^\circ-18'$</td>
<td>$0^\circ-09'$</td>
<td>$1\frac{1}{8}$</td>
<td>$5^\circ-22'$</td>
<td>$2^\circ-41'$</td>
</tr>
<tr>
<td>$\frac{1}{8}$</td>
<td>$0^\circ-36'$</td>
<td>$0^\circ-18'$</td>
<td>$1\frac{1}{4}$</td>
<td>$5^\circ-57\frac{1}{2}'$</td>
<td>$2^\circ-58\frac{3}{4}'$</td>
</tr>
<tr>
<td>$\frac{1}{16}$</td>
<td>$0^\circ-53\frac{1}{2}'$</td>
<td>$0^\circ-26\frac{1}{4}'$</td>
<td>$1\frac{1}{8}$</td>
<td>$6^\circ-33\frac{1}{2}'$</td>
<td>$3^\circ-16\frac{3}{4}'$</td>
</tr>
<tr>
<td>$\frac{1}{4}$</td>
<td>$1^\circ-11\frac{1}{2}'$</td>
<td>$0^\circ-35\frac{3}{4}'$</td>
<td>$1\frac{1}{2}$</td>
<td>$7^\circ-09'$</td>
<td>$3^\circ-34\frac{1}{2}'$</td>
</tr>
<tr>
<td>$\frac{5}{32}$</td>
<td>$1^\circ-29\frac{1}{2}'$</td>
<td>$0^\circ-44\frac{3}{4}'$</td>
<td>$1\frac{1}{8}$</td>
<td>$7^\circ-45'$</td>
<td>$3^\circ-52\frac{3}{4}'$</td>
</tr>
<tr>
<td>$\frac{1}{8}$</td>
<td>$1^\circ-47\frac{1}{2}'$</td>
<td>$0^\circ-53\frac{3}{4}'$</td>
<td>$1\frac{1}{4}$</td>
<td>$8^\circ-20\frac{1}{2}'$</td>
<td>$4^\circ-10\frac{1}{4}'$</td>
</tr>
<tr>
<td>$\frac{1}{16}$</td>
<td>$2^\circ-05\frac{1}{2}'$</td>
<td>$1^\circ-02\frac{3}{4}'$</td>
<td>$1\frac{1}{8}$</td>
<td>$8^\circ-56'$</td>
<td>$4^\circ-28'$</td>
</tr>
<tr>
<td>$\frac{1}{4}$</td>
<td>$2^\circ-23'$</td>
<td>$1^\circ-11\frac{1}{2}'$</td>
<td>$2$</td>
<td>$9^\circ-31\frac{1}{2}'$</td>
<td>$4^\circ-45\frac{3}{4}'$</td>
</tr>
<tr>
<td>$\frac{5}{8}$</td>
<td>$2^\circ-41'$</td>
<td>$1^\circ-20\frac{1}{2}'$</td>
<td>$2\frac{1}{4}$</td>
<td>$10^\circ-43\frac{1}{2}'$</td>
<td>$5^\circ-21\frac{3}{4}'$</td>
</tr>
<tr>
<td>$\frac{1}{4}$</td>
<td>$2^\circ-59'$</td>
<td>$1^\circ-29\frac{1}{2}'$</td>
<td>$2\frac{1}{2}$</td>
<td>$11^\circ-53\frac{3}{4}'$</td>
<td>$5^\circ-56\frac{3}{4}'$</td>
</tr>
<tr>
<td>$\frac{1}{16}$</td>
<td>$3^\circ-10\frac{1}{2}'$</td>
<td>$1^\circ-38\frac{1}{4}'$</td>
<td>$2\frac{3}{8}$</td>
<td>$13^\circ-09\frac{3}{4}'$</td>
<td>$6^\circ-34\frac{3}{4}'$</td>
</tr>
<tr>
<td>$\frac{1}{8}$</td>
<td>$3^\circ-35'$</td>
<td>$1^\circ-47\frac{1}{2}'$</td>
<td>$3$</td>
<td>$14^\circ-15'$</td>
<td>$7^\circ-7\frac{1}{2}'$</td>
</tr>
<tr>
<td>$\frac{5}{16}$</td>
<td>$3^\circ-53'$</td>
<td>$1^\circ-56\frac{1}{2}'$</td>
<td>$3\frac{1}{2}$</td>
<td>$16^\circ-35\frac{1}{2}'$</td>
<td>$8^\circ-17\frac{1}{4}'$</td>
</tr>
<tr>
<td>$\frac{1}{4}$</td>
<td>$4^\circ-10\frac{1}{2}'$</td>
<td>$2^\circ-05\frac{1}{2}'$</td>
<td>$4$</td>
<td>$18^\circ-55\frac{1}{2}'$</td>
<td>$9^\circ-27\frac{3}{4}'$</td>
</tr>
<tr>
<td>$\frac{1}{16}$</td>
<td>$4^\circ-28\frac{1}{2}'$</td>
<td>$2^\circ-14\frac{1}{2}'$</td>
<td>$4\frac{1}{2}$</td>
<td>$21^\circ-37'$</td>
<td>$10^\circ-48\frac{1}{2}'$</td>
</tr>
<tr>
<td>$1$</td>
<td>$4^\circ-46\frac{1}{2}'$</td>
<td>$2^\circ-23\frac{1}{4}'$</td>
<td>$5$</td>
<td>$23^\circ-32'$</td>
<td>$11^\circ-46'$</td>
</tr>
</tbody>
</table>
### Table 7.—U. S. Standard Screw Threads

<table>
<thead>
<tr>
<th>Diam. of Screw</th>
<th>Threads to Inch</th>
<th>Pitch</th>
<th>Depth of Thread</th>
<th>Diam. at Root of Thread</th>
<th>Width of Flat</th>
</tr>
</thead>
<tbody>
<tr>
<td>1/8</td>
<td>20</td>
<td>.0500</td>
<td>.0325</td>
<td>.185</td>
<td>.0063</td>
</tr>
<tr>
<td>3/32</td>
<td>18</td>
<td>.0556</td>
<td>.0361</td>
<td>.2403</td>
<td>.0069</td>
</tr>
<tr>
<td>1/16</td>
<td>16</td>
<td>.0625</td>
<td>.0405</td>
<td>.2936</td>
<td>.0078</td>
</tr>
<tr>
<td>5/64</td>
<td>14</td>
<td>.0714</td>
<td>.0464</td>
<td>.3447</td>
<td>.0089</td>
</tr>
<tr>
<td>3/32</td>
<td>13</td>
<td>.0769</td>
<td>.0499</td>
<td>.4001</td>
<td>.0096</td>
</tr>
<tr>
<td>7/64</td>
<td>12</td>
<td>.0833</td>
<td>.0541</td>
<td>.4542</td>
<td>.0104</td>
</tr>
<tr>
<td>1/8</td>
<td>11</td>
<td>.0909</td>
<td>.0591</td>
<td>.5069</td>
<td>.0114</td>
</tr>
<tr>
<td>9/64</td>
<td>10</td>
<td>.1000</td>
<td>.0649</td>
<td>.6201</td>
<td>.0125</td>
</tr>
<tr>
<td>5/32</td>
<td>9</td>
<td>.1111</td>
<td>.0721</td>
<td>.7307</td>
<td>.0139</td>
</tr>
<tr>
<td>7/32</td>
<td>8</td>
<td>.1250</td>
<td>.0812</td>
<td>.8376</td>
<td>.0156</td>
</tr>
<tr>
<td>11/64</td>
<td>7</td>
<td>.1392</td>
<td>.0928</td>
<td>.9394</td>
<td>.0179</td>
</tr>
<tr>
<td>1/4</td>
<td>6</td>
<td>.1567</td>
<td>.1082</td>
<td>1.1585</td>
<td>.0179</td>
</tr>
<tr>
<td>5/32</td>
<td>5</td>
<td>.1667</td>
<td>.1082</td>
<td>1.2835</td>
<td>.0208</td>
</tr>
<tr>
<td>9/64</td>
<td>4</td>
<td>.1818</td>
<td>.1181</td>
<td>1.3688</td>
<td>.0227</td>
</tr>
<tr>
<td>7/32</td>
<td>3</td>
<td>.2000</td>
<td>.1299</td>
<td>1.4902</td>
<td>.0250</td>
</tr>
<tr>
<td>11/64</td>
<td>2</td>
<td>.2000</td>
<td>.1299</td>
<td>1.6152</td>
<td>.0250</td>
</tr>
<tr>
<td>13/64</td>
<td>2</td>
<td>.2222</td>
<td>.1444</td>
<td>1.7113</td>
<td>.0278</td>
</tr>
<tr>
<td>15/64</td>
<td>2</td>
<td>.2222</td>
<td>.1444</td>
<td>1.9613</td>
<td>.0278</td>
</tr>
<tr>
<td>1/2</td>
<td>2</td>
<td>.2500</td>
<td>.1624</td>
<td>2.1752</td>
<td>.0313</td>
</tr>
<tr>
<td>3/16</td>
<td>2</td>
<td>.2500</td>
<td>.1624</td>
<td>2.4252</td>
<td>.0313</td>
</tr>
<tr>
<td>7/32</td>
<td>2</td>
<td>.2857</td>
<td>.1856</td>
<td>2.5888</td>
<td>.0357</td>
</tr>
<tr>
<td>9/32</td>
<td>2</td>
<td>.2857</td>
<td>.1856</td>
<td>2.8788</td>
<td>.0357</td>
</tr>
<tr>
<td>11/32</td>
<td>2</td>
<td>.3077</td>
<td>.1998</td>
<td>3.1003</td>
<td>.0385</td>
</tr>
<tr>
<td>13/32</td>
<td>2</td>
<td>.3333</td>
<td>.2165</td>
<td>3.3170</td>
<td>.0417</td>
</tr>
<tr>
<td>15/32</td>
<td>2</td>
<td>.3333</td>
<td>.2165</td>
<td>3.5670</td>
<td>.0417</td>
</tr>
<tr>
<td>1/2</td>
<td>2</td>
<td>.3478</td>
<td>.2259</td>
<td>3.7982</td>
<td>.0435</td>
</tr>
<tr>
<td>5/32</td>
<td>2</td>
<td>.3636</td>
<td>.2362</td>
<td>4.0276</td>
<td>.0455</td>
</tr>
<tr>
<td>7/32</td>
<td>2</td>
<td>.3810</td>
<td>.2474</td>
<td>4.2511</td>
<td>.0476</td>
</tr>
<tr>
<td>9/32</td>
<td>2</td>
<td>.4000</td>
<td>.2598</td>
<td>4.4804</td>
<td>.0500</td>
</tr>
<tr>
<td>11/32</td>
<td>2</td>
<td>.4000</td>
<td>.2598</td>
<td>4.7304</td>
<td>.0500</td>
</tr>
<tr>
<td>13/32</td>
<td>2</td>
<td>.4210</td>
<td>.2735</td>
<td>5.2030</td>
<td>.0526</td>
</tr>
<tr>
<td>15/32</td>
<td>2</td>
<td>.4210</td>
<td>.2735</td>
<td>5.4226</td>
<td>.0556</td>
</tr>
</tbody>
</table>
### Table 8.—Sharp "V" Screw Threads (Theoretical)

![Diagram of sharp V screw threads with formulas for pitch and depth]

The formulas are given as:

- \( p = \text{Pitch} = \frac{1}{\text{No. Threads per Inch}} \)
- \( d = \text{Depth} = p \times .0600 \)

<table>
<thead>
<tr>
<th>Diam. of Screw</th>
<th>No. Threads per Inch</th>
<th>Pitch</th>
<th>Depth of Thread</th>
<th>Diam. at Root of Thread</th>
</tr>
</thead>
<tbody>
<tr>
<td>1/8</td>
<td>20</td>
<td>.0500</td>
<td>.0433</td>
<td>.1634</td>
</tr>
<tr>
<td>1/8</td>
<td>18</td>
<td>.0556</td>
<td>.0481</td>
<td>.2163</td>
</tr>
<tr>
<td>1/8</td>
<td>16</td>
<td>.0625</td>
<td>.0541</td>
<td>.2667</td>
</tr>
<tr>
<td>1/8</td>
<td>14</td>
<td>.0714</td>
<td>.0618</td>
<td>.3140</td>
</tr>
<tr>
<td>1/8</td>
<td>12</td>
<td>.0833</td>
<td>.0722</td>
<td>.3557</td>
</tr>
<tr>
<td>1/8</td>
<td>12</td>
<td>.0833</td>
<td>.0722</td>
<td>.4182</td>
</tr>
<tr>
<td>1/8</td>
<td>11</td>
<td>.0909</td>
<td>.0787</td>
<td>.4676</td>
</tr>
<tr>
<td>1/8</td>
<td>10</td>
<td>.1000</td>
<td>.0866</td>
<td>.5301</td>
</tr>
<tr>
<td>1/8</td>
<td>10</td>
<td>.1000</td>
<td>.0866</td>
<td>.5768</td>
</tr>
<tr>
<td>1/8</td>
<td>9</td>
<td>.1111</td>
<td>.0962</td>
<td>.6303</td>
</tr>
<tr>
<td>1/8</td>
<td>9</td>
<td>.1111</td>
<td>.0962</td>
<td>.6826</td>
</tr>
<tr>
<td>1/8</td>
<td>8</td>
<td>.1250</td>
<td>.1083</td>
<td>.7451</td>
</tr>
<tr>
<td>1/8</td>
<td>7</td>
<td>.1429</td>
<td>.1237</td>
<td>.7835</td>
</tr>
<tr>
<td>1/8</td>
<td>7</td>
<td>.1429</td>
<td>.1237</td>
<td>.8776</td>
</tr>
<tr>
<td>1/8</td>
<td>6</td>
<td>.1667</td>
<td>.1443</td>
<td>1.0026</td>
</tr>
<tr>
<td>1/8</td>
<td>6</td>
<td>.1667</td>
<td>.1443</td>
<td>1.0864</td>
</tr>
<tr>
<td>1/8</td>
<td>5</td>
<td>.2000</td>
<td>.1733</td>
<td>1.2114</td>
</tr>
<tr>
<td>1/8</td>
<td>5</td>
<td>.2000</td>
<td>.1733</td>
<td>1.2784</td>
</tr>
<tr>
<td>1/8</td>
<td>4(\frac{1}{2})</td>
<td>.2222</td>
<td>.1924</td>
<td>1.4034</td>
</tr>
<tr>
<td>1/8</td>
<td>4(\frac{1}{2})</td>
<td>.2222</td>
<td>.1924</td>
<td>1.4902</td>
</tr>
<tr>
<td>1/8</td>
<td>4(\frac{1}{2})</td>
<td>.2222</td>
<td>.1924</td>
<td>1.6152</td>
</tr>
<tr>
<td>1/8</td>
<td>4(\frac{1}{2})</td>
<td>.2222</td>
<td>.1924</td>
<td>1.7402</td>
</tr>
<tr>
<td>1/8</td>
<td>4(\frac{1}{2})</td>
<td>.2222</td>
<td>.1924</td>
<td>1.8652</td>
</tr>
<tr>
<td>1/8</td>
<td>4(\frac{1}{2})</td>
<td>.2222</td>
<td>.1924</td>
<td>1.9902</td>
</tr>
<tr>
<td>1/8</td>
<td>4</td>
<td>.2500</td>
<td>.2165</td>
<td>2.0670</td>
</tr>
<tr>
<td>1/8</td>
<td>4</td>
<td>.2500</td>
<td>.2165</td>
<td>2.1920</td>
</tr>
<tr>
<td>1/8</td>
<td>4</td>
<td>.2500</td>
<td>.2165</td>
<td>2.3170</td>
</tr>
<tr>
<td>1/8</td>
<td>4</td>
<td>.2500</td>
<td>.2165</td>
<td>2.4420</td>
</tr>
<tr>
<td>3</td>
<td>3(\frac{1}{2})</td>
<td>.2857</td>
<td>.2474</td>
<td>2.5052</td>
</tr>
<tr>
<td>3</td>
<td>3(\frac{1}{2})</td>
<td>.2857</td>
<td>.2474</td>
<td>2.6301</td>
</tr>
<tr>
<td>3</td>
<td>3(\frac{1}{2})</td>
<td>.2857</td>
<td>.2474</td>
<td>2.7551</td>
</tr>
<tr>
<td>3</td>
<td>3(\frac{1}{2})</td>
<td>.3077</td>
<td>.2666</td>
<td>2.8418</td>
</tr>
<tr>
<td>3</td>
<td>3(\frac{1}{2})</td>
<td>.3077</td>
<td>.2666</td>
<td>2.9668</td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>.3077</td>
<td>.2666</td>
<td>3.0918</td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>.3333</td>
<td>.2886</td>
<td>3.1727</td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>.3333</td>
<td>.2886</td>
<td>3.2977</td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>.3333</td>
<td>.2886</td>
<td>3.4227</td>
</tr>
</tbody>
</table>
Table 9.—Whitworth Standard Screw Threads

<table>
<thead>
<tr>
<th>Diam. of Screw</th>
<th>No. of Threads per inch</th>
<th>Pitch</th>
<th>Depth of Thread</th>
<th>Diam. at Root of Thread</th>
</tr>
</thead>
<tbody>
<tr>
<td>1/4</td>
<td>20</td>
<td>.0500</td>
<td>.0320</td>
<td>.1860</td>
</tr>
<tr>
<td>3/32</td>
<td>18</td>
<td>.0556</td>
<td>.0356</td>
<td>.2414</td>
</tr>
<tr>
<td>9/64</td>
<td>16</td>
<td>.0625</td>
<td>.0400</td>
<td>.2950</td>
</tr>
<tr>
<td>3/8</td>
<td>14</td>
<td>.0714</td>
<td>.0457</td>
<td>.3460</td>
</tr>
<tr>
<td>16</td>
<td>12</td>
<td>.0833</td>
<td>.0534</td>
<td>.3933</td>
</tr>
<tr>
<td>1/2</td>
<td>12</td>
<td>.0833</td>
<td>.0534</td>
<td>.4558</td>
</tr>
<tr>
<td>1</td>
<td>11</td>
<td>.0909</td>
<td>.0582</td>
<td>.5086</td>
</tr>
<tr>
<td>5/8</td>
<td>11</td>
<td>.0909</td>
<td>.0582</td>
<td>.5711</td>
</tr>
<tr>
<td>3/4</td>
<td>10</td>
<td>.1000</td>
<td>.0640</td>
<td>.6219</td>
</tr>
<tr>
<td>7/8</td>
<td>10</td>
<td>.1000</td>
<td>.0640</td>
<td>.6844</td>
</tr>
<tr>
<td>1</td>
<td>9</td>
<td>.1111</td>
<td>.0711</td>
<td>.7327</td>
</tr>
<tr>
<td>7/16</td>
<td>8</td>
<td>.1250</td>
<td>.0800</td>
<td>.8399</td>
</tr>
<tr>
<td>1/4</td>
<td>7</td>
<td>.1429</td>
<td>.0915</td>
<td>.9420</td>
</tr>
<tr>
<td>9/32</td>
<td>6</td>
<td>.1429</td>
<td>.0915</td>
<td>1.0670</td>
</tr>
<tr>
<td>5/32</td>
<td>6</td>
<td>.1667</td>
<td>.1067</td>
<td>1.1616</td>
</tr>
<tr>
<td>3/16</td>
<td>5</td>
<td>.1667</td>
<td>.1067</td>
<td>1.2866</td>
</tr>
<tr>
<td>1/8</td>
<td>5</td>
<td>.2000</td>
<td>.1281</td>
<td>1.3689</td>
</tr>
<tr>
<td>7/32</td>
<td>41/2</td>
<td>.2222</td>
<td>.1423</td>
<td>1.4939</td>
</tr>
<tr>
<td>3/16</td>
<td>4</td>
<td>.2500</td>
<td>.1601</td>
<td>1.7154</td>
</tr>
<tr>
<td>1/8</td>
<td>4</td>
<td>.2500</td>
<td>.1601</td>
<td>1.9298</td>
</tr>
<tr>
<td>1/4</td>
<td>31/2</td>
<td>.2857</td>
<td>.1830</td>
<td>2.1798</td>
</tr>
<tr>
<td>3/16</td>
<td>31/2</td>
<td>.2857</td>
<td>.1830</td>
<td>2.3841</td>
</tr>
<tr>
<td>1/8</td>
<td>31/2</td>
<td>.3077</td>
<td>.1970</td>
<td>2.6341</td>
</tr>
<tr>
<td>3/32</td>
<td>31/2</td>
<td>.3077</td>
<td>.1970</td>
<td>2.8560</td>
</tr>
<tr>
<td>1/16</td>
<td>31/2</td>
<td>.3333</td>
<td>.2134</td>
<td>3.1060</td>
</tr>
<tr>
<td>7/64</td>
<td>3</td>
<td>.3333</td>
<td>.2134</td>
<td>3.3331</td>
</tr>
<tr>
<td>3/16</td>
<td>3</td>
<td>.3333</td>
<td>.2134</td>
<td>3.5731</td>
</tr>
<tr>
<td>1/8</td>
<td>21/2</td>
<td>.3478</td>
<td>.2227</td>
<td>4.0546</td>
</tr>
<tr>
<td>5/32</td>
<td>21/2</td>
<td>.3636</td>
<td>.2328</td>
<td>4.5343</td>
</tr>
<tr>
<td>1/16</td>
<td>21/2</td>
<td>.3810</td>
<td>.2439</td>
<td>5.0121</td>
</tr>
<tr>
<td>1/8</td>
<td>21/2</td>
<td>.4000</td>
<td>.2561</td>
<td>5.4877</td>
</tr>
</tbody>
</table>
Table 10.—British Association Screw Threads

<table>
<thead>
<tr>
<th>Number</th>
<th>Diam. of Screw mm.</th>
<th>Approximate Diam. Inches</th>
<th>Pitch mm.</th>
<th>Approximate Pitch Inches</th>
<th>Depth of Thread mm.</th>
<th>Diam. at Root of Thread mm.</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>6.0</td>
<td>.236</td>
<td>1.0</td>
<td>.0394</td>
<td>.6</td>
<td>4.8</td>
</tr>
<tr>
<td>1</td>
<td>5.3</td>
<td>.209</td>
<td>.9</td>
<td>.0354</td>
<td>.54</td>
<td>4.22</td>
</tr>
<tr>
<td>2</td>
<td>4.7</td>
<td>.185</td>
<td>.81</td>
<td>.0319</td>
<td>.485</td>
<td>3.73</td>
</tr>
<tr>
<td>3</td>
<td>4.1</td>
<td>.161</td>
<td>.73</td>
<td>.0287</td>
<td>.44</td>
<td>3.22</td>
</tr>
<tr>
<td>4</td>
<td>3.6</td>
<td>.142</td>
<td>.66</td>
<td>.0260</td>
<td>.395</td>
<td>2.81</td>
</tr>
<tr>
<td>5</td>
<td>3.2</td>
<td>.126</td>
<td>.59</td>
<td>.0232</td>
<td>.335</td>
<td>2.49</td>
</tr>
<tr>
<td>6</td>
<td>2.8</td>
<td>.110</td>
<td>.53</td>
<td>.0209</td>
<td>.32</td>
<td>2.16</td>
</tr>
<tr>
<td>7</td>
<td>2.5</td>
<td>.098</td>
<td>.48</td>
<td>.0189</td>
<td>.29</td>
<td>1.92</td>
</tr>
<tr>
<td>8</td>
<td>2.2</td>
<td>.087</td>
<td>.43</td>
<td>.0169</td>
<td>.26</td>
<td>1.68</td>
</tr>
<tr>
<td>9</td>
<td>1.9</td>
<td>.075</td>
<td>.39</td>
<td>.0154</td>
<td>.235</td>
<td>1.43</td>
</tr>
<tr>
<td>10</td>
<td>1.7</td>
<td>.067</td>
<td>.35</td>
<td>.0138</td>
<td>.21</td>
<td>1.28</td>
</tr>
<tr>
<td>11</td>
<td>1.5</td>
<td>.059</td>
<td>.31</td>
<td>.0122</td>
<td>.185</td>
<td>1.13</td>
</tr>
<tr>
<td>12</td>
<td>1.3</td>
<td>.051</td>
<td>.28</td>
<td>.0110</td>
<td>.17</td>
<td>.96</td>
</tr>
<tr>
<td>13</td>
<td>1.2</td>
<td>.047</td>
<td>.25</td>
<td>.0098</td>
<td>.15</td>
<td>.9</td>
</tr>
<tr>
<td>14</td>
<td>1.0</td>
<td>.039</td>
<td>.23</td>
<td>.0091</td>
<td>.14</td>
<td>.72</td>
</tr>
<tr>
<td>15</td>
<td>.9</td>
<td>.035</td>
<td>.21</td>
<td>.0083</td>
<td>.125</td>
<td>.65</td>
</tr>
<tr>
<td>16</td>
<td>.79</td>
<td>.031</td>
<td>.19</td>
<td>.0075</td>
<td>.115</td>
<td>.56</td>
</tr>
<tr>
<td>17</td>
<td>.70</td>
<td>.028</td>
<td>.17</td>
<td>.0067</td>
<td>.10</td>
<td>.50</td>
</tr>
<tr>
<td>18</td>
<td>.62</td>
<td>.024</td>
<td>.15</td>
<td>.0059</td>
<td>.09</td>
<td>.44</td>
</tr>
<tr>
<td>19</td>
<td>.54</td>
<td>.021</td>
<td>.14</td>
<td>.0055</td>
<td>.085</td>
<td>.37</td>
</tr>
<tr>
<td>20</td>
<td>.48</td>
<td>.019</td>
<td>.12</td>
<td>.0047</td>
<td>.07</td>
<td>.34</td>
</tr>
<tr>
<td>21</td>
<td>.42</td>
<td>.017</td>
<td>.11</td>
<td>.0043</td>
<td>.065</td>
<td>.29</td>
</tr>
<tr>
<td>22</td>
<td>.37</td>
<td>.015</td>
<td>.10</td>
<td>.0039</td>
<td>.06</td>
<td>.25</td>
</tr>
<tr>
<td>23</td>
<td>.33</td>
<td>.013</td>
<td>.09</td>
<td>.0035</td>
<td>.055</td>
<td>.22</td>
</tr>
<tr>
<td>24</td>
<td>.29</td>
<td>.011</td>
<td>.08</td>
<td>.0031</td>
<td>.05</td>
<td>.19</td>
</tr>
<tr>
<td>25</td>
<td>.25</td>
<td>.010</td>
<td>.07</td>
<td>.0028</td>
<td>.04</td>
<td>.17</td>
</tr>
</tbody>
</table>
### Table 11. French (Metric) Standard Screw Threads

![Diagram of screw thread](image)

<table>
<thead>
<tr>
<th>Diameter of Screw mm.</th>
<th>Pitch mm.</th>
<th>Diameter at Root of Thread mm.</th>
<th>Width of Flat mm.</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>0.5</td>
<td>2.35</td>
<td>.06</td>
</tr>
<tr>
<td>4</td>
<td>0.75</td>
<td>3.03</td>
<td>.09</td>
</tr>
<tr>
<td>5</td>
<td>0.75</td>
<td>4.03</td>
<td>.13</td>
</tr>
<tr>
<td>6</td>
<td>1.0</td>
<td>4.70</td>
<td>.13</td>
</tr>
<tr>
<td>7</td>
<td>1.0</td>
<td>5.70</td>
<td>.16</td>
</tr>
<tr>
<td>8</td>
<td>1.25</td>
<td>6.38</td>
<td>.16</td>
</tr>
<tr>
<td>9</td>
<td>1.25</td>
<td>7.38</td>
<td>.16</td>
</tr>
<tr>
<td>10</td>
<td>1.5</td>
<td>8.05</td>
<td>.19</td>
</tr>
<tr>
<td>11</td>
<td>1.5</td>
<td>9.05</td>
<td>.19</td>
</tr>
<tr>
<td>12</td>
<td>1.75</td>
<td>10.05</td>
<td>.22</td>
</tr>
<tr>
<td>14</td>
<td>2.0</td>
<td>11.40</td>
<td>.25</td>
</tr>
<tr>
<td>16</td>
<td>2.0</td>
<td>13.40</td>
<td>.31</td>
</tr>
<tr>
<td>18</td>
<td>2.5</td>
<td>14.75</td>
<td>.31</td>
</tr>
<tr>
<td>20</td>
<td>2.5</td>
<td>16.75</td>
<td>.31</td>
</tr>
<tr>
<td>22</td>
<td>3.0</td>
<td>18.10</td>
<td>.38</td>
</tr>
<tr>
<td>24</td>
<td>3.0</td>
<td>20.10</td>
<td>.38</td>
</tr>
<tr>
<td>26</td>
<td>3.0</td>
<td>22.10</td>
<td>.38</td>
</tr>
<tr>
<td>27</td>
<td>3.0</td>
<td>23.10</td>
<td>.38</td>
</tr>
<tr>
<td>28</td>
<td>3.0</td>
<td>24.10</td>
<td>.38</td>
</tr>
<tr>
<td>30</td>
<td>3.5</td>
<td>25.45</td>
<td>.44</td>
</tr>
<tr>
<td>32</td>
<td>3.5</td>
<td>27.45</td>
<td>.44</td>
</tr>
<tr>
<td>33</td>
<td>3.5</td>
<td>28.45</td>
<td>.44</td>
</tr>
<tr>
<td>34</td>
<td>3.5</td>
<td>29.45</td>
<td>.44</td>
</tr>
<tr>
<td>36</td>
<td>4.0</td>
<td>30.80</td>
<td>.5</td>
</tr>
<tr>
<td>38</td>
<td>4.0</td>
<td>32.80</td>
<td>.5</td>
</tr>
<tr>
<td>39</td>
<td>4.0</td>
<td>33.80</td>
<td>.5</td>
</tr>
<tr>
<td>40</td>
<td>4.0</td>
<td>34.80</td>
<td>.5</td>
</tr>
<tr>
<td>42</td>
<td>4.5</td>
<td>36.15</td>
<td>.5</td>
</tr>
<tr>
<td>44</td>
<td>4.5</td>
<td>38.15</td>
<td>.5</td>
</tr>
<tr>
<td>45</td>
<td>4.5</td>
<td>39.15</td>
<td>.5</td>
</tr>
<tr>
<td>46</td>
<td>4.5</td>
<td>40.15</td>
<td>.5</td>
</tr>
<tr>
<td>48</td>
<td>5.0</td>
<td>41.51</td>
<td>.63</td>
</tr>
<tr>
<td>50</td>
<td>5.0</td>
<td>43.51</td>
<td>.63</td>
</tr>
<tr>
<td>52</td>
<td>5.0</td>
<td>45.51</td>
<td>.63</td>
</tr>
<tr>
<td>56</td>
<td>5.5</td>
<td>48.86</td>
<td>.69</td>
</tr>
<tr>
<td>60</td>
<td>5.5</td>
<td>52.86</td>
<td>.69</td>
</tr>
<tr>
<td>64</td>
<td>6.0</td>
<td>56.21</td>
<td>.75</td>
</tr>
<tr>
<td>68</td>
<td>6.0</td>
<td>60.21</td>
<td>.81</td>
</tr>
<tr>
<td>72</td>
<td>6.5</td>
<td>63.56</td>
<td>.81</td>
</tr>
<tr>
<td>76</td>
<td>6.5</td>
<td>67.56</td>
<td>.81</td>
</tr>
<tr>
<td>80</td>
<td>7.0</td>
<td>70.91</td>
<td>.88</td>
</tr>
</tbody>
</table>
### Table 12.—International Standard Screw Threads

### Dimensions in Millimeters

<table>
<thead>
<tr>
<th>Diam. of Screw</th>
<th>Pitch</th>
<th>Diam. of Screw</th>
<th>Pitch</th>
<th>Diam. of Screw</th>
<th>Pitch</th>
<th>Diam. of Screw</th>
<th>Pitch</th>
</tr>
</thead>
<tbody>
<tr>
<td>6</td>
<td>1.00</td>
<td>18</td>
<td>2.50</td>
<td>39</td>
<td>4.00</td>
<td>68</td>
<td>6.00</td>
</tr>
<tr>
<td>7</td>
<td>1.00</td>
<td>20</td>
<td>2.50</td>
<td>42</td>
<td>4.50</td>
<td>72</td>
<td>6.50</td>
</tr>
<tr>
<td>8</td>
<td>1.25</td>
<td>22</td>
<td>2.50</td>
<td>45</td>
<td>4.50</td>
<td>76</td>
<td>6.50</td>
</tr>
<tr>
<td>9</td>
<td>1.25</td>
<td>24</td>
<td>3.00</td>
<td>48</td>
<td>5.00</td>
<td>80</td>
<td>7.00</td>
</tr>
<tr>
<td>10</td>
<td>1.50</td>
<td>27</td>
<td>3.00</td>
<td>52</td>
<td>5.00</td>
<td>88</td>
<td>7.50</td>
</tr>
<tr>
<td>11</td>
<td>1.50</td>
<td>30</td>
<td>3.50</td>
<td>56</td>
<td>5.50</td>
<td>96</td>
<td>8.00</td>
</tr>
<tr>
<td>12</td>
<td>1.75</td>
<td>33</td>
<td>3.50</td>
<td>60</td>
<td>5.50</td>
<td>116</td>
<td>9.00</td>
</tr>
<tr>
<td>14</td>
<td>2.00</td>
<td>36</td>
<td>4.00</td>
<td>64</td>
<td>6.00</td>
<td>136</td>
<td>10.00</td>
</tr>
</tbody>
</table>

The "International Standard" is the same, with modifications noted, as that now in general use in France.

### International Standard Threads

At the "Congress International pour L'Unification des Filetages," held in Zurich, October 24, 1898, the following resolutions were adopted:

The Congress has undertaken the task of unifying the threads of machine screws. It recommends to all those who wish to adopt the metric system of threads to make use of the proposed system. This system is the one which has been established by the "Society for the Encouragement of National Industries," with the following modification adopted by this Congress.

1. The clearance at the bottom of thread shall not exceed \( \frac{1}{8} \) part of the height of the original triangle. The shape of the bottom of the thread resulting from said clearance is left to the judgment of the manufacturers. However, the Congress recommends rounded profile for said bottom.

2. The table for Standard Diameters accepted is the one which has been proposed by the Swiss Committee of Action. (This table is given above.) It is to be noticed especially that 1.25 mm. pitch is adopted for 8 mm. diameter, and 1.75 mm. pitch for 12 mm. diameter. The pitches of sizes between standard diameters indicated in the table are to be the same as for the next smaller standard diameter.
The Acme standard thread is an adaptation of the most commonly used style of Worm Thread and is intended to take the place of the square thread.
It is a little shallower than the worm thread, but the same depth as the square thread and much stronger than the latter.
The various parts of the Acme standard thread are obtained as follows:

Width of Point of Tool for Screw Thread = \( \frac{0.3707}{\text{No. of Threads per inch}} - 0.0052 \).

Width of Screw or Nut Thread = \( \frac{0.3707}{\text{No. of Threads per inch}} \).

Diameter of Screw at Root = \( \frac{1}{\left( \frac{\text{No. of Threads per inch}}{100} + 0.020 \right)} \).

Depth of Thread = \( \frac{1}{2 \times \text{No. of Threads per inch}} + 0.010 \).

### Table of Acme 29° Screw Thread Parts

<table>
<thead>
<tr>
<th>( N )</th>
<th>( P )</th>
<th>( D )</th>
<th>( F )</th>
<th>( W )</th>
<th>( S )</th>
<th>( B )</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \text{Number of Threads per Inch} )</td>
<td>( \text{Pitch of Single Thread} )</td>
<td>( \text{Depth of Thread} )</td>
<td>( \text{Width of Top of Thread} )</td>
<td>( \text{Width of Space at Bottom of Thread} )</td>
<td>( \text{Width of Space at Top of Thread} )</td>
<td>( \text{Thickness at Root of Thread} )</td>
</tr>
<tr>
<td>1</td>
<td>1.0</td>
<td>.5100</td>
<td>.3707</td>
<td>.3655</td>
<td>.6203</td>
<td>.6345</td>
</tr>
<tr>
<td>1\frac{1}{4}</td>
<td>.750</td>
<td>.3850</td>
<td>.2780</td>
<td>.2728</td>
<td>.4720</td>
<td>.4772</td>
</tr>
<tr>
<td>2</td>
<td>.500</td>
<td>.2600</td>
<td>.1853</td>
<td>.1801</td>
<td>.3147</td>
<td>.3199</td>
</tr>
<tr>
<td>3</td>
<td>.3333</td>
<td>.1767</td>
<td>.1235</td>
<td>.1183</td>
<td>.2098</td>
<td>.2150</td>
</tr>
<tr>
<td>4</td>
<td>.250</td>
<td>.1350</td>
<td>.0927</td>
<td>.0875</td>
<td>.1573</td>
<td>.1625</td>
</tr>
<tr>
<td>5</td>
<td>.200</td>
<td>.1100</td>
<td>.0741</td>
<td>.0689</td>
<td>.1259</td>
<td>.1311</td>
</tr>
<tr>
<td>6</td>
<td>.1666</td>
<td>.0933</td>
<td>.0618</td>
<td>.0566</td>
<td>.1049</td>
<td>.1101</td>
</tr>
<tr>
<td>7</td>
<td>.1428</td>
<td>.0814</td>
<td>.0529</td>
<td>.0478</td>
<td>.0899</td>
<td>.0951</td>
</tr>
<tr>
<td>8</td>
<td>.125</td>
<td>.0725</td>
<td>.0463</td>
<td>.0411</td>
<td>.0787</td>
<td>.0839</td>
</tr>
<tr>
<td>9</td>
<td>.1111</td>
<td>.0655</td>
<td>.0413</td>
<td>.0361</td>
<td>.0699</td>
<td>.0751</td>
</tr>
<tr>
<td>10</td>
<td>.10</td>
<td>.0600</td>
<td>.0371</td>
<td>.0319</td>
<td>.0629</td>
<td>.0681</td>
</tr>
</tbody>
</table>
### Table 14.—Acme 29° Tap Threads

The Acme standard tap-thread is cut with the same width of tool as the screw-thread and the diameter at the root is the same for tap and screw. Clearance at bottom of thread between screw and nut is obtained by boring the nut blank .020 oversize.

The outside diameter of the tap is made .020 larger than the screw to give clearance between top of screw-thread and bottom of nut.

**Width of Point of Tool for Tap-Thread =**

\[
\frac{3707}{\text{No. of Threads per Inch}} - .0052
\]

**Width of Thread =**

\[
\frac{3707}{\text{No. of Threads per Inch}} - .0052
\]

**Diameter of Tap = Diameter of Screw + .020.**

**Diameter of Tap at Root =**

\[
\text{Diameter of Tap} = \left(\frac{1}{\text{No. of Threads per Inch}} + .040\right)
\]

**Depth of Thread =**

\[
\frac{1}{2 \times \text{No. of Threads per Inch}} + .020.
\]

### Table of Acme Standard 29° Tap-Thread Parts

<table>
<thead>
<tr>
<th>N.</th>
<th>P</th>
<th>D</th>
<th>F</th>
<th>W</th>
<th>S</th>
<th>B</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Number of Threads per Inch</td>
<td>Pitch of Single Thread</td>
<td>Depth of Thread</td>
<td>Width of Top of Thread</td>
<td>Width of Space at Bottom of Thread</td>
<td>Width of Space at Top of Thread</td>
</tr>
<tr>
<td>----</td>
<td>-----------------------------</td>
<td>-----------------------</td>
<td>----------------</td>
<td>-----------------------</td>
<td>--------------------------------</td>
<td>--------------------------------</td>
</tr>
<tr>
<td>1</td>
<td>.10</td>
<td>.5200</td>
<td>.3655</td>
<td>.3655</td>
<td>.6345</td>
<td>.6345</td>
</tr>
<tr>
<td>1½</td>
<td>.750</td>
<td>.3950</td>
<td>.2728</td>
<td>.2728</td>
<td>.4772</td>
<td>.4772</td>
</tr>
<tr>
<td>2</td>
<td>.500</td>
<td>.2700</td>
<td>.1801</td>
<td>.1801</td>
<td>.3199</td>
<td>.3199</td>
</tr>
<tr>
<td>3</td>
<td>.3333</td>
<td>.1867</td>
<td>.1183</td>
<td>.1183</td>
<td>.2150</td>
<td>.2150</td>
</tr>
<tr>
<td>4</td>
<td>.250</td>
<td>.1450</td>
<td>.0875</td>
<td>.0875</td>
<td>.1625</td>
<td>.1625</td>
</tr>
<tr>
<td>5</td>
<td>.200</td>
<td>.1200</td>
<td>.0689</td>
<td>.0689</td>
<td>.1311</td>
<td>.1311</td>
</tr>
<tr>
<td>6</td>
<td>.1666</td>
<td>.1033</td>
<td>.0566</td>
<td>.0566</td>
<td>.1101</td>
<td>.1101</td>
</tr>
<tr>
<td>7</td>
<td>.1428</td>
<td>.0914</td>
<td>.0478</td>
<td>.0478</td>
<td>.0951</td>
<td>.0951</td>
</tr>
<tr>
<td>8</td>
<td>.125</td>
<td>.0825</td>
<td>.0411</td>
<td>.0411</td>
<td>.0839</td>
<td>.0839</td>
</tr>
<tr>
<td>9</td>
<td>.1111</td>
<td>.0755</td>
<td>.0361</td>
<td>.0361</td>
<td>.0751</td>
<td>.0751</td>
</tr>
<tr>
<td>10</td>
<td>.10</td>
<td>.0700</td>
<td>.0319</td>
<td>.0319</td>
<td>.0681</td>
<td>.0681</td>
</tr>
</tbody>
</table>
### Table 15.—Brown & Sharpe 29° Worm Thread

![Diagram of worm thread](image)

- **Pitch** = \( \frac{1}{\text{No. of Threads per inch}} \)
- **Depth of Thread** = \( \frac{.6866}{\text{No. of Threads per inch}} \)
- **Width of Top of Thread** = \( \frac{.335}{\text{No. of Threads per inch}} \)
- **Width of Space at Bottom** = \( \frac{.310}{\text{No. of Threads per inch}} \)
- **Clearance at Bottom of Thread** = \( \frac{10}{\text{Thickness at Pitch Line}} \)
- **Width of Space at Top of Thread** = \( \frac{.665}{\text{No. of Threads per inch}} \)
- **Thickness at Root of Thread** = \( \frac{.69}{\text{No. of Threads per inch}} \)

<table>
<thead>
<tr>
<th>Number of Threads Per Inch</th>
<th>P</th>
<th>D</th>
<th>F</th>
<th>W</th>
<th>T</th>
<th>A</th>
<th>C</th>
<th>S</th>
<th>B</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Pitch of Single Thread</td>
<td>Depth of Thread</td>
<td>Width of Top of Thread</td>
<td>Width at Bottom</td>
<td>Thickness at Pitch Line</td>
<td>Thread Above Pitch Line</td>
<td>Clearance at Bottom of Thread</td>
<td>Width of Space at Top</td>
<td>Thickness at Root of Thread</td>
</tr>
<tr>
<td>1</td>
<td>1.0</td>
<td>.6866</td>
<td>.3350</td>
<td>.3100</td>
<td>.5000</td>
<td>.3183</td>
<td>.05</td>
<td>.665</td>
<td>.69</td>
</tr>
<tr>
<td>1/2</td>
<td>.8</td>
<td>.5492</td>
<td>.2680</td>
<td>.2480</td>
<td>.4000</td>
<td>.2546</td>
<td>.04</td>
<td>.532</td>
<td>.552</td>
</tr>
<tr>
<td>1/4</td>
<td>.6666</td>
<td>.4577</td>
<td>.2233</td>
<td>.2066</td>
<td>.3333</td>
<td>.2122</td>
<td>.0333</td>
<td>.4433</td>
<td>.4599</td>
</tr>
<tr>
<td>2</td>
<td>.5</td>
<td>.3433</td>
<td>.1675</td>
<td>.1550</td>
<td>.2500</td>
<td>.1592</td>
<td>.0250</td>
<td>.3325</td>
<td>.345</td>
</tr>
<tr>
<td>2/4</td>
<td>.4</td>
<td>.2746</td>
<td>.1340</td>
<td>.1240</td>
<td>.2000</td>
<td>.1273</td>
<td>.0200</td>
<td>.2660</td>
<td>.276</td>
</tr>
<tr>
<td>3</td>
<td>.3333</td>
<td>.2289</td>
<td>.1117</td>
<td>.1033</td>
<td>.1666</td>
<td>.1061</td>
<td>.0166</td>
<td>.2216</td>
<td>.2399</td>
</tr>
<tr>
<td>3/4</td>
<td>.2857</td>
<td>.1962</td>
<td>.0957</td>
<td>.0886</td>
<td>.1429</td>
<td>.0909</td>
<td>.0143</td>
<td>.1901</td>
<td>.201</td>
</tr>
<tr>
<td>4</td>
<td>.250</td>
<td>.1716</td>
<td>.0838</td>
<td>.0775</td>
<td>.1250</td>
<td>.0796</td>
<td>.0125</td>
<td>.1637</td>
<td>.1725</td>
</tr>
<tr>
<td>4 1/2</td>
<td>.2222</td>
<td>.1526</td>
<td>.0744</td>
<td>.0689</td>
<td>.1111</td>
<td>.0707</td>
<td>.0111</td>
<td>.1478</td>
<td>.1533</td>
</tr>
<tr>
<td>5</td>
<td>.2</td>
<td>.1373</td>
<td>.0670</td>
<td>.0620</td>
<td>.1000</td>
<td>.0637</td>
<td>.0100</td>
<td>.1330</td>
<td>.138</td>
</tr>
<tr>
<td>6</td>
<td>.1666</td>
<td>.1144</td>
<td>.0558</td>
<td>.0517</td>
<td>.0833</td>
<td>.0531</td>
<td>.0083</td>
<td>.1108</td>
<td>.115</td>
</tr>
<tr>
<td>7</td>
<td>.1428</td>
<td>.0981</td>
<td>.0479</td>
<td>.0443</td>
<td>.0714</td>
<td>.0455</td>
<td>.0071</td>
<td>.0955</td>
<td>.0985</td>
</tr>
<tr>
<td>8</td>
<td>.125</td>
<td>.0858</td>
<td>.0419</td>
<td>.0388</td>
<td>.0625</td>
<td>.0398</td>
<td>.0062</td>
<td>.0818</td>
<td>.0862</td>
</tr>
<tr>
<td>9</td>
<td>.1111</td>
<td>.0763</td>
<td>.0372</td>
<td>.0344</td>
<td>.0555</td>
<td>.0354</td>
<td>.0055</td>
<td>.0739</td>
<td>.0766</td>
</tr>
<tr>
<td>10</td>
<td>.10</td>
<td>.0687</td>
<td>.0335</td>
<td>.0310</td>
<td>.0500</td>
<td>.0318</td>
<td>.005</td>
<td>.0665</td>
<td>.069</td>
</tr>
<tr>
<td>12</td>
<td>.0833</td>
<td>.0572</td>
<td>.0279</td>
<td>.0258</td>
<td>.0416</td>
<td>.0265</td>
<td>.0042</td>
<td>.0551</td>
<td>.0575</td>
</tr>
<tr>
<td>16</td>
<td>.0625</td>
<td>.0429</td>
<td>.0209</td>
<td>.0194</td>
<td>.0312</td>
<td>.0199</td>
<td>.0031</td>
<td>.0409</td>
<td>.0431</td>
</tr>
<tr>
<td>20</td>
<td>.050</td>
<td>.0343</td>
<td>.0167</td>
<td>.0155</td>
<td>.0250</td>
<td>.0159</td>
<td>.0025</td>
<td>.0332</td>
<td>.0345</td>
</tr>
</tbody>
</table>
### TABLE 16.—MACHINE SCREW SIZES

Sizes marked * are A.S.M.E. Standard

Tap Drill Sizes calculated from the formula \( O.D. - \frac{0.9}{N} \) which gives a trifle over \( \frac{3}{2} \) of a full thread. See page 178.

<table>
<thead>
<tr>
<th>Outside diam. (O.D.)</th>
<th>Nearest number, letter or fractional size body drill</th>
<th>Screw gauge No.</th>
<th>No. of threads per in. ((N))</th>
<th>Root diam. exact ((R. D.))</th>
<th>Tap drill diam. (O.D. \frac{0.9}{N})</th>
<th>Nearest number, letter or fractional size tap drill</th>
<th>Pitch diam. or screw thread micrometer reading</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.060</td>
<td>#52-.0635</td>
<td>0</td>
<td>80*</td>
<td>0.0438</td>
<td>0.049</td>
<td>#56-.0465</td>
<td>0.0519</td>
</tr>
<tr>
<td>.073</td>
<td>#49-.073</td>
<td>1</td>
<td>72*</td>
<td>0.055</td>
<td>0.060</td>
<td>#53-.0595</td>
<td>0.064</td>
</tr>
<tr>
<td>.0842</td>
<td>#44-.086</td>
<td>2</td>
<td>48</td>
<td>0.0565</td>
<td>0.065</td>
<td>#52-.0635</td>
<td>0.0704</td>
</tr>
<tr>
<td>.0842</td>
<td>&quot; &quot;</td>
<td>2</td>
<td>56</td>
<td>0.0612</td>
<td>0.068</td>
<td>#51-.067</td>
<td>0.0727</td>
</tr>
<tr>
<td>.086</td>
<td>&quot; &quot;</td>
<td>2</td>
<td>64*</td>
<td>0.0657</td>
<td>0.072</td>
<td>#50-.070</td>
<td>0.0759</td>
</tr>
<tr>
<td>.0973</td>
<td>#39-.0995</td>
<td>3</td>
<td>40</td>
<td>0.0649</td>
<td>0.074</td>
<td>#49-.073</td>
<td>0.0811</td>
</tr>
<tr>
<td>.0973</td>
<td>&quot; &quot;</td>
<td>3</td>
<td>48</td>
<td>0.0696</td>
<td>0.078</td>
<td>#48-.076</td>
<td>0.0835</td>
</tr>
<tr>
<td>.099</td>
<td>&quot; &quot;</td>
<td>3</td>
<td>56*</td>
<td>0.0758</td>
<td>0.083</td>
<td>#45-.082</td>
<td>0.0874</td>
</tr>
<tr>
<td>.1105</td>
<td>#33-.113</td>
<td>4</td>
<td>32</td>
<td>0.0699</td>
<td>0.082</td>
<td>#45-.082</td>
<td>0.0902</td>
</tr>
<tr>
<td>.1105</td>
<td>&quot; &quot;</td>
<td>4</td>
<td>36</td>
<td>0.0744</td>
<td>0.085</td>
<td>#45-.082</td>
<td>0.0924</td>
</tr>
<tr>
<td>.1105</td>
<td>&quot; &quot;</td>
<td>4</td>
<td>40</td>
<td>0.0781</td>
<td>0.088</td>
<td>#44-.086</td>
<td>0.0943</td>
</tr>
<tr>
<td>.112</td>
<td>&quot; &quot;</td>
<td>4</td>
<td>48*</td>
<td>0.0849</td>
<td>0.093</td>
<td>#42-.0935</td>
<td>0.0985</td>
</tr>
<tr>
<td>.1236</td>
<td>(\frac{3}{8})&quot; or #30-.1285</td>
<td>5</td>
<td>30</td>
<td>0.0803</td>
<td>0.093</td>
<td>#42-.0935</td>
<td>0.1019</td>
</tr>
<tr>
<td>.1236</td>
<td>&quot; &quot;</td>
<td>5</td>
<td>32</td>
<td>0.083</td>
<td>0.095</td>
<td>#42-.0935</td>
<td>0.1033</td>
</tr>
<tr>
<td>.1236</td>
<td>&quot; &quot;</td>
<td>5</td>
<td>36</td>
<td>0.0875</td>
<td>0.098</td>
<td>#40-.098</td>
<td>0.1055</td>
</tr>
<tr>
<td>.1236</td>
<td>&quot; &quot;</td>
<td>5</td>
<td>40</td>
<td>0.0912</td>
<td>0.101</td>
<td>#39-.0995</td>
<td>0.1074</td>
</tr>
<tr>
<td>.125</td>
<td>&quot; &quot;</td>
<td>5</td>
<td>44*</td>
<td>0.0955</td>
<td>0.104</td>
<td>#37-.104</td>
<td>0.1102</td>
</tr>
<tr>
<td>.1368</td>
<td>#28-.1405</td>
<td>6</td>
<td>30</td>
<td>0.0935</td>
<td>0.106</td>
<td>#36-.1065</td>
<td>0.1151</td>
</tr>
<tr>
<td>.1368</td>
<td>&quot; &quot;</td>
<td>6</td>
<td>32</td>
<td>0.0962</td>
<td>0.108</td>
<td>#36-.1065</td>
<td>0.1165</td>
</tr>
<tr>
<td>.1368</td>
<td>&quot; &quot;</td>
<td>6</td>
<td>36</td>
<td>1.007</td>
<td>1.111</td>
<td>#34-.111</td>
<td>1.187</td>
</tr>
<tr>
<td>.138</td>
<td>&quot; &quot;</td>
<td>6</td>
<td>40*</td>
<td>1.055</td>
<td>1.115</td>
<td>#33-.113</td>
<td>1.218</td>
</tr>
<tr>
<td>.150</td>
<td>#24-.152</td>
<td>7</td>
<td>28</td>
<td>1.036</td>
<td>1.118</td>
<td>#32-.116</td>
<td>1.268</td>
</tr>
<tr>
<td>.150</td>
<td>&quot; &quot;</td>
<td>7</td>
<td>30</td>
<td>1.067</td>
<td>1.120</td>
<td>#31-.120</td>
<td>1.283</td>
</tr>
<tr>
<td>.150</td>
<td>&quot; &quot;</td>
<td>7</td>
<td>32</td>
<td>1.094</td>
<td>1.122</td>
<td>#31-.120</td>
<td>1.297</td>
</tr>
<tr>
<td>.151</td>
<td>&quot; &quot;</td>
<td>7</td>
<td>36*</td>
<td>1.149</td>
<td>1.125</td>
<td>(\frac{3}{8})&quot; or #31</td>
<td>1.133</td>
</tr>
<tr>
<td>Outside diam. (O. D.)</td>
<td>Nearest number, letter or fractional size body drill</td>
<td>Screw gauge No.</td>
<td>No. of threads per in. (N)</td>
<td>Root diam. exact (R. D.)</td>
<td>Tap drill diam. ((\text{O. D.} - \frac{9}{N}))</td>
<td>Nearest number, letter or fractional size tap drill</td>
<td>Pitch diam. or screw thread micrometer reading</td>
</tr>
<tr>
<td>----------------------</td>
<td>---------------------------------------------------</td>
<td>-----------------</td>
<td>-----------------------------</td>
<td>---------------------------</td>
<td>------------------------------------------</td>
<td>-----------------------------------------------</td>
<td>-----------------------------------------------</td>
</tr>
<tr>
<td>.1631</td>
<td>#19-.166</td>
<td>8</td>
<td>24</td>
<td>.109</td>
<td>.125</td>
<td>(\frac{3}{8})&quot; or #31</td>
<td>.136</td>
</tr>
<tr>
<td>.1631</td>
<td>&quot;        &quot;</td>
<td>8</td>
<td>30</td>
<td>.1198</td>
<td>.133</td>
<td>#30-.1285</td>
<td>.1414</td>
</tr>
<tr>
<td>.1631</td>
<td>&quot;        &quot;</td>
<td>8</td>
<td>32</td>
<td>.1225</td>
<td>.135</td>
<td>#29-.136</td>
<td>.1428</td>
</tr>
<tr>
<td>.164</td>
<td>&quot;        &quot;</td>
<td>8</td>
<td>36*</td>
<td>.1279</td>
<td>.139</td>
<td>#29-.136</td>
<td>.146</td>
</tr>
<tr>
<td>.1763</td>
<td>#16-.177</td>
<td>9</td>
<td>24</td>
<td>.1222</td>
<td>.138</td>
<td>#29-.136</td>
<td>.1492</td>
</tr>
<tr>
<td>.1763</td>
<td>&quot;        &quot;</td>
<td>9</td>
<td>28</td>
<td>.1299</td>
<td>.144</td>
<td>#27-.144</td>
<td>.1531</td>
</tr>
<tr>
<td>.1763</td>
<td>&quot;        &quot;</td>
<td>9</td>
<td>30</td>
<td>.1330</td>
<td>.146</td>
<td>#27-.144</td>
<td>.1546</td>
</tr>
<tr>
<td>.177</td>
<td>&quot;        &quot;</td>
<td>9</td>
<td>32*</td>
<td>.1364</td>
<td>.148</td>
<td>#26-.147</td>
<td>.1567</td>
</tr>
<tr>
<td>.1894</td>
<td>#11-.191</td>
<td>10</td>
<td>24</td>
<td>.1353</td>
<td>.151</td>
<td>#25-.1495</td>
<td>.1623</td>
</tr>
<tr>
<td>.190</td>
<td>&quot;        &quot;</td>
<td>10</td>
<td>30*</td>
<td>.1467</td>
<td>.160</td>
<td>#21-.159</td>
<td>.1684</td>
</tr>
<tr>
<td>.1894</td>
<td>&quot;        &quot;</td>
<td>10</td>
<td>32</td>
<td>.1488</td>
<td>.161</td>
<td>#20-.161</td>
<td>.1691</td>
</tr>
<tr>
<td>.2026</td>
<td>#6-.204</td>
<td>11</td>
<td>24</td>
<td>.1485</td>
<td>.165</td>
<td>#19-.166</td>
<td>.1755</td>
</tr>
<tr>
<td>.2026</td>
<td>&quot;        &quot;</td>
<td>11</td>
<td>28</td>
<td>.156</td>
<td>.170</td>
<td>#18-.1695</td>
<td>.1793</td>
</tr>
<tr>
<td>.2026</td>
<td>&quot;        &quot;</td>
<td>11</td>
<td>30</td>
<td>.159</td>
<td>.172</td>
<td>#18-.1695</td>
<td>.1808</td>
</tr>
<tr>
<td>.2158</td>
<td>(\frac{3}{32})-.218</td>
<td>12</td>
<td>20</td>
<td>.1508</td>
<td>.170</td>
<td>#18-.1695</td>
<td>.1833</td>
</tr>
<tr>
<td>.2158</td>
<td>&quot;        &quot;</td>
<td>12</td>
<td>22</td>
<td>.1568</td>
<td>.174</td>
<td>#17-.174</td>
<td>.1863</td>
</tr>
<tr>
<td>.2158</td>
<td>&quot;        &quot;</td>
<td>12</td>
<td>24</td>
<td>.1617</td>
<td>.178</td>
<td>#16-.177</td>
<td>.1887</td>
</tr>
<tr>
<td>.216</td>
<td>&quot;        &quot;</td>
<td>12</td>
<td>28*</td>
<td>.1696</td>
<td>.183</td>
<td>#14-.182</td>
<td>.1928</td>
</tr>
<tr>
<td>.2289</td>
<td>1(\frac{5}{64})-.234</td>
<td>13</td>
<td>20</td>
<td>.1639</td>
<td>.183</td>
<td>#14-.182</td>
<td>.1964</td>
</tr>
<tr>
<td>.2289</td>
<td>&quot;        &quot;</td>
<td>13</td>
<td>22</td>
<td>.1699</td>
<td>.188</td>
<td>#13-.185</td>
<td>.1994</td>
</tr>
<tr>
<td>.2289</td>
<td>&quot;        &quot;</td>
<td>13</td>
<td>24</td>
<td>.1748</td>
<td>.191</td>
<td>#11-.191</td>
<td>.2018</td>
</tr>
<tr>
<td>.2421</td>
<td>D-.246</td>
<td>14</td>
<td>18</td>
<td>.170</td>
<td>.192</td>
<td>#11-.192</td>
<td>.206</td>
</tr>
<tr>
<td>.2421</td>
<td>&quot;        &quot;</td>
<td>14</td>
<td>20</td>
<td>.1771</td>
<td>.197</td>
<td>#9-.196</td>
<td>.2096</td>
</tr>
<tr>
<td>.2421</td>
<td>&quot;        &quot;</td>
<td>14</td>
<td>22</td>
<td>.1831</td>
<td>.201</td>
<td>#7-.201</td>
<td>.2126</td>
</tr>
<tr>
<td>.242</td>
<td>&quot;        &quot;</td>
<td>14</td>
<td>24*</td>
<td>.1879</td>
<td>.204</td>
<td>#6-.204</td>
<td>.2149</td>
</tr>
<tr>
<td>.2552</td>
<td>F-.257</td>
<td>15</td>
<td>18</td>
<td>.1831</td>
<td>.205</td>
<td>#6-.204</td>
<td>.2191</td>
</tr>
<tr>
<td>.2552</td>
<td>&quot;        &quot;</td>
<td>15</td>
<td>20</td>
<td>.1902</td>
<td>.210</td>
<td>#4-.209</td>
<td>.2227</td>
</tr>
<tr>
<td>.2552</td>
<td>&quot;        &quot;</td>
<td>15</td>
<td>22</td>
<td>.1962</td>
<td>.214</td>
<td>#3-.213</td>
<td>.2257</td>
</tr>
<tr>
<td>.2552</td>
<td>&quot;        &quot;</td>
<td>15</td>
<td>24</td>
<td>.2013</td>
<td>.217</td>
<td>#3-.213</td>
<td>.2282</td>
</tr>
<tr>
<td>Outside diam. (O. D.)</td>
<td>Nearest number, letter or fractional size body drill</td>
<td>Screw gauge No.</td>
<td>No. of threads per in. (N)</td>
<td>Root diam. exact (R. D.)</td>
<td>Tap drill diam. ( \text{O.D.} - \frac{0.9}{N} )</td>
<td>Nearest number, letter or fractional size tap drill</td>
<td>Pitch diam. or screw thread micrometer reading</td>
</tr>
<tr>
<td>----------------------</td>
<td>---------------------------------------------------</td>
<td>----------------</td>
<td>---------------------------</td>
<td>--------------------------</td>
<td>---------------------------------</td>
<td>---------------------------------</td>
<td>---------------------------------</td>
</tr>
<tr>
<td>.2684</td>
<td>I-.272</td>
<td>16</td>
<td>16</td>
<td>.1873</td>
<td>.212</td>
<td>#3-.213</td>
<td>.2278</td>
</tr>
<tr>
<td>.2684</td>
<td>&quot; &quot;</td>
<td>16</td>
<td>18</td>
<td>.1963</td>
<td>.218</td>
<td>#3-.213</td>
<td>.2323</td>
</tr>
<tr>
<td>.2684</td>
<td>&quot; &quot;</td>
<td>16</td>
<td>20</td>
<td>.2034</td>
<td>.223</td>
<td>#2-.221</td>
<td>.2359</td>
</tr>
<tr>
<td>.268</td>
<td>&quot; &quot;</td>
<td>16</td>
<td>22*</td>
<td>.209</td>
<td>.227</td>
<td>#1-.228</td>
<td>.2385</td>
</tr>
<tr>
<td>.2816</td>
<td>L-.290</td>
<td>17</td>
<td>16</td>
<td>.2005</td>
<td>.225</td>
<td>#2-.221</td>
<td>.241</td>
</tr>
<tr>
<td>.2816</td>
<td>&quot; &quot;</td>
<td>17</td>
<td>18</td>
<td>.2095</td>
<td>.231</td>
<td>#1-.228</td>
<td>.2455</td>
</tr>
<tr>
<td>.2816</td>
<td>&quot; &quot;</td>
<td>17</td>
<td>20</td>
<td>.2166</td>
<td>.236</td>
<td>1(\frac{5}{6})-.234</td>
<td>.2491</td>
</tr>
<tr>
<td>.2947</td>
<td>1(\frac{5}{6})-.296</td>
<td>18</td>
<td>16</td>
<td>.2136</td>
<td>.238</td>
<td>1(\frac{5}{6})-.234</td>
<td>.2541</td>
</tr>
<tr>
<td>.2947</td>
<td>&quot; &quot;</td>
<td>18</td>
<td>18</td>
<td>.2226</td>
<td>.244</td>
<td>C-.242</td>
<td>.2586</td>
</tr>
<tr>
<td>.294</td>
<td>&quot; &quot;</td>
<td>18</td>
<td>20*</td>
<td>.229</td>
<td>.249</td>
<td>D-.246</td>
<td>.2615</td>
</tr>
<tr>
<td>.3079</td>
<td>5(\frac{1}{6})-.312</td>
<td>19</td>
<td>16</td>
<td>.2268</td>
<td>.251</td>
<td>(\frac{5}{6})-.250</td>
<td>.2673</td>
</tr>
<tr>
<td>.3079</td>
<td>&quot; &quot;</td>
<td>19</td>
<td>18</td>
<td>.2358</td>
<td>.257</td>
<td>F-.257</td>
<td>.2718</td>
</tr>
<tr>
<td>.3079</td>
<td>&quot; &quot;</td>
<td>19</td>
<td>20</td>
<td>.2429</td>
<td>.262</td>
<td>G-.261</td>
<td>.2754</td>
</tr>
<tr>
<td>.321</td>
<td>P-.323</td>
<td>20</td>
<td>16</td>
<td>.2399</td>
<td>.264</td>
<td>1(\frac{5}{6})-.266</td>
<td>.2804</td>
</tr>
<tr>
<td>.321</td>
<td>&quot; &quot;</td>
<td>20</td>
<td>18</td>
<td>.2489</td>
<td>.271</td>
<td>1(\frac{5}{6})-.266</td>
<td>.2849</td>
</tr>
<tr>
<td>.320</td>
<td>&quot; &quot;</td>
<td>20</td>
<td>20*</td>
<td>.255</td>
<td>.275</td>
<td>I-.272</td>
<td>.2875</td>
</tr>
<tr>
<td>.3474</td>
<td>S-.348</td>
<td>22</td>
<td>16</td>
<td>.2663</td>
<td>.291</td>
<td>L-.290</td>
<td>.3068</td>
</tr>
<tr>
<td>.346</td>
<td>&quot; &quot;</td>
<td>22</td>
<td>18*</td>
<td>.2738</td>
<td>.296</td>
<td>1(\frac{5}{6})-.297</td>
<td>.3099</td>
</tr>
<tr>
<td>.3737</td>
<td>5(\frac{1}{6})-.375</td>
<td>24</td>
<td>14</td>
<td>.2809</td>
<td>.309</td>
<td>N-.302</td>
<td>.3273</td>
</tr>
<tr>
<td>.372</td>
<td>&quot; &quot;</td>
<td>24</td>
<td>18*</td>
<td>.2908</td>
<td>.315</td>
<td>5(\frac{1}{6})-.312</td>
<td>.3314</td>
</tr>
<tr>
<td>.3737</td>
<td>&quot; &quot;</td>
<td>24</td>
<td>18</td>
<td>.3016</td>
<td>.323</td>
<td>P-.323</td>
<td>.3376</td>
</tr>
<tr>
<td>.400</td>
<td>1(\frac{5}{6})-.406</td>
<td>26</td>
<td>14</td>
<td>.3072</td>
<td>.335</td>
<td>Q-.332</td>
<td>.3536</td>
</tr>
<tr>
<td>.398</td>
<td>&quot; &quot;</td>
<td>26</td>
<td>16*</td>
<td>.3168</td>
<td>.341</td>
<td>1(\frac{5}{6})-.344</td>
<td>.3574</td>
</tr>
<tr>
<td>.424</td>
<td>(\frac{7}{16})-.437</td>
<td>28</td>
<td>14*</td>
<td>.3312</td>
<td>.359</td>
<td>1(\frac{5}{6})-.359</td>
<td>.3776</td>
</tr>
<tr>
<td>.4263</td>
<td>&quot; &quot;</td>
<td>28</td>
<td>18</td>
<td>.3452</td>
<td>.370</td>
<td>U-.368</td>
<td>.3857</td>
</tr>
<tr>
<td>.450</td>
<td>2(\frac{5}{6})-.453</td>
<td>30</td>
<td>14*</td>
<td>.3572</td>
<td>.385</td>
<td>W-.386</td>
<td>.4036</td>
</tr>
<tr>
<td>.4526</td>
<td>&quot; &quot;</td>
<td>30</td>
<td>16</td>
<td>.3715</td>
<td>.396</td>
<td>X-.397</td>
<td>.412</td>
</tr>
</tbody>
</table>
TABLE 17.—UNITED STATES STANDARD SCREW SIZES

(Tap drill sizes calculated from the formula \( O.D. - \frac{0.9}{N} \) which gives a trifle over \( \frac{3}{8} \) of a full thread. See page 178)

<table>
<thead>
<tr>
<th>Outside diameter (O.D.)</th>
<th>Number of threads per inch</th>
<th>Root diameter (exact) R.D.</th>
<th>Tap drill diameter ( (O.D. - \frac{0.9}{N}) )</th>
<th>Nearest number letter or fractional size tap drill</th>
<th>Pitch diameter (screw thread micrometer reading)</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \frac{1}{8} )</td>
<td>20</td>
<td>0.185</td>
<td>0.205</td>
<td>#6 -0.204</td>
<td>0.2175</td>
</tr>
<tr>
<td>( \frac{5}{32} )</td>
<td>18</td>
<td>0.2403</td>
<td>0.262</td>
<td>G-0.261</td>
<td>0.2764</td>
</tr>
<tr>
<td>( \frac{3}{32} )</td>
<td>16</td>
<td>0.2936</td>
<td>0.319</td>
<td>( \frac{5}{16} )-0.312</td>
<td>0.3344</td>
</tr>
<tr>
<td>( \frac{1}{16} )</td>
<td>14</td>
<td>0.3447</td>
<td>0.373</td>
<td>U-0.368</td>
<td>0.3911</td>
</tr>
<tr>
<td>( \frac{1}{16} )</td>
<td>13</td>
<td>0.4001</td>
<td>0.431</td>
<td>( \frac{3}{64} )-0.421</td>
<td>0.4501</td>
</tr>
<tr>
<td>( \frac{5}{64} )</td>
<td>12</td>
<td>0.4542</td>
<td>0.485</td>
<td>( \frac{3}{16} )-0.484</td>
<td>0.5084</td>
</tr>
<tr>
<td>( \frac{5}{64} )</td>
<td>11</td>
<td>0.5069</td>
<td>0.544</td>
<td>( \frac{1}{12} )-0.531</td>
<td>0.566</td>
</tr>
<tr>
<td>( \frac{1}{8} )</td>
<td>10</td>
<td>0.5201</td>
<td>0.560</td>
<td>( \frac{3}{64} )-0.656</td>
<td>0.6851</td>
</tr>
<tr>
<td>( \frac{1}{8} )</td>
<td>9</td>
<td>0.5307</td>
<td>0.575</td>
<td>( \frac{4}{64} )-0.766</td>
<td>0.8029</td>
</tr>
<tr>
<td>1</td>
<td>8</td>
<td>0.8376</td>
<td>0.888</td>
<td>( \frac{7}{8} )-0.875</td>
<td>0.9188</td>
</tr>
<tr>
<td>( \frac{1}{4} )</td>
<td>7</td>
<td>0.9394</td>
<td>0.995</td>
<td>( \frac{5}{64} )-0.984</td>
<td>1.0322</td>
</tr>
<tr>
<td>( \frac{1}{4} )</td>
<td>6</td>
<td>1.0644</td>
<td>1.120</td>
<td>( \frac{1}{4} )-1.125</td>
<td>1.1572</td>
</tr>
<tr>
<td>( \frac{1}{4} )</td>
<td>6</td>
<td>1.1585</td>
<td>1.225</td>
<td>( \frac{1}{8} )-1.187</td>
<td>1.2668</td>
</tr>
<tr>
<td>( \frac{1}{4} )</td>
<td>6</td>
<td>1.2835</td>
<td>1.350</td>
<td>( \frac{1}{16} )-1.312</td>
<td>1.3918</td>
</tr>
<tr>
<td>( \frac{1}{8} )</td>
<td>5</td>
<td>1.3888</td>
<td>1.461</td>
<td>( \frac{1}{16} )-1.437</td>
<td>1.507</td>
</tr>
<tr>
<td>( \frac{1}{8} )</td>
<td>5</td>
<td>1.4902</td>
<td>1.570</td>
<td>( \frac{11}{16} )-1.562</td>
<td>1.6201</td>
</tr>
<tr>
<td>2</td>
<td>4( \frac{1}{2} )</td>
<td>1.7113</td>
<td>1.800</td>
<td>( \frac{1}{4} )-1.750</td>
<td>1.8557</td>
</tr>
</tbody>
</table>
### Table 18.—S. A. E. Standard Screw Sizes

(Tap drill sizes calculated from the formula $O.D. - \frac{0.9}{N}$ which gives a trifle over $\frac{3}{4}$ of a full thread. See page 178)

<table>
<thead>
<tr>
<th>Outside diameter (O.D.)</th>
<th>Number of threads per inch</th>
<th>Root diameter (exact) R.D.</th>
<th>Tap drill diameter ( (O.D. - \frac{0.9}{N}) )</th>
<th>Nearest number letter or fractional size tap drill</th>
<th>Pitch diameter (screw thread micrometer reading)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\frac{1}{4}$</td>
<td>28</td>
<td>0.2036</td>
<td>0.228</td>
<td>#1-0.228</td>
<td>0.2268</td>
</tr>
<tr>
<td>$\frac{5}{16}$</td>
<td>24</td>
<td>0.2584</td>
<td>0.275</td>
<td>I-0.272</td>
<td>0.2854</td>
</tr>
<tr>
<td>$\frac{3}{8}$</td>
<td>24</td>
<td>0.3209</td>
<td>0.338</td>
<td>$2\frac{1}{64}$-0.328</td>
<td>0.3479</td>
</tr>
<tr>
<td>$\frac{7}{16}$</td>
<td>20</td>
<td>0.3725</td>
<td>0.392</td>
<td>$2\frac{5}{64}$-0.390</td>
<td>0.405</td>
</tr>
<tr>
<td>$\frac{1}{2}$</td>
<td>20</td>
<td>0.435</td>
<td>0.455</td>
<td>$2\frac{9}{64}$-0.453</td>
<td>0.4675</td>
</tr>
<tr>
<td>$\frac{9}{16}$</td>
<td>18</td>
<td>0.4903</td>
<td>0.512</td>
<td>$\frac{1}{2}$-0.500</td>
<td>0.5264</td>
</tr>
<tr>
<td>$\frac{5}{8}$</td>
<td>18</td>
<td>0.5528</td>
<td>0.575</td>
<td>$\frac{3}{4}$-0.562</td>
<td>0.5889</td>
</tr>
<tr>
<td>$\frac{11}{16}$</td>
<td>16</td>
<td>0.6063</td>
<td>0.631</td>
<td>$\frac{5}{8}$-0.625</td>
<td>0.6469</td>
</tr>
<tr>
<td>$\frac{3}{4}$</td>
<td>16</td>
<td>0.6688</td>
<td>0.694</td>
<td>$1\frac{1}{16}$-0.687</td>
<td>0.7094</td>
</tr>
<tr>
<td>$\frac{7}{8}$</td>
<td>14</td>
<td>0.7822</td>
<td>0.811</td>
<td>$1\frac{3}{16}$-0.812</td>
<td>0.8344</td>
</tr>
<tr>
<td>1</td>
<td>14</td>
<td>0.9072</td>
<td>0.936</td>
<td>$1\frac{5}{16}$-0.937</td>
<td>0.9594</td>
</tr>
</tbody>
</table>

### Reasons for Finer Pitches

Threads in automobile work are cut in hard tough materials and do not require to be as coarse as threads cut in cast iron. A screw or bolt of a given size and of finer pitch has greater root diameter and consequently greater strength than a U. S. Std. screw of same size. A fine pitch screw or nut may be set up tighter and does not shake loose as readily as one of coarse pitch.
Table 19.—U. S. Standard Bolts and Nuts

Rough

<table>
<thead>
<tr>
<th>Dia. of Bolt</th>
<th>Threads per Inch</th>
<th>Across Flats A</th>
<th>Across Corners</th>
<th>Thickness</th>
<th>Depth of Thread</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>B</td>
<td>C</td>
<td>Head</td>
<td>Nut</td>
</tr>
<tr>
<td>$\frac{1}{2}$</td>
<td>20</td>
<td>$\frac{1}{4}$</td>
<td>$\frac{1}{4}$</td>
<td>$\frac{1}{4}$</td>
<td>$\frac{1}{4}$</td>
</tr>
<tr>
<td>$\frac{3}{8}$</td>
<td>18</td>
<td>$\frac{1}{8}$</td>
<td>$\frac{1}{8}$</td>
<td>$\frac{1}{8}$</td>
<td>$\frac{1}{8}$</td>
</tr>
<tr>
<td>$\frac{1}{4}$</td>
<td>16</td>
<td>$\frac{1}{4}$</td>
<td>$\frac{1}{4}$</td>
<td>$\frac{1}{4}$</td>
<td>$\frac{1}{4}$</td>
</tr>
<tr>
<td>$\frac{3}{16}$</td>
<td>14</td>
<td>$\frac{3}{16}$</td>
<td>$\frac{3}{16}$</td>
<td>$\frac{3}{16}$</td>
<td>$\frac{3}{16}$</td>
</tr>
<tr>
<td>$\frac{1}{8}$</td>
<td>13</td>
<td>$\frac{1}{8}$</td>
<td>$\frac{1}{8}$</td>
<td>$\frac{1}{8}$</td>
<td>$\frac{1}{8}$</td>
</tr>
<tr>
<td>$\frac{3}{32}$</td>
<td>12</td>
<td>$\frac{3}{32}$</td>
<td>$\frac{3}{32}$</td>
<td>$\frac{3}{32}$</td>
<td>$\frac{3}{32}$</td>
</tr>
<tr>
<td>$\frac{1}{16}$</td>
<td>11</td>
<td>$\frac{1}{16}$</td>
<td>$\frac{1}{16}$</td>
<td>$\frac{1}{16}$</td>
<td>$\frac{1}{16}$</td>
</tr>
<tr>
<td>$\frac{1}{32}$</td>
<td>10</td>
<td>$\frac{1}{32}$</td>
<td>$\frac{1}{32}$</td>
<td>$\frac{1}{32}$</td>
<td>$\frac{1}{32}$</td>
</tr>
<tr>
<td>$\frac{3}{64}$</td>
<td>9</td>
<td>$\frac{3}{64}$</td>
<td>$\frac{3}{64}$</td>
<td>$\frac{3}{64}$</td>
<td>$\frac{3}{64}$</td>
</tr>
<tr>
<td>$\frac{5}{64}$</td>
<td>8</td>
<td>$\frac{5}{64}$</td>
<td>$\frac{5}{64}$</td>
<td>$\frac{5}{64}$</td>
<td>$\frac{5}{64}$</td>
</tr>
<tr>
<td>$\frac{1}{8}$</td>
<td>7</td>
<td>$\frac{1}{8}$</td>
<td>$\frac{1}{8}$</td>
<td>$\frac{1}{8}$</td>
<td>$\frac{1}{8}$</td>
</tr>
<tr>
<td>$\frac{3}{16}$</td>
<td>6</td>
<td>$\frac{3}{16}$</td>
<td>$\frac{3}{16}$</td>
<td>$\frac{3}{16}$</td>
<td>$\frac{3}{16}$</td>
</tr>
<tr>
<td>$\frac{1}{16}$</td>
<td>5</td>
<td>$\frac{1}{16}$</td>
<td>$\frac{1}{16}$</td>
<td>$\frac{1}{16}$</td>
<td>$\frac{1}{16}$</td>
</tr>
<tr>
<td>$\frac{3}{32}$</td>
<td>4</td>
<td>$\frac{3}{32}$</td>
<td>$\frac{3}{32}$</td>
<td>$\frac{3}{32}$</td>
<td>$\frac{3}{32}$</td>
</tr>
<tr>
<td>$\frac{1}{16}$</td>
<td>3</td>
<td>$\frac{1}{16}$</td>
<td>$\frac{1}{16}$</td>
<td>$\frac{1}{16}$</td>
<td>$\frac{1}{16}$</td>
</tr>
<tr>
<td>$\frac{3}{32}$</td>
<td>2</td>
<td>$\frac{3}{32}$</td>
<td>$\frac{3}{32}$</td>
<td>$\frac{3}{32}$</td>
<td>$\frac{3}{32}$</td>
</tr>
<tr>
<td>$\frac{1}{16}$</td>
<td>1</td>
<td>$\frac{1}{16}$</td>
<td>$\frac{1}{16}$</td>
<td>$\frac{1}{16}$</td>
<td>$\frac{1}{16}$</td>
</tr>
<tr>
<td>$\frac{3}{32}$</td>
<td>0.5</td>
<td>$\frac{3}{32}$</td>
<td>$\frac{3}{32}$</td>
<td>$\frac{3}{32}$</td>
<td>$\frac{3}{32}$</td>
</tr>
<tr>
<td>$\frac{1}{16}$</td>
<td>0</td>
<td>$\frac{1}{16}$</td>
<td>$\frac{1}{16}$</td>
<td>$\frac{1}{16}$</td>
<td>$\frac{1}{16}$</td>
</tr>
</tbody>
</table>

Note.—U. S. Government Standard Bolts and Nuts are made to above U. S. or Sellers' Standard Rough Dimensions. The sizes of finished bolt heads and nuts are the same as the sizes of the rough ones, that is for finished work the forgings must be larger than for rough, thus the same wrench may be used on both black and finished heads and nuts. This excellent practice specified by the Government is now generally followed in all commercial work.
TABLE 20.—WROUGHT IRON WELDED PIPE AND PIPE THREADS
Brigg’s Standard

A—Outside diameter of perfect thread or actual outside diameter of pipe.

B—Inside diameter of pipe.

C—Root diameter of thread at end.

D—Outside diameter of thread at end.

E—Length of perfect thread = \( P(4.8 + 0.8A) \).

F—Total length of thread.

H—Height (or depth) of thread = \( 0.8 \times \frac{1}{N} \).

N—Number of threads per inch.

\( P \)—Pitch of thread = \( \frac{1}{N} \).

Taper of thread, \( \frac{3}{4}” \) per foot or 1 in 32 to axis of pipe.

<table>
<thead>
<tr>
<th>Diameter of tube in ins.</th>
<th>Nominal inside</th>
<th>Actual inside B</th>
<th>Actual outside A</th>
<th>Nominal weight per ft. lbs.</th>
<th>Pipe thread dimensions</th>
<th>Size of tap drill</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>C</td>
<td>D</td>
</tr>
<tr>
<td>3/8</td>
<td>0.270</td>
<td>0.405</td>
<td></td>
<td>.24</td>
<td>.057</td>
<td>.334</td>
</tr>
<tr>
<td>1/2</td>
<td>0.364</td>
<td>0.540</td>
<td></td>
<td>.42</td>
<td>.104</td>
<td>.433</td>
</tr>
<tr>
<td>5/8</td>
<td>0.494</td>
<td>0.675</td>
<td></td>
<td>.55</td>
<td>.192</td>
<td>.567</td>
</tr>
<tr>
<td>3/4</td>
<td>0.623</td>
<td>0.840</td>
<td></td>
<td>.83</td>
<td>.305</td>
<td>.702</td>
</tr>
<tr>
<td>7/8</td>
<td>0.824</td>
<td>1.050</td>
<td></td>
<td>1.11</td>
<td>.533</td>
<td>.911</td>
</tr>
<tr>
<td>1</td>
<td>1.048</td>
<td>1.315</td>
<td></td>
<td>1.66</td>
<td>.863</td>
<td>1.144</td>
</tr>
<tr>
<td>11/8</td>
<td>1.380</td>
<td>1.660</td>
<td></td>
<td>2.24</td>
<td>1.496</td>
<td>1.488</td>
</tr>
<tr>
<td>7/4</td>
<td>1.610</td>
<td>1.900</td>
<td></td>
<td>2.67</td>
<td>2.038</td>
<td>1.727</td>
</tr>
<tr>
<td>2</td>
<td>2.067</td>
<td>2.375</td>
<td></td>
<td>3.60</td>
<td>3.355</td>
<td>2.200</td>
</tr>
<tr>
<td>21/8</td>
<td>2.468</td>
<td>2.875</td>
<td></td>
<td>5.73</td>
<td>4.783</td>
<td>2.618</td>
</tr>
</tbody>
</table>

Note: Brigg’s Standard Pipes and Pipe Fittings go to 12” diameter.

It is frequently necessary to drill holes which are to be tapped for pipes and fittings. The Brigg’s Standard is used throughout the United States. The “nominal” inside diameter of the pipe is used to designate the size of the pipe. The fact that there is considerable difference in the “nominal” diameter and actual diameter, especially in the smaller sizes, is often confusing.

For a distance (E) the outside diameter of the pipe tapers and the diameter at the root of the thread follows the same taper. In this part the threads have full depth. For the next two threads the taper at the root continues, but the outside not being tapered these threads
have imperfect tops. The remaining threads are increasingly imperfect on the top and also on the bottom because of the chamfer or bell mouth of the threading die.

In extra strong or double extra strong or hydraulic pipe the additional thickness is on the inside of the pipe and does not effect the thread dimensions. When cutting threads on pipe or pipe fittings set the tool at right angles to the axis of the piece. See page 197.

**Table 21.—The Metric System of Measurement**

**Measures of Length**

1 Millimeter (mm.) = 0.03937079 inch, or about \( \frac{1}{8} \) inch

10 Millimeters = 1 Centimeter (cm.) = 0.3937079 inch

10 Centimeters = 1 Decimeter (dm.) = 3.937079 inch

10 Decimeters = 1 meter (m.) = 39.37079 inches, 3.2808992 feet, or 1.09361 yards

10 Meters = 1 Decameter (Dm.) = 32.808992 feet

10 Decameters = 1 Hectometer (Hm.) = 19.927817 rods

10 Hectometers = 1 Kilometer (Km.) = 1093.61 yards, or 0.6213724 mile

10 Kilometers = 1 Myriameter (Mm.) = 6.213724 miles

1 inch = 2.54 cm., 1 foot = 0.3048 m., 1 yard = 0.9144 m., 1 rod = 0.5029 Dm., 1 mile = 1.6093 Km.

**Measures of Weight**

1 Gramme (g.) = 15.4324874 gr. Troy, or 0.03215 oz. Troy, or 0.03527398 oz. avoirdupois.

10 Grammes = 1 Decagramme (Dg.) = 0.3527398 oz. avoirdupois.

10 Decagrammes = 1 Hectogramme (Hg.) = 3.527398 oz. avoirdupois.

10 Hectogrammes = 1 Kilogramme (Kg.) = 2.20462125 lb.

1000 Kilogrammes = 1 Tonne (T.) = 2204.62125 lb. or 1.1023 tons of 2000 lb. or 0.9842 ton of 2240 lb. or 19.68 cwts.

1 grain = 0.0648 g., 1 oz. avoirdupois = 28.35 g., 1 lb. = 0.4536 Kg., 1 ton 2000 lb. = 0.9072 T., 1 ton 2240 lb. = 1.016 T., or 1016 Kg.

**Measures of Capacity**

1 Liter (l.) = 1 cubic decimeter = 61.0270515 cubic in., or 0.03531 cu. ft. or 1.0567 liquid qts. or 0.908 dry qt. or 0.26417 Amer. gal.

10 Liter = 1 Decaliter (Dl.) = 2.6417 gal., or 1.135 pk.

10 Decaliters = 1 Hectoliter (hl.) = 2.8375 bu.

10 Hectoliters = 1 Kiloliter (Kl.) = 61027.0515 cu. in., or 28.375 bu.

1 cu. foot = 28.317 l., 1 gallon, Amer. = 3.785 l., 1 gallon, Brit. = 4.543 l.
### Table 22.—Metric Conversion Table

<table>
<thead>
<tr>
<th>Metric</th>
<th>Conversion Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>millimeters</td>
<td>× 0.03937 = Inches</td>
</tr>
<tr>
<td>millimeters</td>
<td>= 25.400 × Inches</td>
</tr>
<tr>
<td>meters</td>
<td>× 3.2809 = Feet</td>
</tr>
<tr>
<td>meters</td>
<td>= 0.3048 × Feet</td>
</tr>
<tr>
<td>kilometers</td>
<td>× 0.621377 = Miles</td>
</tr>
<tr>
<td>kilometers</td>
<td>= 1.6093 × Miles</td>
</tr>
<tr>
<td>square centimeters</td>
<td>× 0.15500 = Square inches</td>
</tr>
<tr>
<td>square centimeters</td>
<td>= 6.4515 × Square inches</td>
</tr>
<tr>
<td>square meters</td>
<td>× 10.76410 = Square feet</td>
</tr>
<tr>
<td>square meters</td>
<td>= 0.09290 × Square feet</td>
</tr>
<tr>
<td>square kilometers</td>
<td>× 247.1098 = Acres</td>
</tr>
<tr>
<td>square kilometers</td>
<td>= 0.00405 × Acres</td>
</tr>
<tr>
<td>hectares</td>
<td>× 2.471 = Acres</td>
</tr>
<tr>
<td>hectares</td>
<td>= 0.4047 × Acres</td>
</tr>
<tr>
<td>cubic centimeters</td>
<td>× 0.061025 = Cubic inches</td>
</tr>
<tr>
<td>cubic centimeters</td>
<td>= 16.3866 × Cubic inches</td>
</tr>
<tr>
<td>cubic meters</td>
<td>× 35.3156 = Cubic feet</td>
</tr>
<tr>
<td>cubic meters</td>
<td>= 0.02832 × Cubic feet</td>
</tr>
<tr>
<td>cubic meters</td>
<td>= 1.308 = Cubic yards</td>
</tr>
<tr>
<td>cubic meters</td>
<td>= 0.765 × Cubic yards</td>
</tr>
<tr>
<td>ers</td>
<td>× 61.023 = Cubic inches</td>
</tr>
<tr>
<td>ers</td>
<td>= 0.01639 × Cubic inches</td>
</tr>
<tr>
<td>ers</td>
<td>× 0.26418 = U. S. gallons</td>
</tr>
<tr>
<td>ers</td>
<td>= 3.7854 = U. S. gallons</td>
</tr>
<tr>
<td>ouns</td>
<td>× 15.4324 = Grains</td>
</tr>
<tr>
<td>ouns</td>
<td>= 0.0648 × Grains</td>
</tr>
<tr>
<td>ouns</td>
<td>× 0.03527 = Ounces, av’dupois</td>
</tr>
<tr>
<td>ouns</td>
<td>= 28.3495 × Ounces, av’dupois</td>
</tr>
<tr>
<td>ograms</td>
<td>× 2.2046 = Pounds</td>
</tr>
<tr>
<td>ograms</td>
<td>= 0.4536 × Pounds</td>
</tr>
<tr>
<td>oz’s per sq. centimeter</td>
<td>× 14.2231 = Lb. per sq. inch</td>
</tr>
<tr>
<td>oz’s per sq. centimeter</td>
<td>= 0.0703 × Lb. per sq. inch</td>
</tr>
<tr>
<td>ogram per cubic meter</td>
<td>× 0.06243 = Lb. per cubic foot</td>
</tr>
<tr>
<td>ogram per cubic meter</td>
<td>= 16.01890 × Lb. per cubic foot</td>
</tr>
<tr>
<td>tric tons (1000 kilog’s)</td>
<td>× 1.1023 = Tons (2000 lb.)</td>
</tr>
<tr>
<td>tric tons (1000 kilog’s)</td>
<td>= 0.9072 × Tons (2000 lb.)</td>
</tr>
<tr>
<td>horse-power</td>
<td>× 1.3405 = Horse-powers</td>
</tr>
<tr>
<td>horse-power</td>
<td>= 0.746 × Horse-powers</td>
</tr>
<tr>
<td>horse-power</td>
<td>× 3.9683 = B. T. units</td>
</tr>
<tr>
<td>horse-power</td>
<td>= 2.520 = B. T. units</td>
</tr>
<tr>
<td>dollars</td>
<td>× 1.93 = Dollars</td>
</tr>
<tr>
<td>dollars</td>
<td>= 5.18 × Dollars</td>
</tr>
</tbody>
</table>

By courtesy of the American Machinist, New York.
## Table 23.—Decimal Equivalents of the Number and Letter Sizes of Twist Drills

<table>
<thead>
<tr>
<th>No.</th>
<th>Size in decimals</th>
<th>No.</th>
<th>Size in decimals</th>
<th>No.</th>
<th>Size in decimals</th>
<th>No.</th>
<th>Size in decimals</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>.2280</td>
<td>21</td>
<td>.1590</td>
<td>41</td>
<td>.0960</td>
<td>61</td>
<td>.0390</td>
</tr>
<tr>
<td>2</td>
<td>.2210</td>
<td>22</td>
<td>.1570</td>
<td>42</td>
<td>.0935</td>
<td>62</td>
<td>.0380</td>
</tr>
<tr>
<td>3</td>
<td>.2130</td>
<td>23</td>
<td>.1540</td>
<td>43</td>
<td>.0890</td>
<td>63</td>
<td>.0370</td>
</tr>
<tr>
<td>4</td>
<td>.2090</td>
<td>24</td>
<td>.1520</td>
<td>44</td>
<td>.0860</td>
<td>64</td>
<td>.0360</td>
</tr>
<tr>
<td>5</td>
<td>.2055</td>
<td>25</td>
<td>.1495</td>
<td>45</td>
<td>.0820</td>
<td>65</td>
<td>.0350</td>
</tr>
<tr>
<td>6</td>
<td>.2040</td>
<td>26</td>
<td>.1470</td>
<td>46</td>
<td>.0810</td>
<td>66</td>
<td>.0330</td>
</tr>
<tr>
<td>7</td>
<td>.2010</td>
<td>27</td>
<td>.1440</td>
<td>47</td>
<td>.0785</td>
<td>67</td>
<td>.0320</td>
</tr>
<tr>
<td>8</td>
<td>.1990</td>
<td>28</td>
<td>.1405</td>
<td>48</td>
<td>.0760</td>
<td>68</td>
<td>.0310</td>
</tr>
<tr>
<td>9</td>
<td>.1960</td>
<td>29</td>
<td>.1360</td>
<td>49</td>
<td>.0730</td>
<td>69</td>
<td>.0292</td>
</tr>
<tr>
<td>10</td>
<td>.1935</td>
<td>30</td>
<td>.1285</td>
<td>50</td>
<td>.0700</td>
<td>70</td>
<td>.0280</td>
</tr>
<tr>
<td>11</td>
<td>.1910</td>
<td>31</td>
<td>.1200</td>
<td>51</td>
<td>.0670</td>
<td>71</td>
<td>.0260</td>
</tr>
<tr>
<td>12</td>
<td>.1890</td>
<td>32</td>
<td>.1160</td>
<td>52</td>
<td>.0635</td>
<td>72</td>
<td>.0250</td>
</tr>
<tr>
<td>13</td>
<td>.1850</td>
<td>33</td>
<td>.1130</td>
<td>53</td>
<td>.0595</td>
<td>73</td>
<td>.0240</td>
</tr>
<tr>
<td>14</td>
<td>.1820</td>
<td>34</td>
<td>.1110</td>
<td>54</td>
<td>.0550</td>
<td>74</td>
<td>.0225</td>
</tr>
<tr>
<td>15</td>
<td>.1800</td>
<td>35</td>
<td>.1100</td>
<td>55</td>
<td>.0520</td>
<td>75</td>
<td>.0210</td>
</tr>
<tr>
<td>16</td>
<td>.1770</td>
<td>36</td>
<td>.1065</td>
<td>56</td>
<td>.0465</td>
<td>76</td>
<td>.0200</td>
</tr>
<tr>
<td>17</td>
<td>.1730</td>
<td>37</td>
<td>.1040</td>
<td>57</td>
<td>.0430</td>
<td>77</td>
<td>.0180</td>
</tr>
<tr>
<td>18</td>
<td>.1695</td>
<td>38</td>
<td>.1015</td>
<td>58</td>
<td>.0420</td>
<td>78</td>
<td>.0160</td>
</tr>
<tr>
<td>19</td>
<td>.1660</td>
<td>39</td>
<td>.0995</td>
<td>59</td>
<td>.0410</td>
<td>79</td>
<td>.0145</td>
</tr>
<tr>
<td>20</td>
<td>.1610</td>
<td>40</td>
<td>.0980</td>
<td>60</td>
<td>.0400</td>
<td>80</td>
<td>.0135</td>
</tr>
</tbody>
</table>

## Letter Sizes of Drills

<table>
<thead>
<tr>
<th>Letter</th>
<th>Size in decimals</th>
<th>Letter</th>
<th>Size in decimals</th>
</tr>
</thead>
<tbody>
<tr>
<td>A (\frac{1}{64})</td>
<td>.234</td>
<td>N</td>
<td>.302</td>
</tr>
<tr>
<td>B</td>
<td>.238</td>
<td>O (\frac{5}{64})</td>
<td>.316</td>
</tr>
<tr>
<td>C</td>
<td>.242</td>
<td>P (\frac{2}{64})</td>
<td>.323</td>
</tr>
<tr>
<td>D</td>
<td>.246</td>
<td>Q</td>
<td>.332</td>
</tr>
<tr>
<td>E (\frac{1}{4})</td>
<td>.250</td>
<td>R (\frac{1}{32})</td>
<td>.339</td>
</tr>
<tr>
<td>F</td>
<td>.257</td>
<td>S</td>
<td>.348</td>
</tr>
<tr>
<td>G (\frac{1}{64})</td>
<td>.261</td>
<td>T (\frac{3}{64})</td>
<td>.358</td>
</tr>
<tr>
<td>H (\frac{1}{64})</td>
<td>.266</td>
<td>U</td>
<td>.368</td>
</tr>
<tr>
<td>I</td>
<td>.272</td>
<td>V (\frac{5}{64})</td>
<td>.377</td>
</tr>
<tr>
<td>J</td>
<td>.277</td>
<td>W (\frac{2}{64})</td>
<td>.386</td>
</tr>
<tr>
<td>K (\frac{2}{64})</td>
<td>.281</td>
<td>X</td>
<td>.397</td>
</tr>
<tr>
<td>L</td>
<td>.290</td>
<td>Y (\frac{1}{32})</td>
<td>.404</td>
</tr>
<tr>
<td>M (\frac{1}{64})</td>
<td>.295</td>
<td>Z</td>
<td>.413</td>
</tr>
</tbody>
</table>
INDEX

A

Accuracy, 19
  of live center, 99
Acme standard screw threads, table of, 306
  standard tap threads, table of, 307
  thread, cutting an, 202
Adjustable reamers, 140
Adjusting work on centers, 91
A.L.A.M.—see S.A.E. Std., 313
Alignment of centers, 88, 105
Allowance for reaming, 139
Angle, clearance, 63
  cutting, 62
    definition of, 163
  helical, 200
  plate, 209
  rake, 64
  slant, of thread, 200
  wrench, 223
Angles and corresponding tapers, table of, 299
  classification of, 164
  dimensioning of, 164 and 166
  measurement of, 167
  turning of, 165
Annealing steel, 89
Apprentice, definition of, 16
Apron mechanism, 48
Arbor press, 120
  for shell reamers, 189
A.S.M.E., standard screw sizes, 309
Assembler, definition of, 15
Attachment, taper, 156
Automobile threads—see S.A.E. Std., 313

B

Babbit, 266
Babbitting a bearing, 267
Back belt—see Reserve belt, 198
Back gears, 40 and 53
Backlash—see Lost motion, 105 and 160
Bearing, babbitting a, 267
Babbitt, 266
Metal—see Babbit, 266
  running fit for a, 280
Bed, lathe, 29
  belt ratio, 43
  belts, cementing, 283
  changing, 42
  fastening, 281
  lacing, 281
  Bench hand, definition of, 16
  Bench plate, 229
    work, definitions of, 14 and 217
    tools used in, 217
  Bent lathe tools, 60
  Bevel protractor, 167
  Vernier, reading of, 287
  Bits for turning tools, 64
  Blue vitriol, 232
  Body size drills—tables, 309 to 313
  Bolts and nuts, U.S. Std., table of, 314
  Boring bell-mouthed holes, caution against, 149
  bored holes, 148
  cut, measuring the, 150
  reasons for, 147
  in a lathe, 145
  tapers, 160
  threads, 198
  tools, 145
  Boring mill, definition of, 13
  Brass, turning tool for, 60
  Braising, 265
  Briggs standard pipe sizes, 315
  British Association standard screw threads, table of, 303
  British standard (Whitworth) threads, table of, 302
  Brown and Sharpe tapers, 153 and table of, 297
  Brown and Sharpe 29° worm threads, table of, 308

C

Calculating change gears for thread cutting, 187
  cutting speeds, 73
  gear velocities, 279
  pulley speeds, 277
Caliper, 75
  gauging with a, 76
  hermaphrodite, 84
  measuring with an inside, 149
  measuring with an outside, 76
  reading a, 77 and 149
  setting a, 77 and 149
  transfer, 76
  Vernier, 286
  Cape chisel, 235
  Carbon steel, 370
Carbon steel, precaution when centering, 83
Care of files, 245
when starting lathe, 32
Carefulness, 18
Carriage, details of lathe, 31
Cast iron, care in centering, 83
iron, care in chipping, 238
iron, care in turning, 84
Cat head, 117
Cementing belts, 283
Center holes—see Centers, work
punch, 86
reamer, 87
rest, 115
square, 85
Centering, 82
machine, 84
methods, 84
Centers, lathe, 100
lathe, alignment of, 88
lathe, cleaning, 100
lathe, grinding, 101
lathe, oiling, 104
lathe, removing, 31
lathe, truing, 100
Centers, work, drilling, 87
work, locating, 83
work, sizes of, 88
Change gear calculation for simple train, 187
gear calculation for compound train, 188
gear calculation for metric thread, 202
gear progression, 188
gear train, 186
gears, quick, 46
gears, removing, 46
Characteristics of a machinist, 18
Chart of threads, 174
Chasing dial, 199
Chattering, 130 and 148
Check nut—action of, 224
Chip—definition of, 70
Chipping, 235
operation of, 237
Chisels—various kinds of, 235
grinding, 236
holding, 237
Chucking work, 125
reamers, 138
Chucks, 125
adjusting work in, 127
mounting, on spindle, 127
removing, 127
Circles, locating centers of, for drilling layout, 233
Circular cut files, 251
Classification of machine shop employees 14
Clamp, U, 209
Clamping work on face plate, 212
Cleaning machine, 25
Clearance angles of drills, 133
angles of lathe tools, 63
Cold chisels, 235
Combination drill and countersink, 87
set, 85
Compound gears, 38 and 188
gears, velocity of, 280
rest, 165, 195
rest, cutting threads with, 195
rest, setting, 165
Conversion table, metric, 317
Copper solution, 231
Coppers, soldering, 283
Cored holes, care in boring, 148
Countershaft, 34
Countersink, 87
Crank-shaft turning, 123
Cross belt—see Reverse belt, 198
feed, 49
Curved cut files, 251
Cut, finishing, 108
of files, 240
roughing, 107
Cutting Acme threads, 203
angle, 62
double threads, 205
inside threads, 198
metric threads, 202
off in a lathe, 128
off tool, 128
speed calculations, 72
speed, conditions governing, 71
speeds for drills, lathe work, etc.,
table of, 293
speeds for various metals, 73
square threads, 199
tools, angles of, 62
tools, bent, 60
tools, bits for, 65
tools, blades for, 65
tools, brass, 60
tools, cutting-off, 128
tools, forged, chart of, 6
tools, forming, 114
tools, goose-neck, 60
tools, grinding of, 67
tools, height of, 64
tools, lubrication of, 104, 136
tools, parting—see Cutting-off, 128
tools, patent, chart of, 64
INDEX

Cutting tools, setting of, 101
  tools, shoulder, 113
  tools, spotting, 60, 141
  tools, spring, 60, 114
  tools, square-nose, 60, 100
  tools, thread, U.S. Std., 60, 191, 195;
    square, 199; Acme, 202
  tools, turning, 60
  U.S. Std. threads, 190, 195

D
Decarbonisation of steel, 83
Decimal equivalents of number and letter
  sizes of drills, table of, 318
  equivalents of an inch, table of, 292
Diamond point cutting tool, 60
Die-stock, 182
Die, threading, 182
Dimensions of threads, 173
Divider, 230
Dogs, lathe, 82
Dont’s in filing, 252
  in tool grinding, 68
Double threads, cutting, 205
Draw filing, 250
Draw-in chuck, 126
Drill gauge, 133
  holder, 142
Drilling in a lathe, 131, 141
  machine, definition of, 7
Drills, cutting lubricants for, 137
  farmer, 135
  feeds of, 136
  flat, 132
  for brass, 135
  removing broken, 89
  sharpening, 133
  speeds of, 136
  table of letter and number sizes of,
    318
  tap, calculating sizes of, 178
  tap, table of sizes of, for machine
    screws, 309
  twist, 132
Drive fit, 281
Duplicate pieces, turning, 108

E
Ecocentric, turning, 123
Emery cloth, 110
Engine lathe, 22
Equipment, machine shop, 5
Expansion mandrels, 121
  reamers—see Adjustable reamers, 140
  21

F
Face plate work, 208
Facing, 92
  radial, 128
  tool—see Side tool, 60, 92
Fastening a belt, 281
Feather, 47
Feed, definition of, 70
  direction of, 103
  for drilling, 136
  mechanism, 31, 43
  operation of lathe, 43
File card, 245
Files, blunt, 245
  care of, 245
  convexity of, 244
  cuts of, 240
  handles, 245, 252
  holding, 246
  needle handle, 252
  pinning of, 245
  shapes of, 242
  taper of, 244
Filing, bench, 240
  cross, 246
  draw, 250
  in a lathe, 109
  purpose of, 109, 240
  soft metals, 250
Finishing cut, 108
Fits, machine, 280
Fitting of tapers, 157
Flat drill, 123
  scraper, 255
Floor hand, definition of, 16
  work, definition of, 14
Fluted machine reamer, 138
Flux for brazing, 265
  for soldering, 262
Follower rest, 118
Force fit, 281
Forged lathe tools, 60
Forming tool, 114
Frame, hack-saw, 226
French standard screw threads, table of,
  304
Frosting scraped surfaces, 255

G
Gauge, Acme thread, 202
  center, 101
  drill, 133
  height, 231
  pitch, 192
INDEX

Gauge, surface, 86
taper, 157
thread—see Pitch gauge, 192
Gauging tapers, 158
and measuring, definitions, 76
Gear, intermediate, 37
train, compound, 38, 138
train, simple, 37
translating, for metric threads, 203
Geared head, 54
Gearing, terms used in, 183
compound, 38, 138
for thread cutting, 183
Gears, back, 40, 52
change, 43
change, calculating for threads, 187
quick change, 46
tumbler, 43, 45, 185
velocity ratio of, 36, 289
Geometrical progression, 283
Gouge chisel, 236
Graduations on cross feed screw, 104
Grinding chisels, 236
cutting tools, 67
drills, 133
lathe centers, 101
machine, definition of, 12
Grooving—see Necking, 114

H
Hack saws, 225
Half center, 93
Half round scraper, 259
Hammers, 218
Hand reamers, 139
Hardening steel, 271
Hardness of dead center, 101
of cold chisels, 235
of hack saws, 225
of lathe tools, 274
of scrapers, 255
of screw drivers, 219
Head, geared, 54
Headstock, 29, 41
Height gauge, 261
Helical angle, 200
Hermaphrodite caliper, 84
High speed steel, 275
speed drills, speeds and feeds for, 137
speed tools, speeds and feeds for, 72
Holder, boring tool, 147
die, 132
drill, 142
Holders, tool, patent, 64
Holding work in a lathe, 82
Horizontal boring mill, definition of, 13

I
Increment cut (reamers), 140
Independent chuck, setting work in, 127
Index plate, 188
Indicators, test, 106, 210
speed, 41
Inside thread, cutting an, 198
Intermediate gear, 46
gear bracket, 46
International screw threads, table of, 305

J
Jarno taper, 154
Jaws, reversible chuck, 126

K
Keenness, angle of—see Cutting angle, 62
Keys, feather, 47
Knurling, 118

L
Lacing belts, 281
Land, 138
Lathe, definition of, 6, 32
high duty, 51
parts of, 27
turret, 96
units of, 29
Laying out work, 229
operation of, 232
out work for drilling, 233
tools used in, 229
Lead number, 187
of thread, 171
screw, 32
Left hand threads, 197
Letter sizes of drills, table of, 318
Limit gauges, 78
Lock pin, 39
Lost motion of cross feed screw, 105
motion of taper attachment, 160
Lubrication of cutting-off tools, 129
of drills, 137
of taps, 180
of thread tool, 193
of threading die, 183
of turning tools, 104

M
Machine hand, definition of, 16
operator, definition of, 15
reamers, 138
INDEX

Machine screw sizes, table of, 309
  shop, definition of, 4
  shop equipment, 5
Machine, definition of, 16
  specialized, definition of, 15
Machine’s helper, 15
Mandrel, 119
  expansion, 121
  gang, 122
  press, 120
  standard, 119
  taper, 121
  taper shank, 123
  threaded, 123
  using a, 120
Mandrel press (arbor press), 120
Measuring a hole, 149
  rod, 209
  threads (three wire method), 194
  with a caliper, 76
Metall’s experiment, 271
Meter, 202
Metric conversion table, 317
  measures, tables of, 316
  thread, calculations for, 202
  thread, cutting a, 202
  threads, tables of, 304, 305
Micrometer, 78
  screw-thread, 289
  ten-thousandths, 288
Milling machine, definition of, 9
  Morse standard tapers, 153
  standard tapers, table of, 295
Multiple threads, 204

N
Necking, 114
Needle handle files, 252
Number sizes of drills, table of, 318
Nurling—see Knurling, 118
Nut arbor, 122
Nuts, table of standard bolts and, 314

O
Oiling a machine, 25

P
Parallels, 210
Parting tool—see Cutting-off tool, 128
Parts of lathe, 26
Patent tool holders, 64
Pinning of files, 245
Pins, taper, and reamers, table of, 298
Pipe threads, table of, 315
Pitch diameter of machine screws, table of, 309
  diameter of thread, 289
  gauge for threads, 192
Planer, definition of, 8
Plate, angle, 209
  bench, 229
  index, 188
  surface, 229, 255
Polishing in a lathe, 110
Prick punch, 229
Progression, change gear, 188
  geometrical, 283
Protractor, bevel, 167
  bevel, with Vernier, 287
Pulleys, speeds of, 85, 277
Punch, center, 86
  prick, 229

Q
Questions on angles, 168
  babbitting, 269
  boring, 150
  centering work, 89
  chipping, 389
  chucking, 130
  cutting speeds, 73
  cutting tools, 69
  drilling, 137 and 144
  face plate work, 213
  facing, 95
  files, 253
  gear velocities, 38
  hack saws, 228
  hardening and tempering, 276
  lathe construction, 33, 51
  laying out work, 233
  measuring, 81
  pulley speeds, 38
  reaming, 144, 150
  scraping, 260
  screw-drivers, 220
  soldering, 266
  tapers, 161
  taper turning, 162
  tapping, 181
  thread cutting, 206
  threading dies, 183
  threads, 176
  turning, 111, 123
  wrenches, 224
Quick change gears, 46
Quill, 40
Scrapers, sharpening, 256
  three cornered, 260
Scraping, 254
  curved surfaces, 259
  flat surfaces, 257
  reasons for, 254
  tools and gauges, 255
Screw cutting—see Thread cutting, 170
  -driver, 219
  gear, 44
  -plate—see Die stock, 182
  slotting hack saws, 228
Scriber, 230
Scribing lines, 231
Setting of boring tool, 149
  of cutting-off tool, 129
  of thread tool, 191
  of turning tool, 101
  over tail stock for tapered work, 154
Shaft—counter, 34
  jack, 278
  line, 34 and 278
Shanks, taper, purpose of, 152
Shaper, definition of, 8
Shapes of files, 242
Shell reamers, 139
Shoulders, turning, 112
Shoulder turning tool, 113
Shrink fit, 281
Side tool, 60, 92
Sizes of machine screws, table of, 309
Skill, definition of, 217
Slant angle of thread, 200
  angle of square thread tool, calculating, 199
Sliding fit, 280
Slope of tool—see Rake angle
Society Automobile Engineers Std.
  threads, 313
Soldering, 261
  coppers, 263
  fluxes, 262
  operation of, 264
Speed, cutting, calculations for, 72
  definition of, 70
  indicator, 41
  of work for polishing in a lathe, 111
  setting the, 107
Speeds, cutting, table of, 293
  of drills, 136
  of pulleys, 35, 70, 277
  spindle, 39
Spelter, 285
Spindle speeds, direct, 39
  speeds, indirect, 40
INDEX

Spotting the center, 141
tool, 141
of work for center rest, 115
Spring chuck, 126
collet, 126
of boring tools, 149
Square, combination, 85
machinist's, 249
thread, 174, 199
thread, cutting a, 199
Squaring ends, 93
Standard tapers, 153
machine tools, 5
threads, 172
Steady rest—see Center rest, 115
Steel, 269
carbon tool, 270
hardening, 271
heat treatment of high-speed, 276
high-speed, 275
machine, 270
soft, 270
tempering, 273
Straight edge, 255
Stud, shouldered, 206
Suggestions for beginners, 28
Surface gauge, 86

T

Tables, list of, 291
Tail stock, details of, 30
Tap drills, 178
drills for machine screw sizes, tables of, 309, 312, 313
Taper attachment, 158
files, 244
filing a, 158
holes, fitting, 161
pins and reamers, table of, 298
reamers, 140
turning, 154
Tapered work, threading, 197
Tapers, 152
boring, 160
Brown and Sharpe, table of, 297
Brown and Sharpe, use of, 153
gauging, 157
Jarno, 153
measuring, 156
Morse standard, table of, 295
Morse standard, use of, 153
per foot and corresponding angles, table of, 299
Tapping, lubricants for, 180
operation of, 179

Tape, 177
relief of, 178
Taylor-White process, 276
Temper colors, 273, 274
Tempering steel, 273
Test indicators, 106, 210
Testing filed work, 248
scraped work, 257
Thread, Acme, cutting an, 202
boring a, 198
catching the, after resetting tool, 197
without reverse belt, 198
cutting a, 190-
hints, 191
mechanism, 32, 43
with compound rest, 195
without reverse belt, 198
dimensions, 173
left hand, cutting a, 197
micrometer, 289
metric, calculations for, 202
cutting a, 202
parts of, 170
slant angle of, 200
stop, 193
square, calculating lead angle of, 200
cutting a, 199
Threading die, 182

Threads, Acme screw, table of, 306
Acme tap, table of, 307
automobile—see S. A. E. Std., 313
British Association Std., screw, table of, 303
calculating change gears for, 187
chart of standard shapes of, 174
counting, per inch, 192
cutting double, 205
multiple, 205
formula for finding root dia. of U. S. Std., 176
French standard screw, table of, 304
International standard screw, table of, 305
measuring, 194
multiple, 204
pipe, table of Brigg's standard, 315
S. A. E. Std. screw, table of, 313
taper, cutting, 197
U. S. Std. screw, table of, 300
Whitworth standard screw, table of, 302
worm, B. & S. 29°, table of, 308
Three cornered scraper, 260
Three wire method of measuring threads, 194
Time element, the, 70
INDEX

Tinning a soldering copper, 264
a surface, 265

Tool bits, 65
blades, 65
boring, 145
cutting-off, 128
cornering—see Shoulder turning, 113
diamond point, 60
forming, 114
holders, 64
knurling, 118
-maker, definition of, 16
parting—see Cutting-off, 128
setting the, 101
shoulder turning, 113
side, 60, 92
spotting, 141
spring, 60
square-nose, 60, 100
thread, 191
with side rake, 195

Tools, cutting, lubrication of—see Lubrication.
 forged lathe (chart), 61
grinding, 67
used in bench work, 217
in face plate work, 208
in lay-out work, 229

Train, gear, simple, 37
gear compound, 38
pulley, 278

Trammels, 231

Translating gear for metric threads, 203

Tumbler gears, 43, 185

Turning duplicate pieces, 108
in a lathe, 96
the finishing cut, 108
the roughing cut, 107
shoulders, 112

Turret lathe, 96

Twist drills, 132

Twist drills, letter sizes, table of, 318
number sizes, table of, 318

U

Unit parts of lathe, 29

U. S. Std. bolts and nuts, table of, 314
screw sizes, table of, 312
screw threads, calculating sizes of tap drills for, 17
dimensions of, 173
measuring a, 195
table of tap drills for, 312

V

V-thread, 174
measuring a, 195

Velocities of gears, rules for, 279

Vernier, principle of, 284
caliper, 286
for bevel protractor, 287
for micrometer, 288
reading a, 286

Vertical boring mill, definition of, 13
Vixen file, 251

W

Whitworth standard screw threads, table of, 302
standard screw threads, measuring, 195

Wiggier, 209

Worm threads, table of, 308
Wrench, tap, 177

Wrenches, 221
chart of, 222
ratchet, 223
use of, 221
value of angle, 223
<table>
<thead>
<tr>
<th>RETURN</th>
<th>CIRCULATION DEPARTMENT</th>
</tr>
</thead>
<tbody>
<tr>
<td>TO</td>
<td>202 Main Library</td>
</tr>
<tr>
<td>LOAN PERIOD</td>
<td>2 3</td>
</tr>
<tr>
<td>HOME USE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>4 5 6</td>
</tr>
</tbody>
</table>

ALL BOOKS MAY BE RECALLED AFTER 7 DAYS

RENEWALS AND RECHARGES MAY BE MADE 4 DAYS PRIOR TO DUE DATE.
LOAN PERIODS ARE 1-MONTH, 3-MONTHS, AND 1-YEAR.
RENEWALS: CALL (415) 642-3405

DUE AS STAMPED BELOW

ALTAPLE OCT 8 1 1989

APR 14 96

RECEIVED

MAR 1 5 1996

CIRCULATION DEPT.