1.0 SCOPE

1.1 The interpretations in this specification apply to all drawings/models specifying 1E0100 Forger Instructions or 1E0576 Foundry Instructions that use the profile tolerance system to control cast or forged surfaces and machine allowance with respect to a datum reference system. 1E2600 Interpretation does not appear on drawings/models.

1.2 Table of Contents

1.3 Index

2.0 INTRODUCTION

2.1 Beginning in the mid-sixties, the ever increasing pressure to improve productivity forced manufacturers to use more and more automated manufacturing processes such as tape controlled (numerically controlled) machines and transfer lines.

2.2 Unfortunately, most automated machines cannot equalize the location of the part to adjust or balance machine allowance, which leaves to chance the occurrence of machining problems that vary from cutting air and failure to clean-up where there is not enough machine stock, to breaking up tools when machine stock exceeds the amount that was programmed on the tape.

2.3 In order to maximize productivity on automated machines, it is essential that a profile tolerance system be employed to control location of the surfaces of the rough casting or forging in a fixture to provide a predictable amount of stock (material) to be removed by machining.

3.0 APPLICATION

3.1 The interpretations in this specification are applicable to drawings/models using the profile tolerance system as part of Specifications 1E0100 Forger Instructions and 1E0576 Foundry Instructions. Specification 1E2600 will not appear on drawings/models.

3.2 The current profile tolerance system has been applied to castings since 1970 and to forgings since 1974 to establish positive, precise communication between engineering, purchasing, casting supplier, forging supplier, manufacturing, and inspection, and to provide the means to optimize production.

Although it is not the primary purpose, the contents of this specification can also be used as a self-teaching training manual which conveys the basic concepts of the profile tolerance system to forging and casting suppliers as well as to Caterpillar Inc. Manufacturing and Inspection personnel.

4.0 IDENTIFICATION

Referring to Figure 1 and the following may identify casting and forging drawings that use the profile tolerance system:

4.1 The key identifying feature is the dimension and tolerance (Dim and Tol) block shown in the upper right corner.

4.2 A profile (⌓) tolerance is normally related to a functional set of Datums (ABC) and a processing set of Datums (XYZ) in the Dim and Tol block. Datums XYZ are used to inspect the rough casting or forging, locate it for machining, or locate it for assembly when the casting or forging is welded into an assembly before machining. Datums ABC relate to function of the completed part and are normally used to inspect the completed part including all unmachined surfaces that remain on the completed part.

4.3 The machining required symbol (▽) is used to indicate surfaces that must be machined.

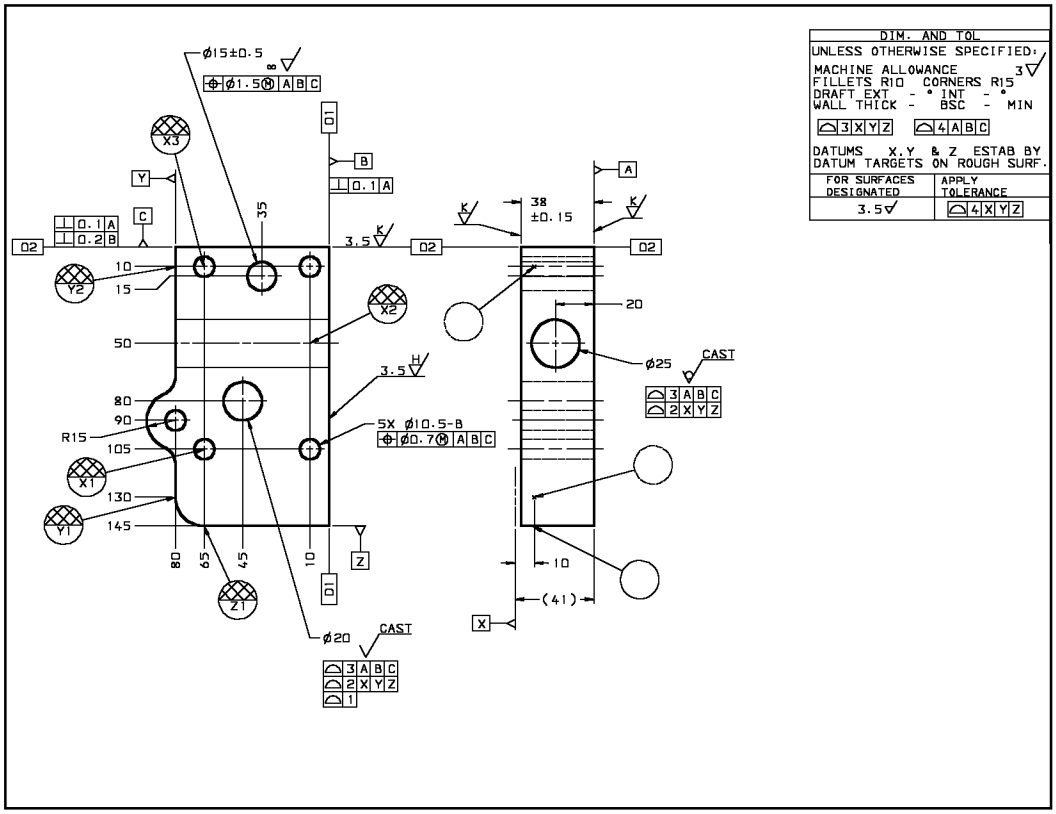


Figure 1 - Identification-Profile Tolerance System

5.0 PROFILE TOLERANCE SYSTEM

The profile tolerance system employs datums, datum targets, basic dimensions, and profile tolerances to define and control the rough casting or rough forging geometry relative to a processing datum system and also to control all portions of the rough contour that remain on the finished part relative to a functional datum system. The profile tolerance drawing is essentially a view of the finished part with the missing portions of the rough shape (portions removed by machining) defined as an overlay by means of the machine allowance call-out.

6.0 BASIC SPECIFICATIONS

6.1 For the convenience of the reader, pertinent information on the profile tolerance system has been extracted from Specifications 1E0011, 1E0012, and 1E2122 and combined in this specification to provide a self teaching manual on the definition of casting and forging contour by profile tolerance.

6.2 Portions of definitions in 1E0011, 1E0012, and 1E2122 which are not applicable to castings or forgings have been deleted and those portions that are applicable have been expanded. It is not intended that 1E2600 override any of the basic specifications, however should a conflict in interpretation occur, Specifications 1E0011, 1E0012, and 1E2122 shall prevail.

6.3 Basic specifications 1E0011, 1E0012, 1E2122, 1E0100, and 1E0576 are related to 1E2600 as follows:

6.3.1 1E0011 drawing interpretation and tolerances specification covers interpretation and tolerances necessary to achieve uniform conformance to drawing requirements and applies to all (past and present) Caterpillar drawings/models.

6.3.2 1E0012 geometric tolerance interpretation specification covers the interpretation of geometric tolerances and is applicable to all caterpillar drawings/models as a part of 1E0011 (1E0012 is not shown on drawings/models). Included in 1E0012 are complete definitions for datums, datum targets, basic dimensions, and profile tolerance.

6.3.3 1E2122 surface texture specification defines the surface texture requirements and covers interpretation of the surface texture symbols indicating machining required, machining prohibited, and supplier responsibility.

6.3.4 1E0100 forger instructions specification includes general requirements such as markings, forging cleaning, and forging quality. 1E0100 also includes tolerances on mismatch, flash extension, and scale pits or dress out which are further restrictions of the profile tolerance and which must also be included within the profile tolerance zone along with variations in forging size and location relative to a datum system. Although profile tolerance is normally all inclusive, the flash extensions caused by split dies on forgings produced on an upsetter are an exception. Refer to Paragraph 13.3.

6.3.5 1E0576 foundry instructions specification includes general requirements for castings. While profile tolerance is normally all inclusive and all material must fit within the profile tolerance zone, the tolerance on 1E0576 foundry cleaning (fin and gate removal) is an exception. Refer to paragraph 12.1.

7.0 INTERPRETATION - PROFILE TOLERANCE SYSTEM

7.1 Interpretation of the profile tolerance system contained herein applies only to castings and forgings as illustrated in Figure 1.

7.2 Millimeter dimensions and tolerances are shown in figures and text to illustrate interpretation of geometric symbols and are not necessarily recommended values.

7.3 General

7.3.1 Interpretation - Symbols and Terms - See 1E0011, 1E0012, and 1E2122 for complete definitions.

7.3.1.1 Feature - A portion of a part such as a hole, boss, flat surface, diameter, etc.

7.3.1.2 Basic - Basic in this specification refers to the theoretically exact size or shape defined by basic dimensions.

7.3.1.3 Basic Dimensions - Dimensions used without tolerances to describe the theoretically exact size, shape, or location of a feature that forms the basis from which permissible variations are established by profile tolerances.

7.3.1.4 Datum - A datum is a point, axis, or plane used as the location from which form, position, or profile tolerances are checked. Datums are indicated by symbols.

7.3.1.5 ▷—[X] Or [–X–] Datum feature symbols indicate a datum.

7.3.1.6  Datum target symbol indicates a locating area (specified area on the rough casting or rough forging surface) used to establish a datum.

7.3.1.7 ⌖ All around symbol is used to indicate that profile of a surface tolerance applies all around the part.

7.3.1.8 ⌓ Profile of a surface symbol is used to indicate the tolerance on the basic size, shape, or location of a feature. Profile tolerance applies to surfaces in contrast to conventional tolerance that applies to dimensions. For example, on a round hole profile tolerance is applied to the radius whereas conventional tolerance is applied to the diameter.

7.3.1.9 ⌓ | 4 Feature control frame that reads profile of a surface with a tolerance of 4 mm divided equally about the basic or perfect profile.
Former callout (appears on drawings released before Jan 1988)
⌓ | ±2 Profile of a surface with a tolerance of ± 2 mm from the basic or perfect profile.

7.3.1.10 Profile Tolerance Callout Related to Datums

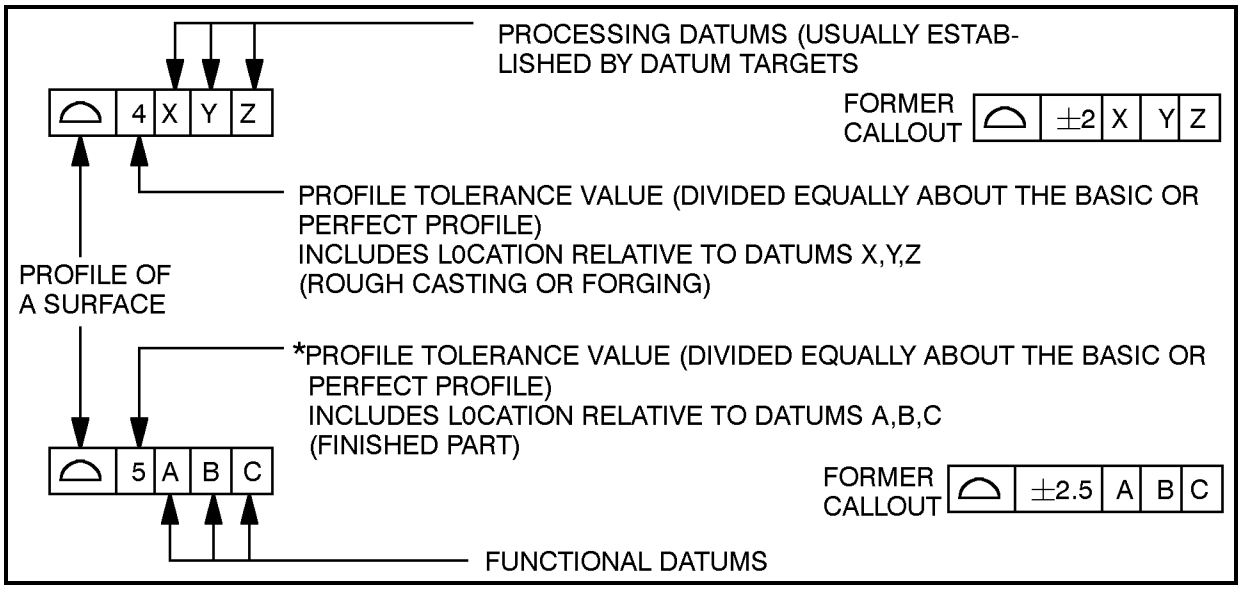


Figure 1

***Note:** Tolerance on cast or forged surfaces related to Datums A, B, C, is normally 1 mm larger than tolerance related to Datums X, Y, Z to allow for machining.

7.3.1.11 Phantom Line —·-·— **-** Used to show machine allowance on drafted surfaces, to define cast or forged outline that must be dimensioned, and to illustrate change in direction of cast or forged surfaces.

7.3.2 1E2122 - Surface Texture Symbols

7.3.2.1 ✓ **Machining Required Symbol -** A horizontal bar attached to the short leg of the surface texture symbol indicates that machining is required and that the casting or forging supplier must provide the machine allowance as specified in the **DIM AND TOL** block within the profile of a surface tolerance shown in relation to Datums X, Y, and Z.

7.3.2.2 5 **Machining Required with Basic Machine Allowance -** The 5 preceding the machining required symbol is the basic dimension for machine allowance. The profile tolerance related to Datums X, Y, and Z is applicable to the machining allowance.

7.3.2.3 ∞ **Machining Required with Infinite Machine Allowance -** The 8 preceding the machining required symbol indicates that the area is cast or forged solid. This symbol is usually applied to holes and indicates that the hole is to be machined (drilled) from solid material. This symbol replaces for holes.

7.3.2.4 **Machining Prohibited Symbol -** A circle shown in the vee of the basic surface texture symbol indicates that machining is prohibited and the surface must be produced by the casting or forging process without removal of material.

CAST
FRG

7.3.2.5 **Supplier Responsibility Symbol - CAST** or **FRG** shown on a horizontal bar attached to the long leg of the surface texture symbol indicates that the surface must be produced by the casting or forging supplier using a foundry or forging operation such as cast, core, coin, forge, trim, or punch or an optional machining operation such as drill, mill, or grind.

CAST

7.3.2.6 Indicates that the casting supplier must machine the surface or feature such as a **SCRIBE AND MILL**.

7.3.2.7 **Surface Texture Symbol -** Has been used only with small holes on some casting and forging drawings. The absence of a machining required symbol indicates the area will be cast or forged solid and the hole will be machined (drilled) from solid material. is discontinued and is replaced by ∞

7.4 Interpretation - Profile Tolerance - Feature

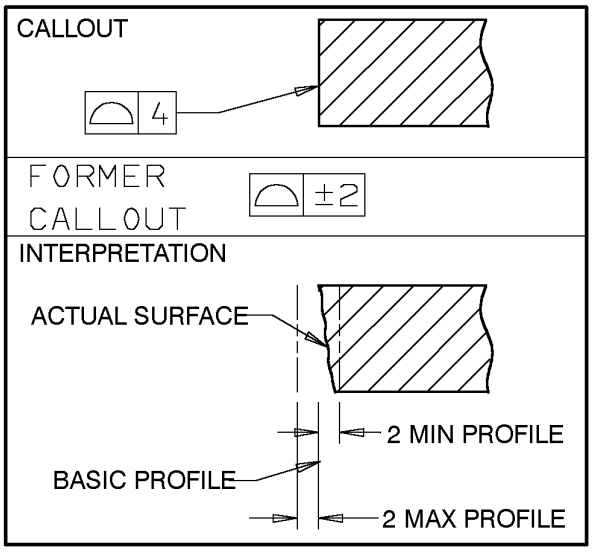


Figure 2 – Profile Tol - Feature

The callout ⌓|4 attached to the basic or perfect profile of the rough casting or rough forging may vary anywhere between the two boundaries identified with min profile and max profile. This callout only controls size of the feature and cannot control location without adding a relationship to a datum system.

7.5 Casting and forging requirements priority for interpreting drawings shall be as follows:

7.5.1 First Priority - Dimensions, tolerances, machine allowances, draft, and casting and forging outline illustrated by phantom lines on the body of the drawing.

7.5.2 Second Priority - Machine allowance, fillets, radii, draft, wall thickness, and profile tolerances shown in the **DIM AND TOL** block.

7.5.3 Third Priority - Second general machine allowance and profile tolerance shown in an extension to the regular **DIM AND TOL** block. Refer to Figure 39.

7.5.4 Fourth Priority - Dimensions and tolerances shown in specifications 1E0100 and 1E0576.

7.5.5 Fifth Priority - 1E2600.

8.0 LOCATING SYSTEM

The locating system is one of the key features of the profile tolerance system and is used to orient or position the casting or forging in a datum reference system consisting of three mutually perpendicular datum planes (normally X, Y, Z) which are established by datum targets located on the surface of the rough casting or forging. Datum planes (normally A, B, C), used to inspect finished parts, are established by functional surfaces or features (normally machined) indicating positioning or fit up of the part at assembly.

Processing (X, Y, Z) and functional (A, B, C) datum systems shown on the drawing provide a common reference for all measurements used by the patternmaker or diesinker, foundry or forge shop inspection, receiving inspection, planning, tool design, machine shop, and final inspection with everyone using the same basis for measurement.

1E2600 is intended primarily to interpret processing requirements for suppliers and Caterpillar manufacturing and inspection personnel and uses X, Y, Z datums in most illustrations. The same rules and logic may be used to interpret requirements associated with the application of the A, B, and C functional datums.

8.1 Datum Targets

Parts shall be made such that parting lines, gates, trademarks, part number, cast date, etc., do not fall within an area designated as a datum target area. Datum targets are used to establish datum planes or axes on rough castings or rough forgings. The commonly used datum targets (locators) are shown in Figure 3. For complete information on datum targets refer to specification 1E0012.

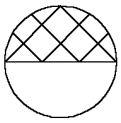
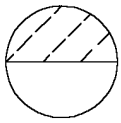
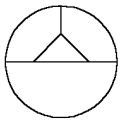
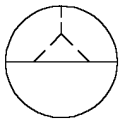
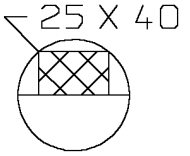
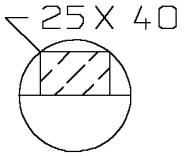
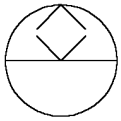
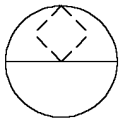
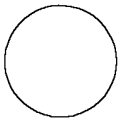
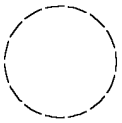
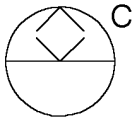
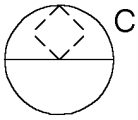
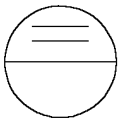
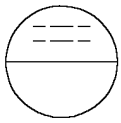
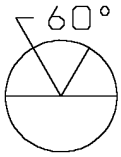
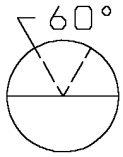
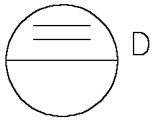
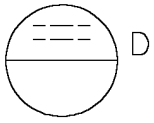
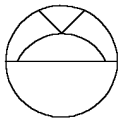
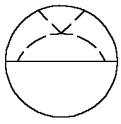
TARGET TYPE	VISIBLE TARGET	HIDDEN TARGET	TARGET TYPE	VISIBLE TARGET	HIDDEN TARGET
Ø12 – 15			EQUALIZER 3 WAY (3 JAW CHUCK)		
RECTANGULAR			CENTERING VEES		
SUPPLIMENTAL			CENTERING VEES (SEPARATED-CONTACT-POINTS)		
EQUALIZER 2 WAY			VEE		
EQUALIZER 2 WAY (SEPARATED CONTACT POINTS)			ROCKER		

Figure 3 - Datum Targets

The diagram in the upper part of the datum target symbol shows the type of target being applied. A letter (X, Y, or Z) in the lower left part indicates which datum the target establishes. A number (1, 2, or 3) in the lower right part identifies the first, second, or third datum target used to establish a datum. For Example  or  .

8.2 Relationship - Datum Targets to Datums

8.2.1 Three mutually perpendicular datum planes (X, Y, and Z), established by datum targets on the rough cast or rough forged surfaces, form the theoretical planes from which measurements are taken and tolerances applied. (See Figures 4, 5, and 6).

8.2.2 The datum targets represent the locators in the tooling fixture used to position and hold the rough castings and rough forgings for machining.

8.2.3 The block type structure shown in Figure 4 rep-resents a rough casting or forging which is positioned on 3 datum targets (X1, X2, X3) to level the rough casting or forging or make it square with the measuring system. Datum plane X, shown in phantom, is the theoretical plane, established by contacting 3 datum targets, which is used as a reference plane or zero location for all dimensions that are perpendicular to Datum X.

8.2.4 Datum X, which is the first datum plane, positions or supports the rough casting or forging in an up and down direction but the rough casting or forging is still free to move sideways and slide back and forth.

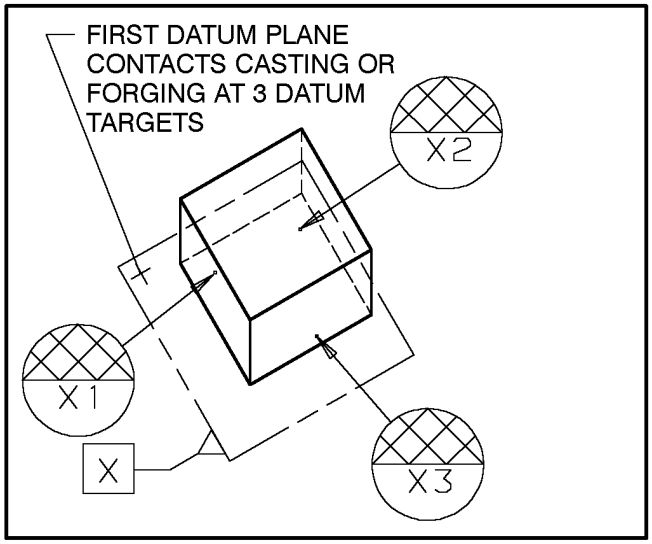


Figure 4 – Datum X – 3 Datum Targets

8.2.5 With Datum X established as in Figure 4, the rough casting or forging is positioned against 2 datum targets that serve to fix or lock it in a horizontal direction to prevent lateral movement.

8.2.6 Datum plane Y, shown in phantom in Figure 5 as the second datum plane, is the theoretical plane, perpendicular to Datum X, which is established by contacting the two datum targets and is used as a reference plane or zero location for all dimensions that are perpendicular to Datum Y.

8.2.7 Datum X positions or supports the rough casting or forging and prevents up and down movement and Datum Y prevents lateral movement but the rough casting or forging block is still free to slide back and forth.

8.2.8 With Datum Plane X established in Figure 4 and Datum Plane Y established in Figure 5, the rough casting or forging is positioned against the final datum target which serves to lock or fix the rough casting or forging and prevents a back and forth movement.

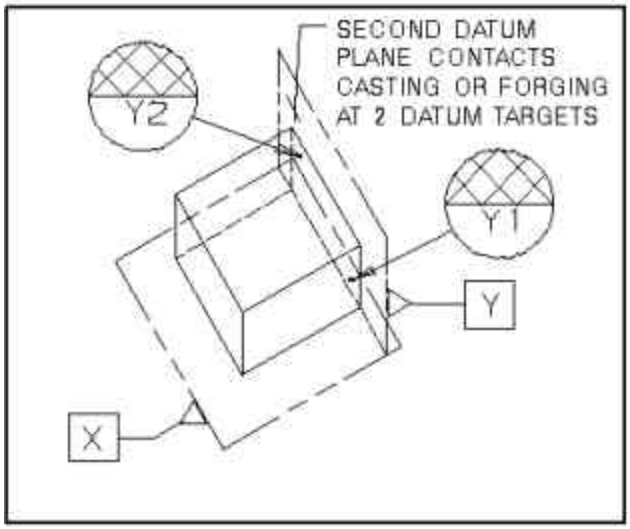


Figure 5 – Datum Y – 2 Datum Targets

8.2.9 Datum Plane Z, shown in phantom in Figure 6 as the third datum plane, is the theoretical plane, perpendicular to datum planes X and Y, which is established by contacting the final datum target and is used as a reference plane or zero location for all dimensions perpendicular to Datum Z.

8.2.10 With Datums X, Y, and Z established to locate or position the rough casting or forging against the 6 datum targets, it may be secured (clamped down) for machining or inspection.

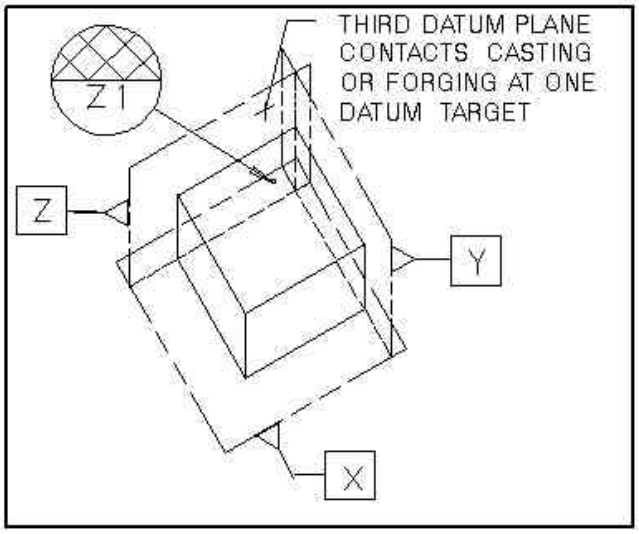


Figure 6 – Datum Z – 1 Datum Target

8.3 Relationship - Datum Targets and Datums to Profile Tolerance

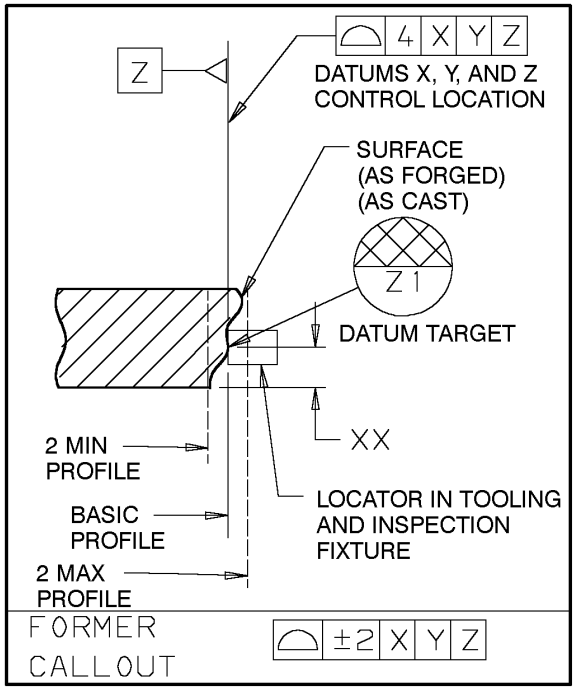


Figure 7 – Datum Targets – Datums – Profile Tol

8.3.1 Figure 7 illustrates the relationship of datum targets and datums to profile tolerance. All datum targets are dimensioned to provide the full tolerance intended for castings and forgings and also to provide more consistent location for machining. Note that Datum Target Z1, a 12 to 15 mm diameter area, is located XX mm from the bottom of the cross section and is used to establish Datum Plane Z.

8.3.2 It is important to understand that there is no location tolerance on Datum Target Z1 because it is located on Datum Plane Z which is the theoretical plane from which all dimensions perpendicular to Datum Plane Z are measured.

8.3.3 ⌓ | 4 | X | Y | Z | Indicates that the profile tolerance relative to Datums X, Y, and Z is 4 mm and controls both size and location. The profile tolerance forms two boundaries identified in Figure 7 as min profile and max profile, which lie 2 mm on either side of the basic or theoretically perfect profile. The as cast or as forged surface may fall anywhere within the 4 mm tolerance zone as shown by the dashed lines in Figure 7 with the exception of the Z1 datum target area which coincides with the basic or perfect profile.

8.3.4 The concept of zero tolerance on datum targets is explained by visualizing datum target Z1 contacting a locator in a tooling fixture. Regardless of the magnitude of the casting or forging variations, the location of the spot contacting the locator always lies on the theoretical reference plane which is the zero location or beginning point for measurement of all dimensions that are perpendicular to Datum Z.

9.0 BASIC DIMENSIONS

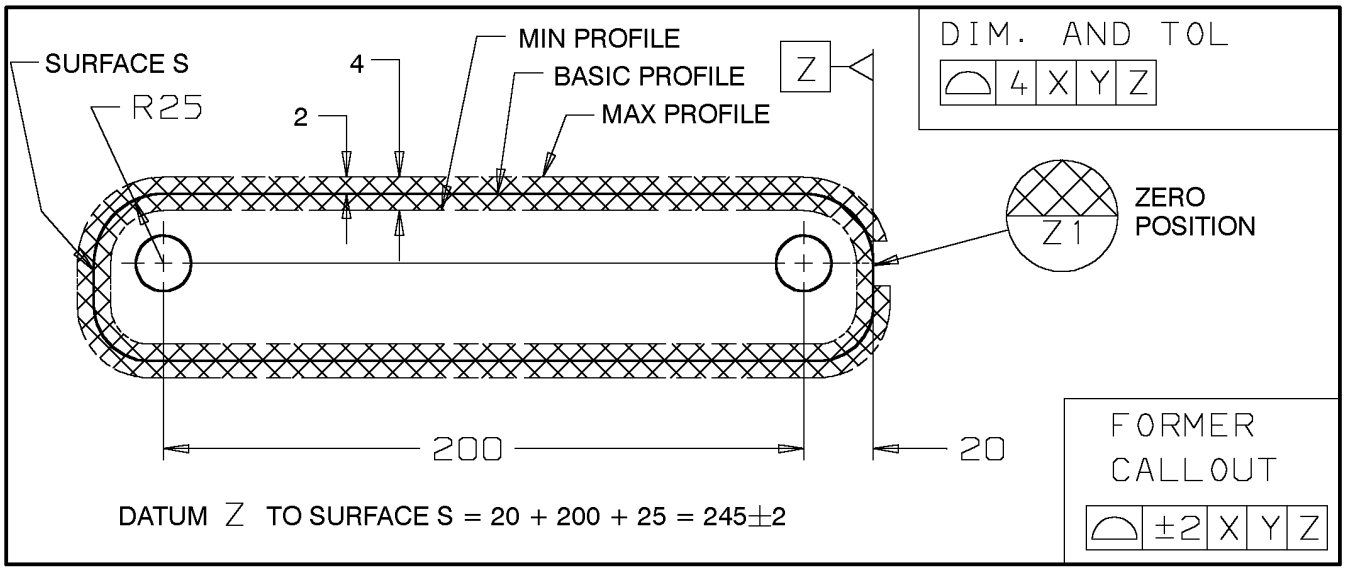


Figure 8 - Basic Dimensions

9.1 Figure 8 illustrates the use of basic dimensions (see Paragraph 7.3.1.3) to determine the basic profile, or perfect location, of a surface in relation to a datum.

9.2 Datum target Z1 is located on the centerline of the holes and establishes datum plane Z which is the reference plane or zero location for all dimensions that are perpendicular to Datum Z, or all horizontal dimensions in this figure.

9.3 To establish the basic profile of Surface S, begin at datum plane Z, the reference plane, and add the 20 mm dimension to the centerline of the hole, the 200 mm dimension between hole centers, and the 25 mm radius defining the upper left contour, which gives a total of 245 mm from datum plane Z to Surface S. Basic dimensions are used without tolerances to establish basic profile or perfect form. For simplicity, it is assumed that datum planes X and Y have been established in a similar manner in another view of the drawing.

9.4 The 4 mm profile tolerance related to datum planes X, Y, and Z shown in the **DIM AND TOL** block controls both size and location and is applied to the configuration or outline of the casting or forging shown on the drawing, which represents the basic profile or perfect form. The maximum profile is established by constructing a boundary 2 mm outside of the basic profile, while the minimum profile is formed by constructing a boundary 2 mm inside the basic profile.

9.5 The minimum and maximum profiles are shown in Figure 8 as dashed or broken lines with the resulting tolerance zones shown as crosshatched areas 4 mm wide and equally distributed 2 mm on both sides of the basic profile.

9.6 The actual casting or forging surface may vary anywhere between the two theoretical boundaries representing the 4 mm tolerance zone except for one specific area. The area of the casting or forging which contacts datum target Z1 lies on datum plane Z which is precisely in the center of the profile tolerance zone. In relation to datum target Z1 the overall length is 245 ± 2 mm, while overall length measured at other positions could be 245 ± 4 mm.

9.7 When a dimension on the drawing that must be used to establish the basic casting or forging profile is toleranced (XX.X ± X.X) rather than basic, the dimension is used as a basic dimension and the tolerance is not applied in determining the basic profile.

10.0 MACHINE ALLOWANCE

10.1 General

10.1.1 The machining required symbol $\sqrt{}$ identifies surfaces of castings and forgings that must be machined and is the key to machine allowance, direction of parting, and draft (machine allowance is from a design required surface not from <> dimensions). The parting line is normally parallel to surfaces identified by a machining required symbol without machine allowance $\sqrt{}$, whereas draft is normally permitted on surfaces identified by a machining required symbol with a machine allowance $5\sqrt{}$ that is larger than the machine allowance specified in the DIM AND TOL block.

10.1.2 Machine allowance tolerances on castings and forgings provide a minimum value that will assure complete removal of the decarburized layer and maintain cutter depth below imbedded scale or burned-on surface sand particles.

10.1.3 Special machine allowance either larger or smaller than provided in the DIM AND TOL block may appear on the body of the drawing to control processing such as to provide additional machine stock that permits another part to be made from the rough casting or forging, to protect a surface during carburizing by providing additional material that will be removed by machining before hardening, to reduce machining stock in holes to permit broaching, or to require trimming on forgings.

10.2 Surface Without Draft - Machine Allowance

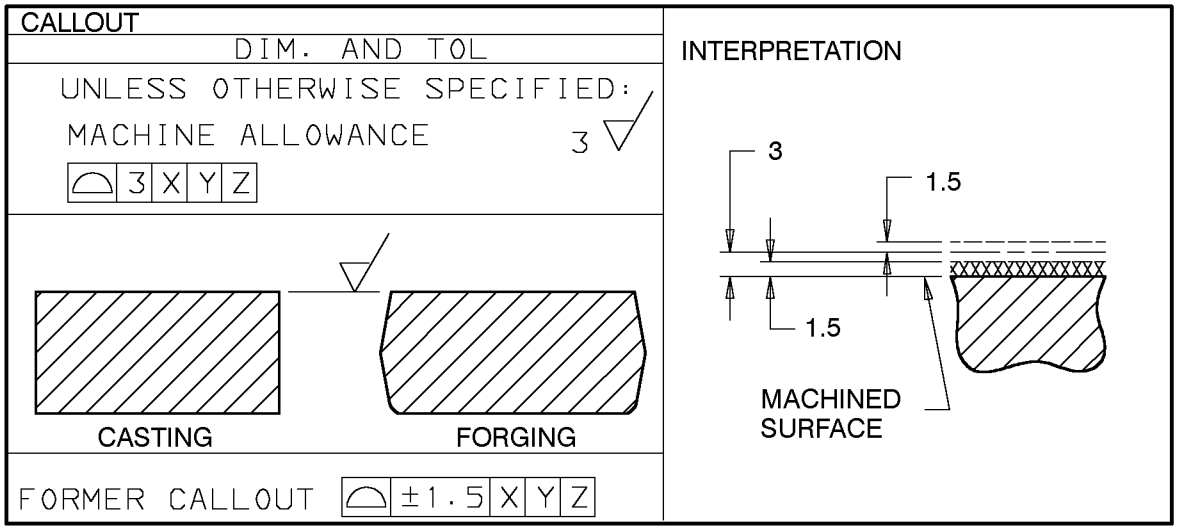


Figure 9 - Machine Allowance - Surface Without Draft

10.2.1 In Figure 9, the drawing callout for both the casting and the forging shows a single machining required symbol ▽ on an extension line to indicate that the surface must be machined. The absence of a machine allowance value to the left of the machining required symbol indicates that the dimension for machine allowance 3 ▽ , shown in the **DIM AND TOL** block applies.

10.2.2 Castings and forgings that use the profile tolerance system are essentially pictures of the finished part and surfaces identified by a machining required symbol are shown as machined. In the interpretation in Figure 9, basic machine allowance is illustrated pictorially by a phantom line representing the rough surface of the basic profile of the casting or forging which lies 3 mm from the machined surface and is subject to the general 3 mm profile tolerance shown related to Datums X, Y, and Z in the **DIM AND TOL** block. The machine allowance varies from 1.5 to 4.5 mm.

10.2.3 Figure 10 is a graphic illustration of basic casting or forging profile where surfaces without draft are machined to make the finished part.

10.2.4 Using the data in the **DIM AND TOL** block in Figure 9, machine allowance varies from 1.5 to 4.5 mm representing minimum and maximum casting or forging size shown as dashed lines.

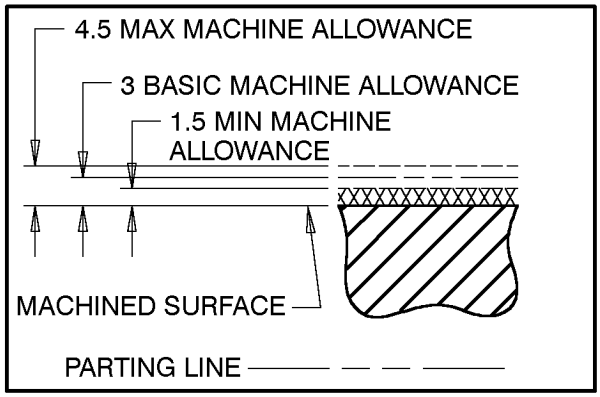


Figure 10 – Machine Allowance - Surface Without Draft

10.3 Drafted Surface - Machine Allowance

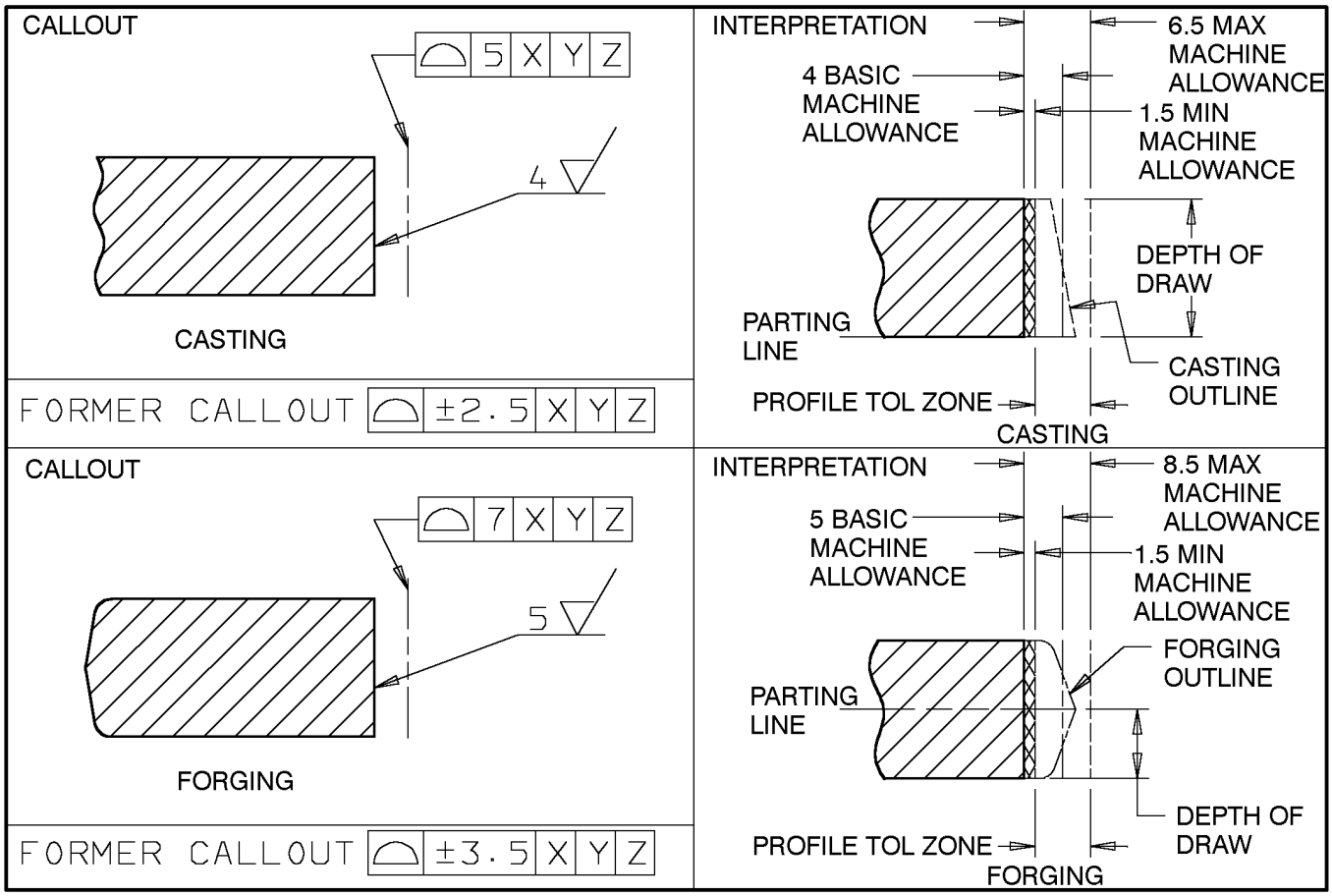


Figure 11 - Machine Allowance - Drafted Surface (Parted Perpendicular to Machined Surface)

10.3.1 In Figure 11, the callout used to indicate machine allowance for drafted surfaces on both castings and forgings consists of three elements as follows: (1) A machine allowance, (larger than the general machine allowance value in the **DIM AND TOL** block) shown preceding the machining required symbol, (2) A phantom line shown parallel to the machined surface, and (3) A special profile tolerance, (larger than the tolerance in the **DIM AND TOL** block) attached to the phantom line. Whenever these three conditions are shown in combination on the drawing, the surface so identified may be drafted with the draft included in the profile tolerance zone. The callouts show a 5 mm machine allowance with a 7 mm profile tolerance for the forging and a 4 mm machine allowance with a 5 mm profile tolerance for the casting to illustrate that forgings normally require more draft than castings.

10.3.2 In the interpretation the $5\sqrt{}$ machining required callout means that the phantom line representing basic machine allowance is located 5 mm from the machined surface. The profile tolerance attached to the phantom line means that a 7 mm tolerance zone is centered on the phantom line (5 mm basic machine allowance) and provides a minimum machine allowance of 1.5 mm and a maximum machine allowance of 8.5 mm. Draft is illustrated pictorially by a dashed line and shows a typical casting or forging outline when the parting line is perpendicular to the machined surface. Depth of draw for draft extends from the parting line to the cast or forged face.

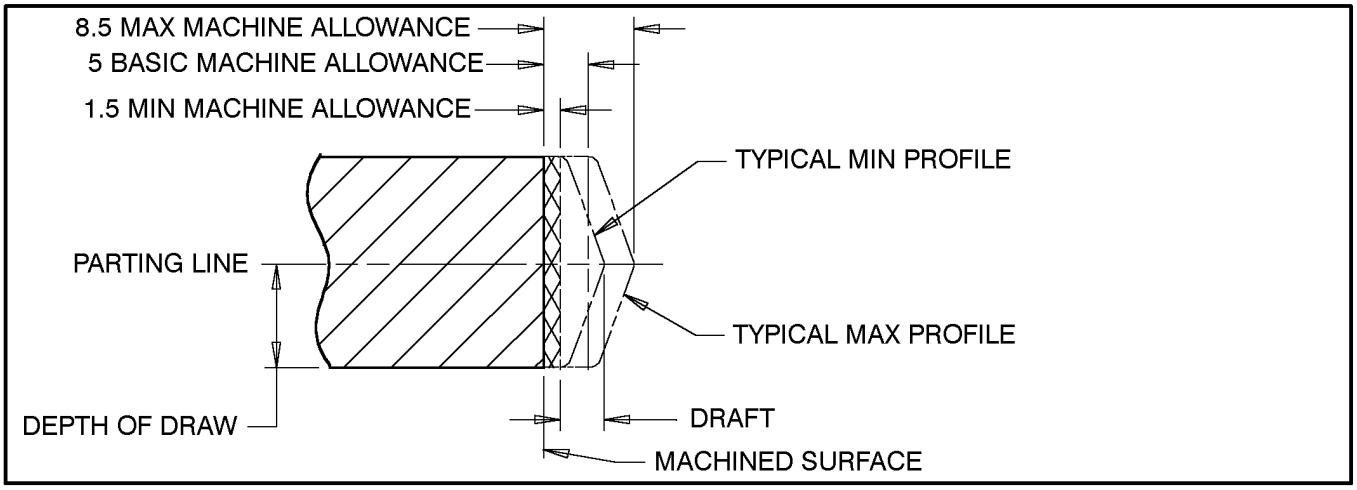


Figure 12 - Graphic Illustration of Machine Allowance on a Drafted Surface

10.3.3 Figure 12 graphically illustrates typical forging shapes for the forging in Figure 11. Machine allowance on the drafted surface is 5 ± 3.5 mm. Machine allowance and profile tolerance for a drafted surface consists of three elements as follows:

10.3.3.1 The 1.5 mm minimum machine allowance.

10.3.3.2 The draft.

10.3.3.3 The profile tolerance zone.

10.3.4 Machine allowance may vary from 1.5 to 8.5 mm and the contour of the rough casting or forging may fall anywhere between the minimum and maximum profile boundaries. A specific amount of draft is not required in the machine allowance zone and all draft may be removed by the supplier if required to bring the part contour within the profile boundaries.

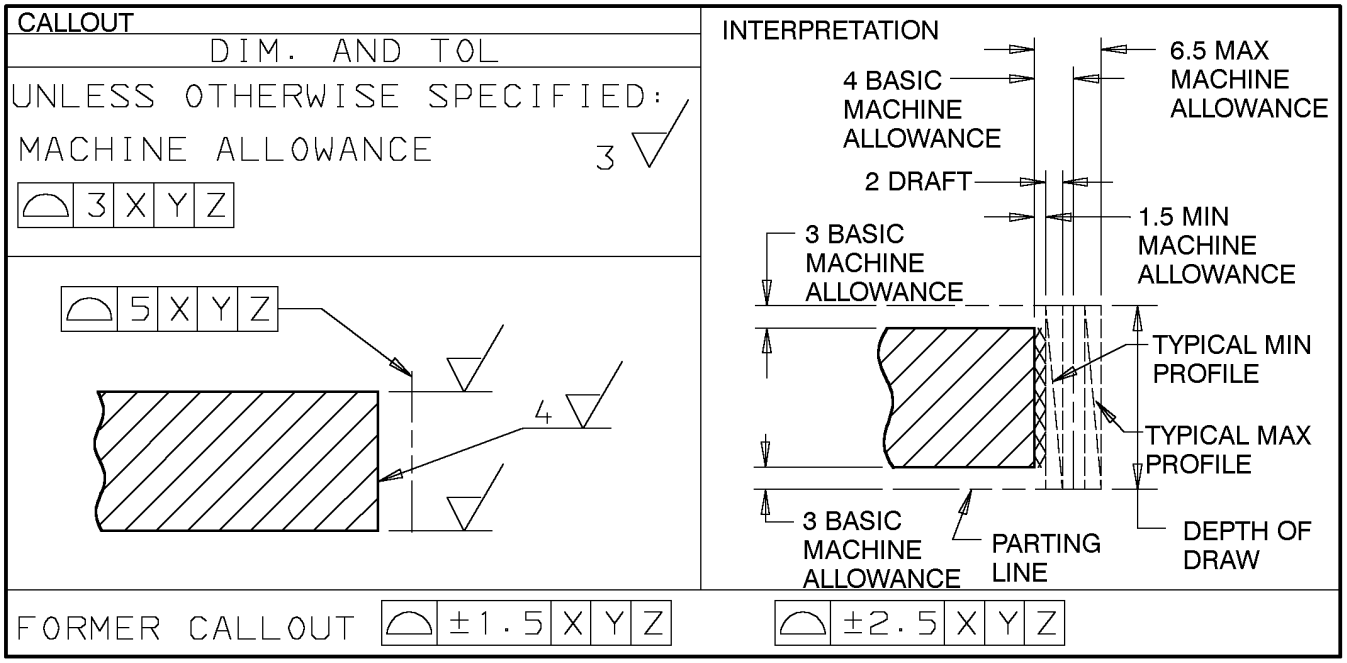


Figure 13 - Machine Allowance - Drafted Surface - Edge Parted - Casting

10.3.5 Figure 13 illustrates machine allowance where the drafted surface is parted on an edge. This type of parting is commonly used for castings.

10.3.6 The callout shows a 4 mm basic machine allowance with the machining required symbol for the vertical surface and a 5 mm profile tolerance attached to the phantom line. The machining required symbols for the horizontal surfaces (shown without values) apply the 3 mm basic machine allowance and the 3 mm general profile tolerance shown in the **DIM AND TOL** block.

10.3.7 The interpretation graphically illustrates the 1.5 mm minimum machine allowance, the 4 mm basic machine allowance, and the 6.5 mm maximum machine allowance that applies to the vertical machined surface. Dashed lines illustrate typical minimum and maximum profiles.

10.3.8 The parting line and basic machine allowance are shown by a phantom line 3 mm below the lower horizontal surface. Another phantom line is shown 3 mm above the upper horizontal line to show basic machine allowance. Depth of draw for draft is shown to extend from the parting line to the upper phantom line and consists of the vertical machined dimension plus the two 3 mm basic machine allowances. As shown previously, each 3 mm basic machine allowance is subject to the 3 mm general profile tolerance.

10.4 Mean Size - Mean size is a calculated size used by Caterpillar work standards to determine machining requirements. Mean size is calculated differently for non-drafted and drafted surfaces as follows:

10.4.1 Non-Drafted Surface - Mean size is obtained by subtracting half the general profile tolerance (.5 x 3 mm) from the maximum machine allowance (4.5 mm). Mean size is 3 mm and coincides with the basic machine allowance as illustrated by the horizontal surfaces in Figure 14.

10.4.2 Drafted Surface - Mean size is obtained by subtracting half the general profile tolerance (.5 x 3 mm) (note that this is not the profile tolerance used to establish maximum machine allowance) from the maximum machine allowance (6.5 mm). Mean size is 5 mm and does not coincide with the basic machine allowance as illustrated by the vertical surface in Figure 14.

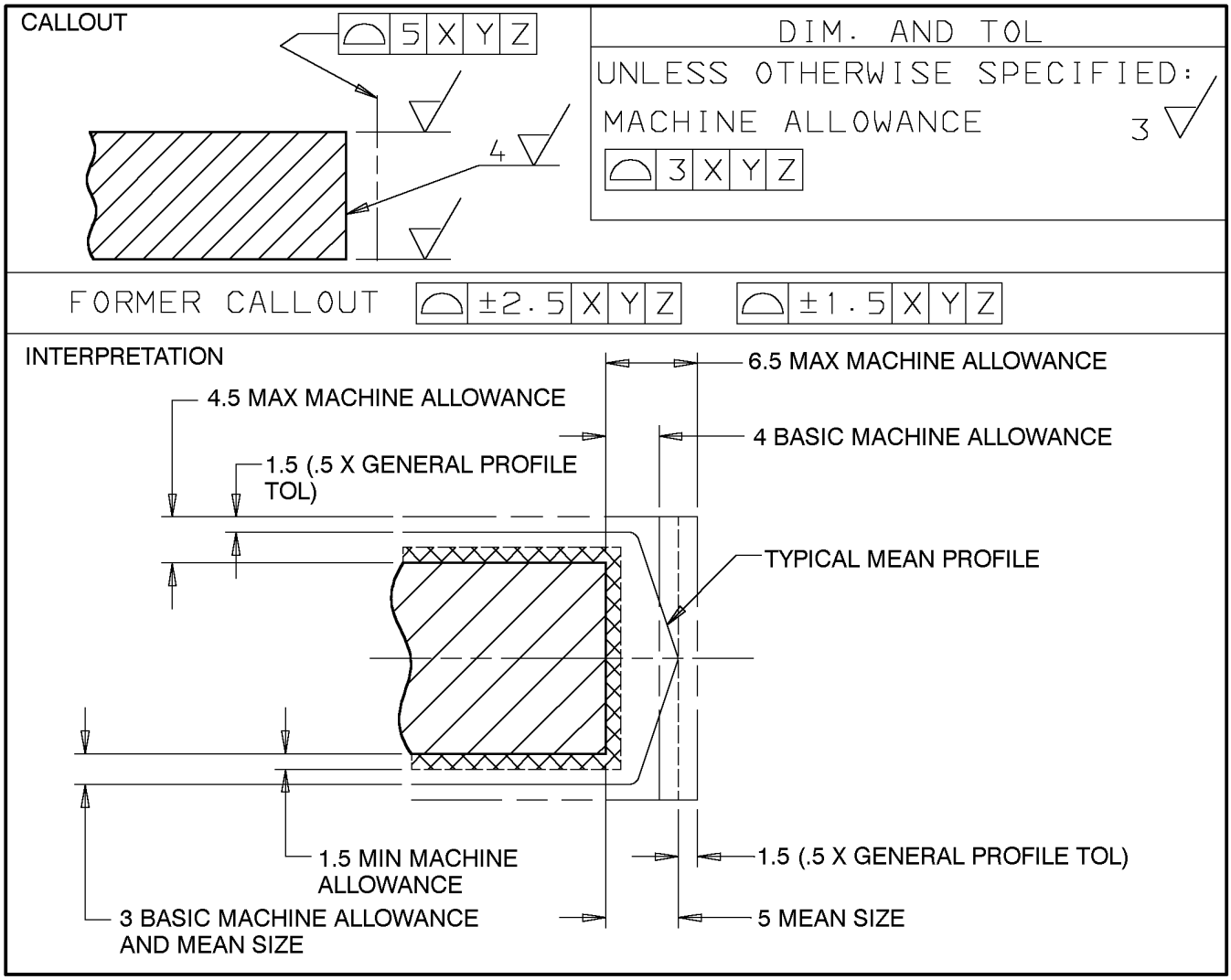


Figure 14 - Mean Size Calculation

10.5 Machine Allowance Requiring Close or Straight Trim

10.5.1 Basic machine allowance values on parted drafted surfaces of forgings as illustrated in Figures 11 and 12, generally provide for normal forging trim. Where a machine allowance value that cannot encompass the draft is specified to reduce machining, the forging will require a close trim or in some cases a straight trim as illustrated in Figure 15.

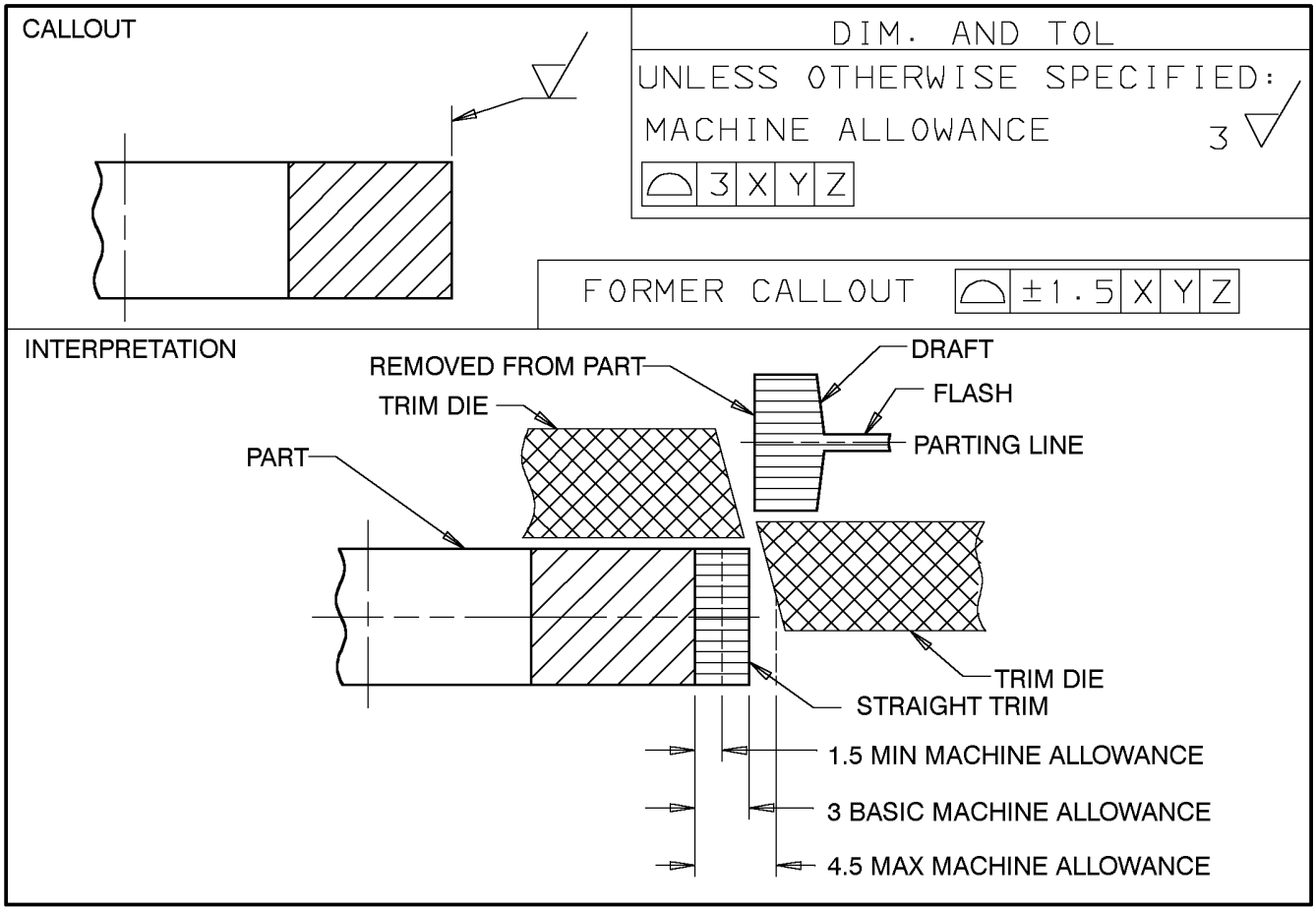


Figure 15 - Machine Allowance - Forging Draft Removed - Parted Surface

10.5.2 Figure 15 is a graphic illustration of machine allowance where all draft must be removed from the parted surface to minimize machining.

10.5.3 Note that no machine allowance value is shown with the machining required symbol attached to the extension line representing the machined surface, this indicates that the 3 mm basic machine allowance shown in the forging **DIM AND TOL** block applies. The 3 mm basic machine allowance does not include draft; therefore the forging supplier must trim the part so it is within the max material condition.

10.5.4 The 3 mm basic machine allowance is subject to the 3 mm general profile tolerance shown in the **DIM AND TOL** block that provides a machine allowance of 1.5 mm minimum to 4.5 mm maximum.

10.5.5 In this illustration, trimmer dies have been used to straight trim the forging and remove all draft and flash extension.

10.5.6 The type of forging trim required is controlled by the machine allowance callout for a surface that is machined on the completed part. Refer to Figures 29, 30, 31, and 32 for the type of forging trim required when the trimmed surface remains on the completed part.

10.6 Draft - Non Parted Surface - Figure 16 illustrates machine allowance on a drafted surface where the machined surface does not cross the parting line, such as the surface which is machined on the end of a hub to accommodate a bearing.

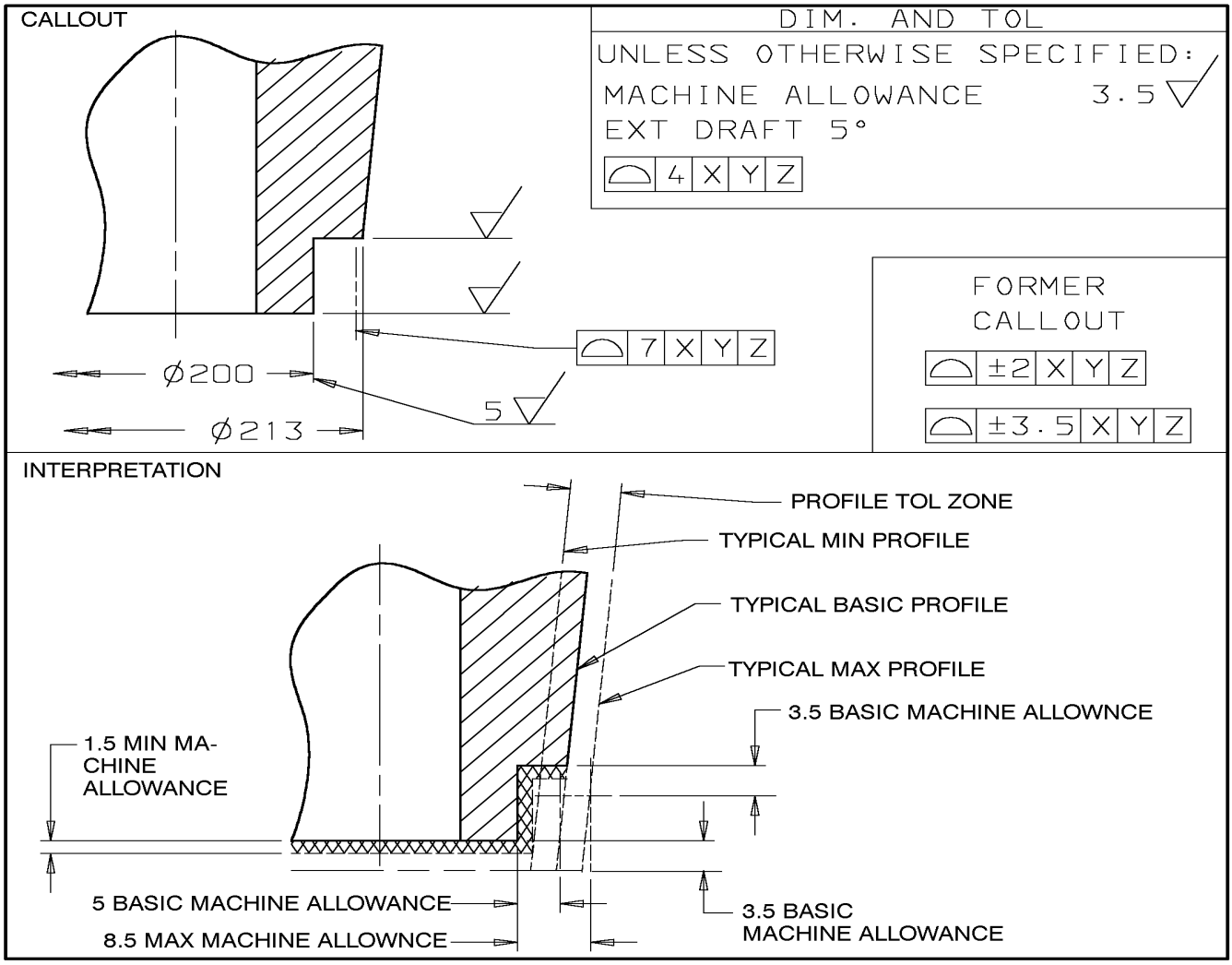


Figure 16 - Machine Allowance/Draft - Non-Parted Surface

10.6.1 The callout shows 3 machined surfaces, indicated by the machining required symbols. A 5 mm basic machine allowance is shown with the machining required symbol for the vertical surface and a 7 mm profile tolerance is attached to the phantom line. The machining required symbols, shown without machine allowance, provide the 3.5 mm basic machine allowance shown in the **DIM AND TOL** block along with the 4 mm general profile tolerance.

10.6.2 Graphic interpretation shows the 1.5 mm minimum machine allowance, the 5 mm basic machine allowance, and the 8.5 mm maximum machine allowance that applies to the vertical machined surface.

10.6.3 Basic casting or forging size is shown centered between the two boundaries formed by the 4 mm general profile tolerance in the **DIM AND TOL** block. Typical minimum, typical basic, and typical maximum profiles are shown as dashed lines. The vertical phantom line is shown as 5 mm basic machine allowance while the horizontal phantom line is shown as 3.5 mm basic machine allowance. The profile of the casting or forging may vary anywhere within the 1.5 to 8.5 mm machine allowance zone represented by the two vertical broken lines, the profile will normally resemble the dashed lines (typical profiles) in this type of application.

10.7 Machine Allowance – Corners - Figure 17 illustrates machine allowance on a corner where the machined surface has a chamfer or radius.

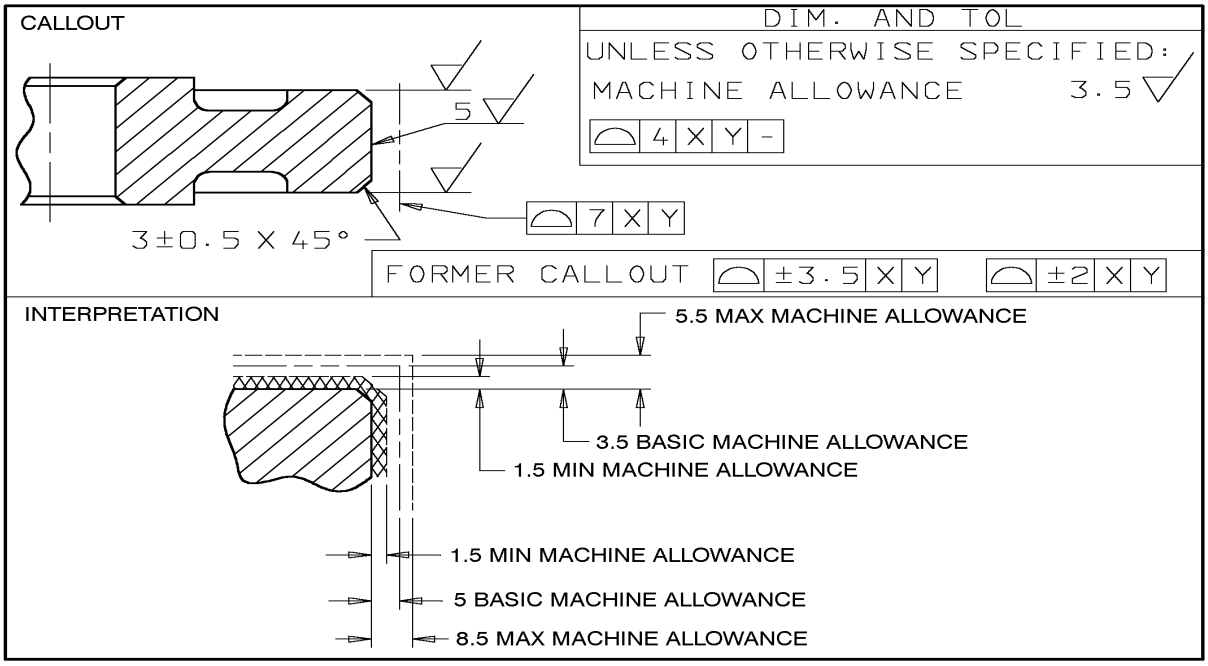


Figure 17 - Machine Allowance/Corner

10.7.1 The callout shows 3 machined surfaces indicated by the machining required symbols. The 5 mm basic machine allowance is called out on the vertical surface and a 7 mm profile tolerance is attached to the phantom line. The machining required symbols, shown without values, provide the 3.5 mm basic machine allowance shown in the **DIM AND TOL** block along with the 4 mm general profile tolerance.

10.7.2 The interpretation shows machine allowance in the corner area formed by two intersecting phantom lines (two intersecting machine allowance zones). The 1.5 mm minimum machine allowance is shown as a crosshatched zone adjacent to the machined surfaces. The casting or forging surface may fall anywhere within the machine allowance zone identified by minimum and maximum machine allowance.

Caution: Foundry suppliers should not interpret the square corner of the two intersecting phantom lines as a requirement to provide a square corner on the casting. Square corners on gray iron castings normally cause chill, which is prohibited in machine allowance zones by the Caterpillar specifications covering cast gray iron materials.

10.8 Machine Allowance – Fillets - Figure 18 illustrates machine allowance in a fillet area formed by two intersecting machine allowance zones in applications such as the mounting flange and machined diameter of a hub.

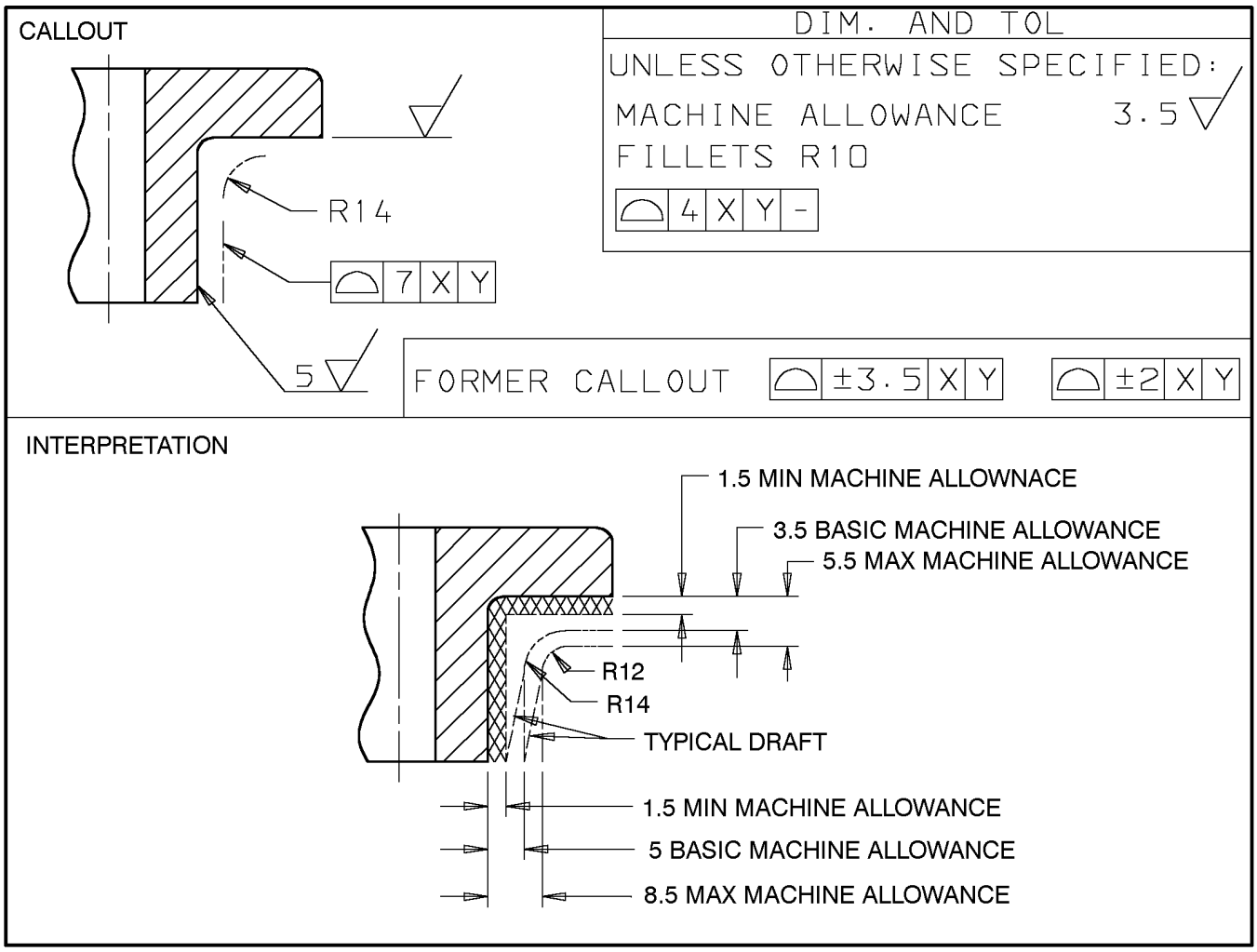


Figure 18 - Machine Allowance - Drafted Surface - Fillet

10.8.1 The callout shows two machined surfaces indicated by machining required symbols. A 5 mm basic machine allowance is called out on the vertical surface and a 7 mm profile tolerance is attached to the phantom line. The machining required symbol, shown without a machine allowance, provides the 3.5 mm basic machine allowance shown in the **DIM AND TOL** block along with the 4 mm general profile tolerance. Also a 10 mm fillet radius is shown in the **DIM AND TOL** block to cover fillets that remain on the finished part and a 14 mm fillet radius is shown tangent to the phantom line in the machine allowance zone.

10.8.2 The interpretation graphically illustrates minimum and maximum machine allowance where two intersecting machine allowance zones form a fillet. The 14 mm fillet radius is interpreted as being tangent to both the 3.5 mm and the 5 mm basic machine allowances represented graphically by the two phantom lines.

10.8.3 The maximum casting or forging size, illustrated by a dashed line, is 5.5 mm from the horizontal machined surface and 8.5 mm from the vertical machined surface. The fillet radius associated with maximum casting or forging size is determined by subtracting the smaller of the two profile tolerances from the basic fillet radius. In this example, the 14 mm basic fillet radius minus half the 4 mm general profile tolerance gives a 12 mm fillet radius on the zone defining maximum casting or forging size.

10.8.4 The casting or forging surface may lie anywhere within the zone between minimum and maximum machine allowance, which permits a sharp inside corner and zero vertical draft, however, the supplier needs some fillet radius and some vertical draft to make the casting or forging. Typical draft associated with minimum and maximum casting or forging size is indicated by the dashed lines.

10.9 Machine Allowance - Phantom Line Override

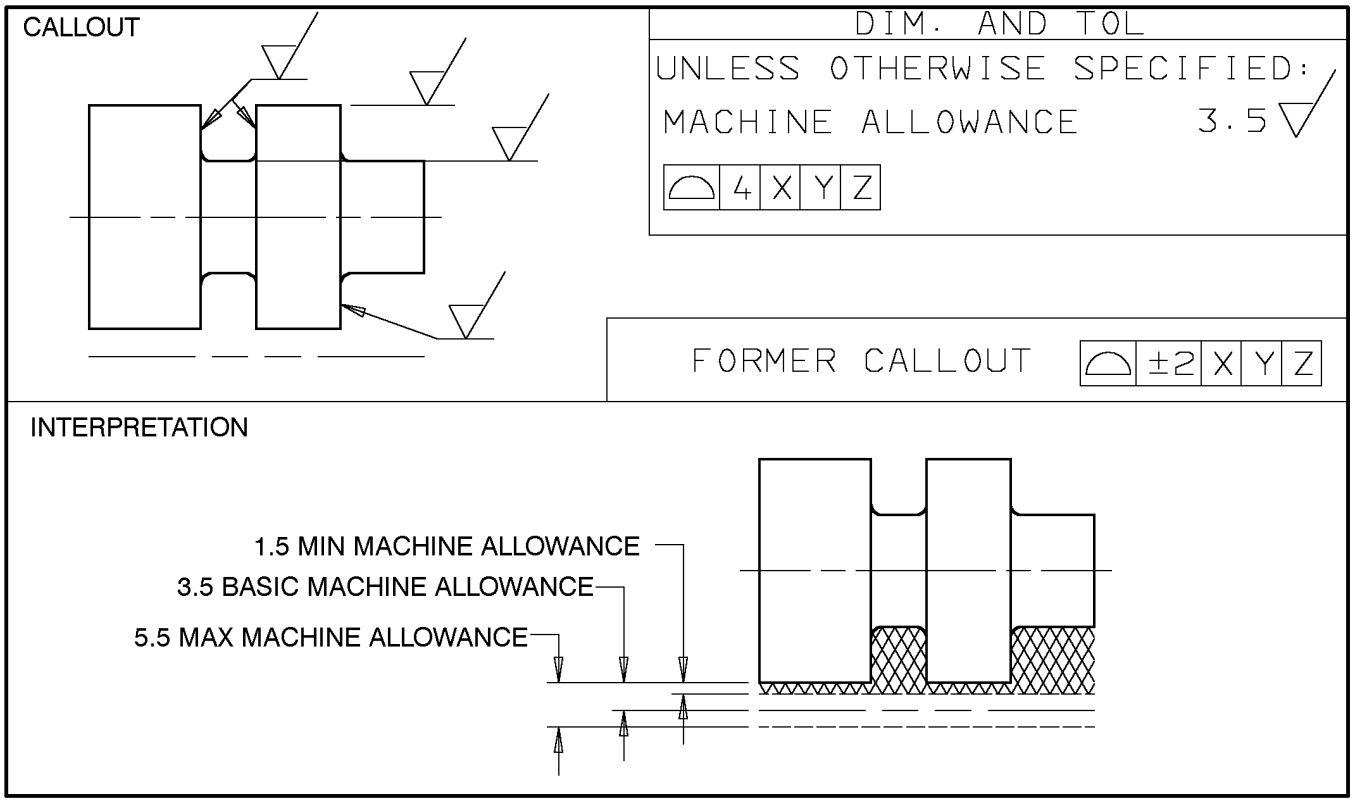


Figure 19 - Machine Allowance - Phantom Line Override

10.9.1 The callout in Figure 19 shows machining required symbols for the outside diameter, sides and bottom of groove, shoulder and reduced end diameter that provide the 3.5 mm basic machine allowance and 4 mm general profile tolerance. A phantom line is also shown to override the normal machine allowance interpretation for determining casting or forging contour.

10.9.2 The interpretation graphically illustrates that the casting or forging outlines do not follow the contour of the groove and reduced diameter when the phantom line is used.

10.10 Machine Allowance - Dimensioned Phantom Line

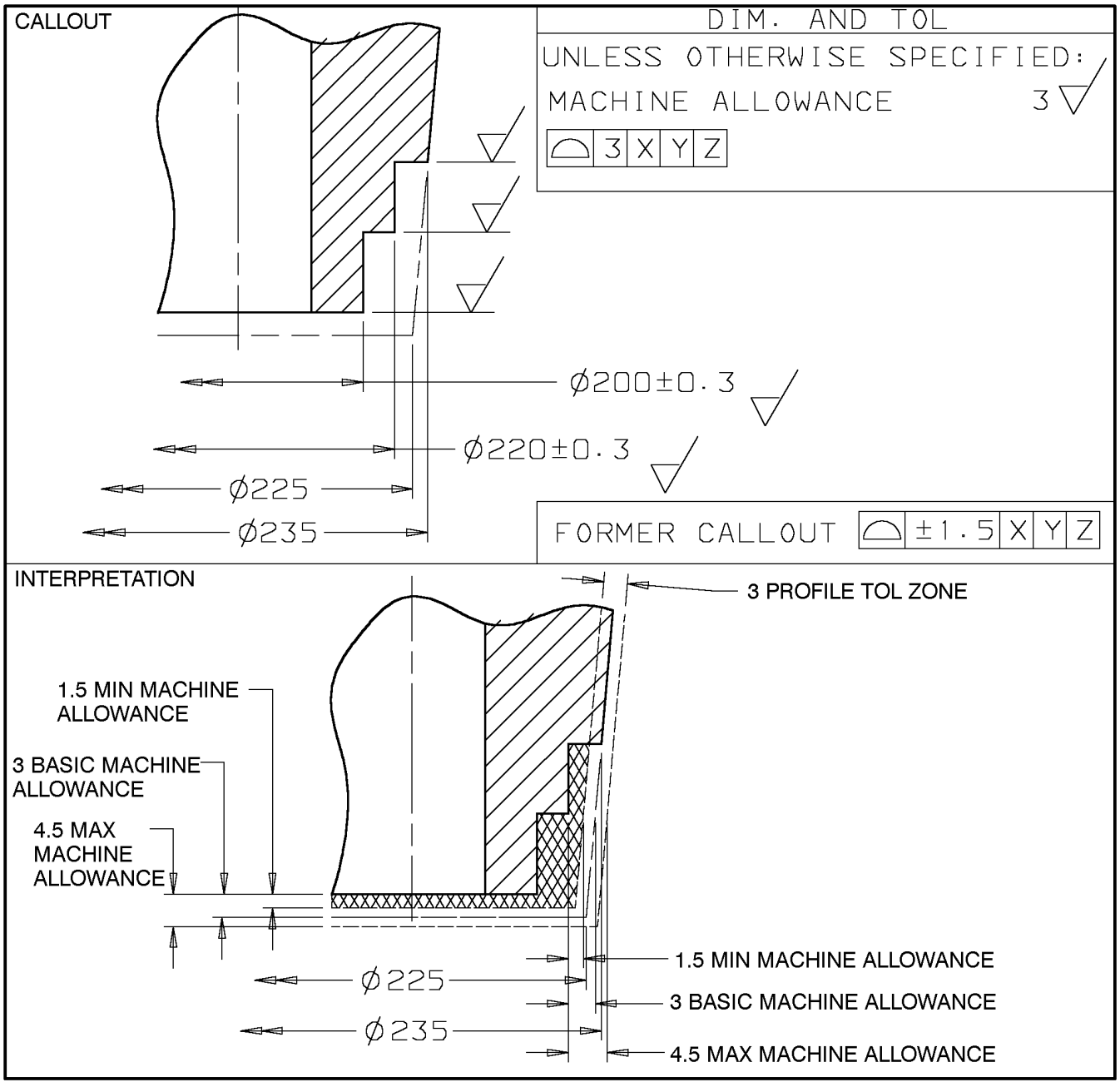


Figure 20 - Machine Allowance - Dimensioned Phantom Line

10.10.1 The callout, representing two machined diameters on a part such as a hub, uses a dimensioned phantom line to define machine allowance. A phantom line is shown extending from the unmachined drafted surface at the upper right of the illustration downward to the dimensioned intersection with the horizontal phantom line. The **DIM AND TOL** block shows 3 mm basic machine allowance and 3 mm general profile tolerance. The two phantom lines are shown intersecting at 225 mm diameter while the smallest machined diameter is 200 ±0.3 mm and the diameter of the uppermost horizontal machined surface is 235 mm.

10.10.2 The interpretation graphically illustrates minimum and maximum machine allowance associated with a dimensioned phantom line. The point of origin of the drafted phantom line is shown at the intersection with the horizontal phantom line, which represents the 3 mm basic machine allowance. The 225 mm diameter indicates that 3 mm basic machine allowance and draft were applied to the 220 mm machined diameter (3 mm basic machine allowance applied to each radius of the machined diameter and basic draft applied for the distance between the face of the 220 machined diameter and the horizontal phantom line representing 3 mm basic machine allowance). The 3 mm general profile tolerance in the **DIM AND TOL** block applies to all cast or forged surfaces as interpreted in the illustration.

10.10.3 The 225 mm and 235 mm diameters relate to cast or forged surfaces and are covered by the 3 mm general profile tolerance in the **DIM AND TOL** block which is applied to each surface, or in this case, the radius rather than the diameter.

11.0 DRAFT AND PARTING LINE

Draft angle is a basic dimension that applies wherever draft exists on the completed part to define the basic profile.

11.1 Draft Illustrated Pictorially - Draft greater than 2 degree external or 3 degree internal is shown pictorially on the drawing only on unmachined surfaces that remain on the finished part. Forging draft is always greater than the above amounts and is shown pictorially which provides positive identification for location of the parting line, origin of draft, and matching draft except where forgings are machined all over. Refer to Article 11.4 for interpretation of draft and parting line when the forging is machined all over.

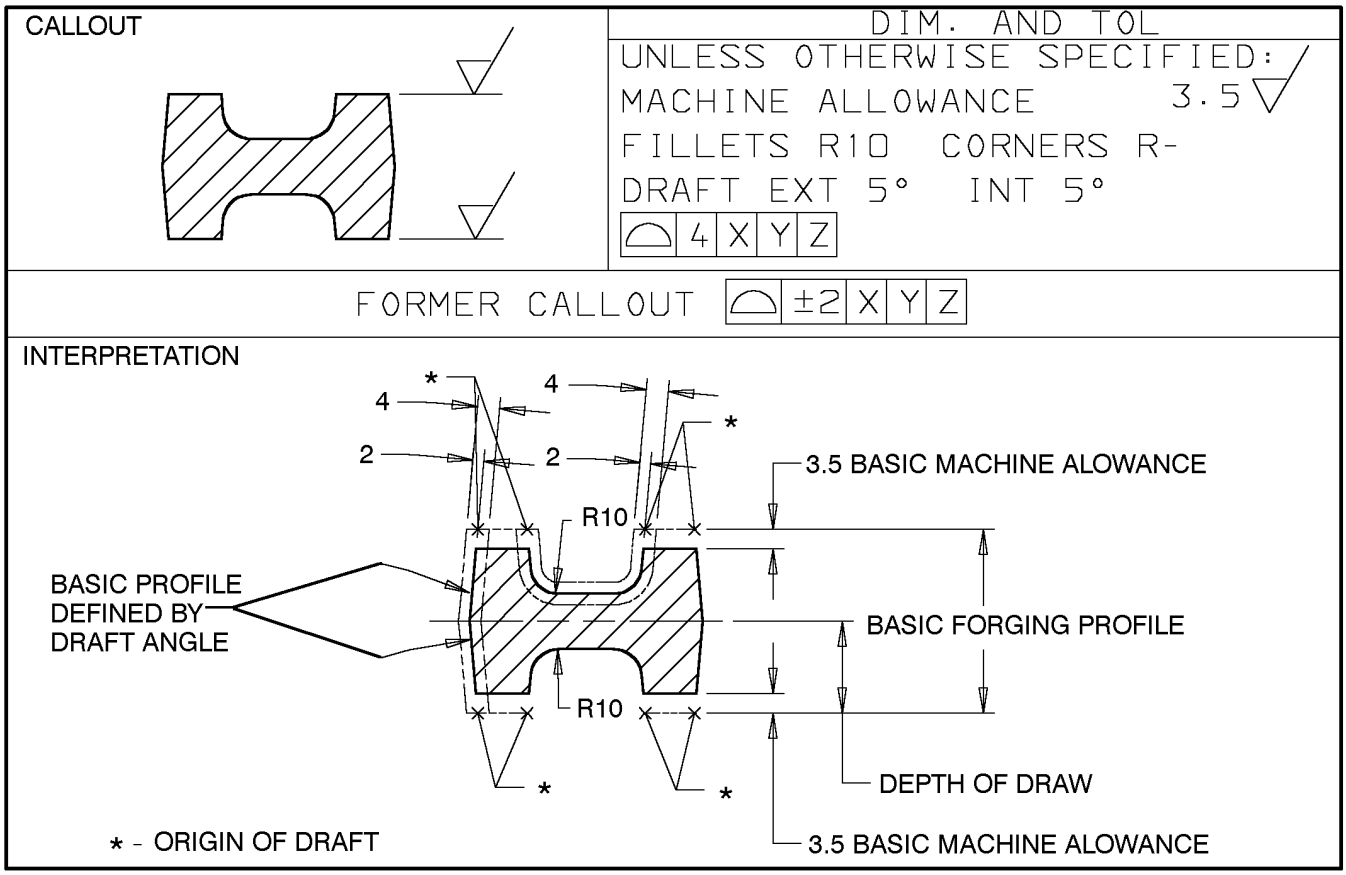


Figure 21 - 5º Forging Draft Shown Pictorially

11.2 Figure 21 illustrates draft and parting line on a forging where 5 degree draft remains on the finished part.

11.3 The callout shows four machined surfaces identified by two machining required symbols, 5 degree external and 5 degree internal draft shown pictorially with the values listed in the **DIM AND TOL** block along with 3.5 mm basic machine allowance and a 4 mm general profile tolerance.

11.4 The callouts are interpreted graphically to illustrate parting line, non-drafted forged surfaces, and origins of draft. A Horizontal parting line is shown at the center of the forging. Horizontal phantom lines are shown representing 3.5 mm basic machine allowance and basic forging profile. Depth of draw required for draft is shown extending from parting line to phantom line. Origins of draft are identified by asterisks (*) at the intersections of extended draft lines with the phantom lines representing the forged surfaces.

11.5 Draft, fillet radii, and all forged surfaces including the 3.5 mm basic machine allowance are subject to the 4 mm general profile tolerance in the **DIM AND TOL** block. The 4 mm profile tolerance zones are shown graphically by broken lines equally spaced on both sides of the basic profile to illustrate tolerance on external draft, internal draft, and fillet radii.

11.6 Draft Not Shown Pictorially - Normal draft for castings does not exceed 2 degrees external and 3 degrees internal and is not normally shown pictorially on the drawing. This makes it necessary to rely upon rules and definitions to interpret casting draft and parting. The same rules are also used to interpret draft and parting on forgings that are machined all over.

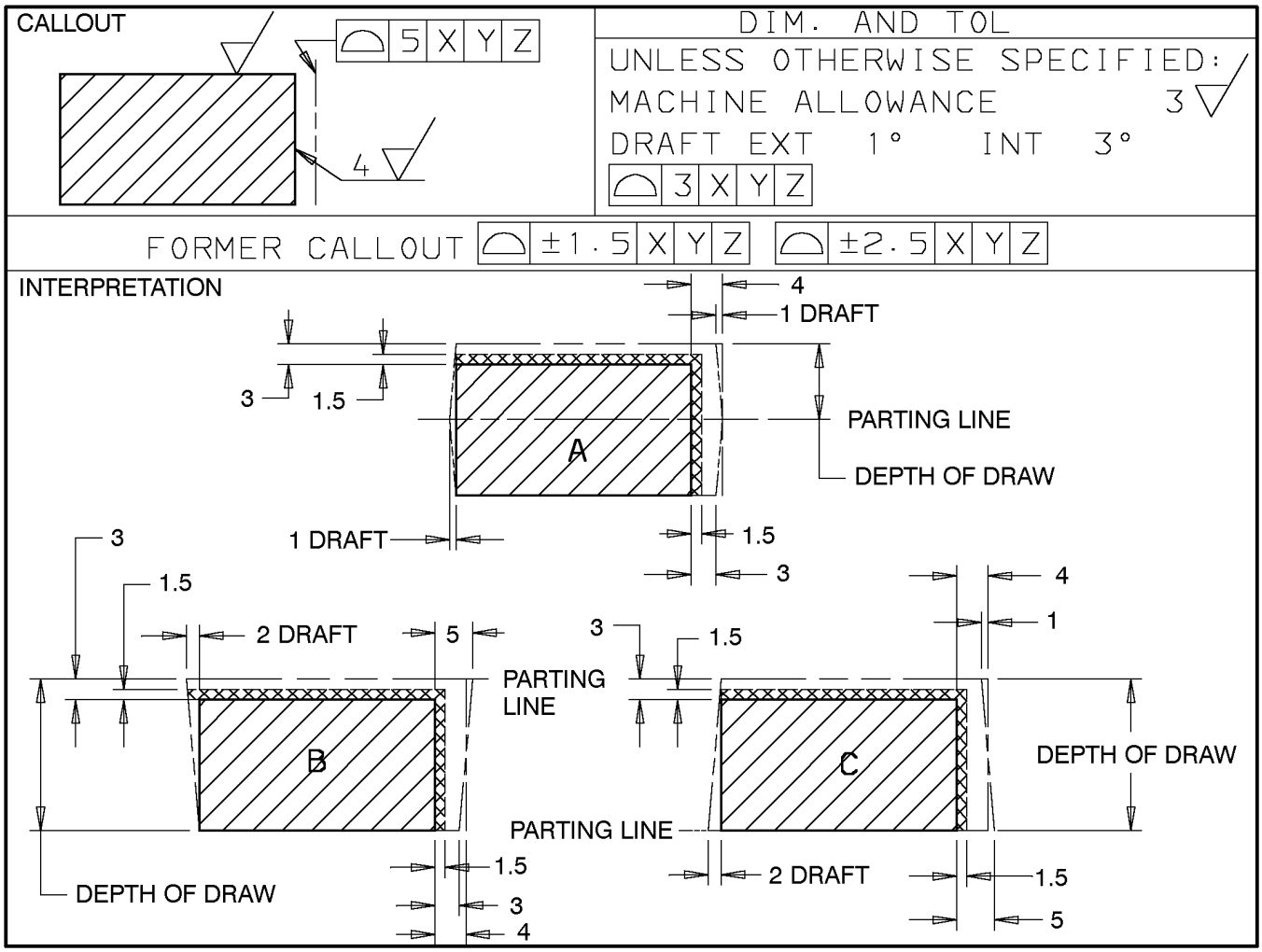


Figure 22 - Interpretation - Casting Draft and Direction of Parting - Not Shown Pictorially

11.7 Figure 22 illustrates direction of parting, draft, and basic profile of a casting where draft is not shown pictorially on the drawing and direction of parting is indicated by special machine allowance callouts.

11.8 The callout shows a rectangular casting cross section with two surfaces machined, indicated by the machining required symbols. A 4 mm basic machine allowance is shown on one of the vertical surfaces and a 5 mm profile tolerance is attached to the phantom line. The **DIM AND TOL** block shows 3 mm basic machine allowance, 1 degree external and 3 degree internal draft, and 3 mm general profile tolerance.

11.9 The callouts are interpreted graphically to illustrate draft, parting line, basic casting profile, and depth of draw. The machining required symbol shown without a value indicates that the horizontal machined surface is not drafted and direction of the parting line will be parallel to this surface. The 4 mm basic machine allowance indicates that a given amount of draft is included within the machine allowance on the vertical machined surface. The supplier has the option to part the casting on either edge shown in illustrations B and C, or at the center of the casting as shown in illustration A, or anywhere in between; but a supplier cannot change the existing direction of parting without first negotiating a change.

11.10 In all three illustrations, a horizontal phantom line is used to show the 3 mm basic machine allowance, which also represents the basic casting profile. The 1.5 mm minimum machine allowance is illustrated by a crosshatched zone adjacent to the machined surfaces. Draft is shown by diagonal phantom lines, and a vertical phantom line is used to represent the 4 mm basic machine allowance, which provides a point of reference for the various types of draft. A draft of 2 mm is shown in Illustrations B and C with the only difference being points of origin. Illustration A, which is parted in the center and has only 1 mm of draft, is preferred because machine allowance can be reduced.

11.11 Normal machine allowance for drafted surfaces of castings is large enough to permit full draft required for edge parting as shown in illustrations B and C because it is not usually known in advance where the casting supplier will locate the parting line to optimize production. The normal machine allowance selected for drafted surfaces of forgings is only large enough to provide for half draft associated with parting line located at or near the center as shown in illustration A. Interpretation of draft and parting line of forgings that are machined all over should agree with Figure 22, Illustration A.

11.12 Positive Draft - Draft which is not shown pictorially is considered to be positive or plus (added to the surface) unless otherwise specified on the drawing. Positive draft adds to external dimensions and subtracts from internal dimensions.

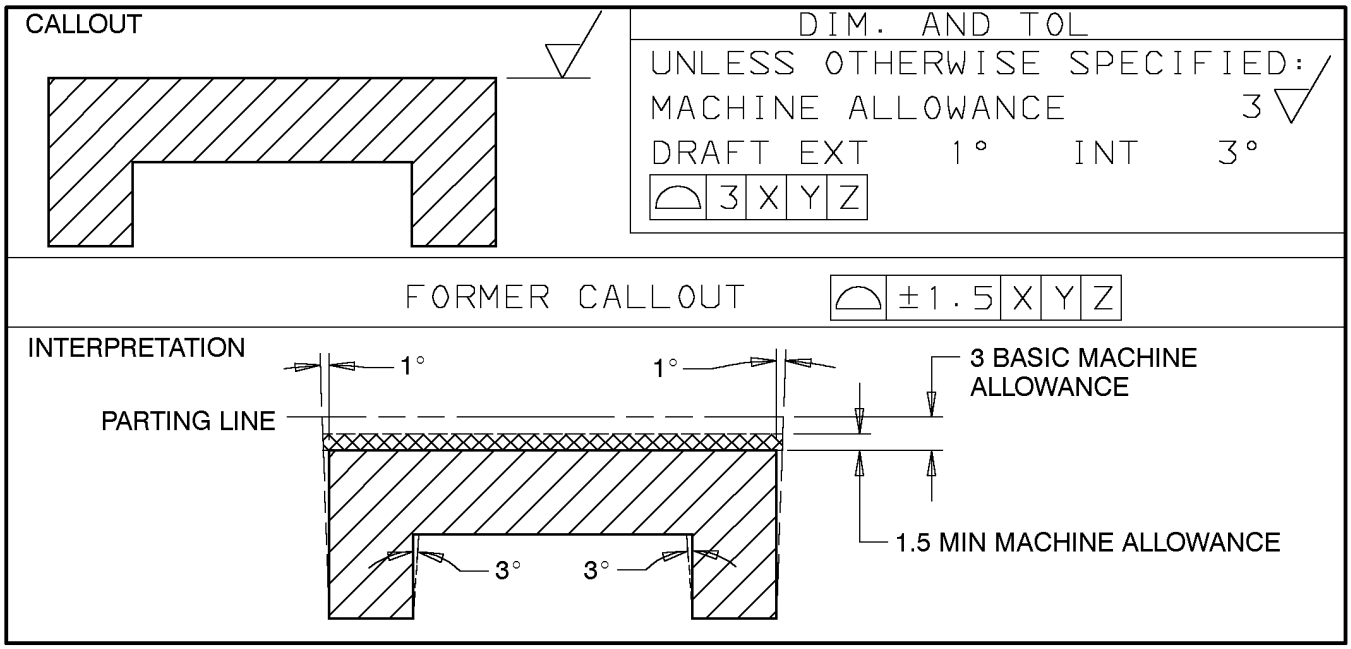


Figure 23 - Positive Draft - Not Shown Pictorially

11.13 Figure 23 illustrates positive or plus draft. The callout shows a machined surface, indicated by the machining required symbol, and 3 mm basic machine allowance, 1 degree external and 3 degree internal draft, and 3 mm general profile tolerance in the **DIM AND TOL** block.

11.14 The interpretation illustrates the 1 degree external and 3 degree internal draft, which has been added to the basic nondrafted view. A horizontal phantom line representing 3 mm basic machine allowance is also used to illustrate the parting line and a crosshatched zone is used to show the 1.5 mm minimum machine allowance.

11.15 1 degree external and 3 degree internal draft is normally shown in the **DIM AND TOL** block for sand castings as shown in Figure 23. The internal draft shown in the **DIM AND TOL** block is interpreted to mean internal surface formed by the pattern and pockets in external surfaces (such as shown with 3 degree draft in Figure 23). Draft on internal surfaces formed by cores is intended to be 1 degree.

11.16 Parting Location - Origin of Draft - Figure 24 illustrates direction of parting, draft, and basic profile of a casting when the draft remains on the completed part but is not shown pictorially nor indicated by a special machine allowance.

11.16.1 The callout illustrates a part 200 mm long that is machined on the top and bottom surfaces. Three methods of parting with corresponding draft are shown in the interpretation. Location of the parting line is at the discretion of the supplier unless otherwise specified by additional dimensioning. Interpretation A shows the parting line at the phantom line that represents basic machine allowance for the top surface. Origin of draft is shown at the phantom line that represents basic machine allowance for the bottom surface. Length at the bottom surface is 200 mm and at the top surface is 200 mm plus draft. Depth of draw is shown as the dimension between top and bottom cast surfaces.

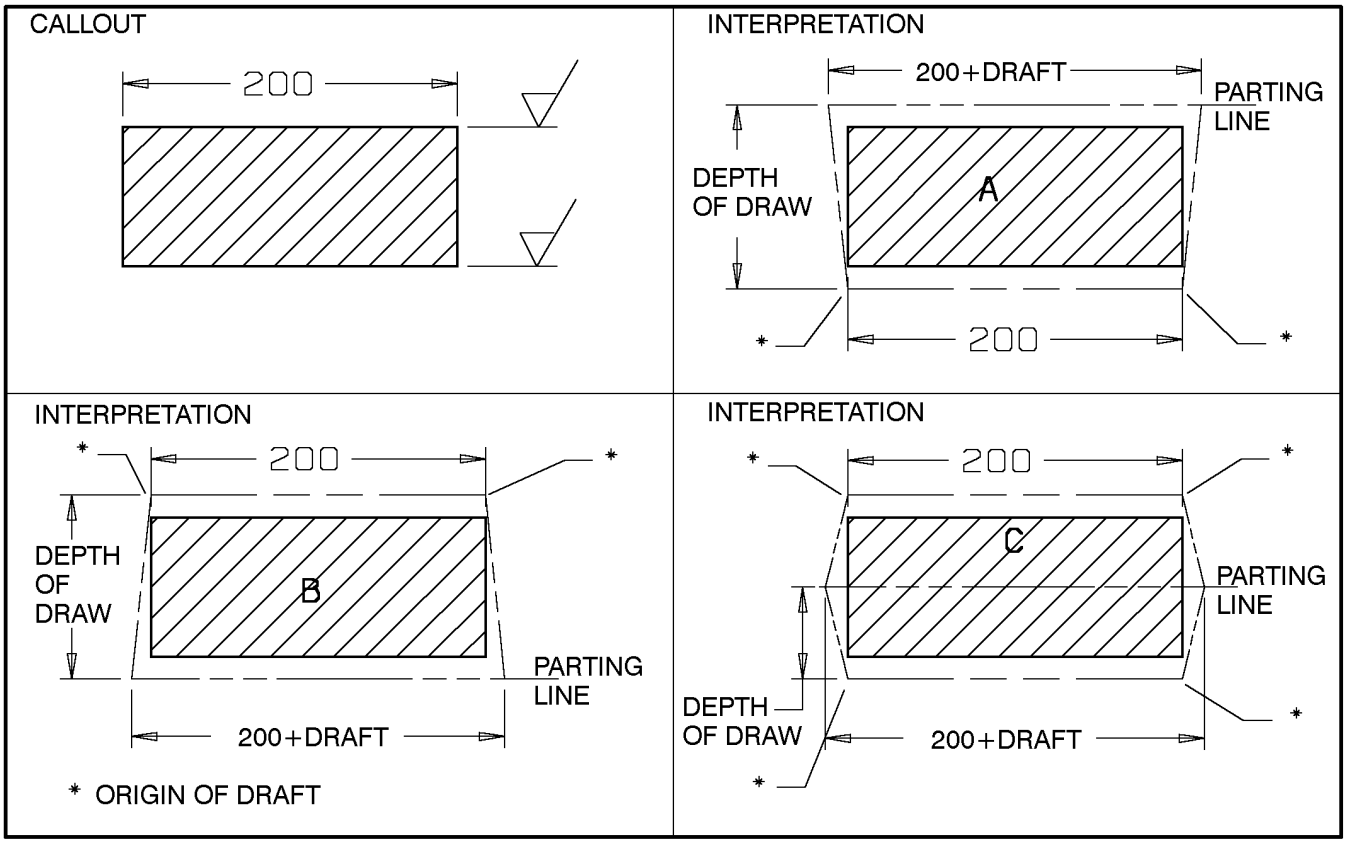


Figure 24 - Parting Location - Origin of Draft Not Shown Pictorially

11.16.2 Interpretation B shows the parting line at the phantom line that represents basic machine allowance for the bottom surface. Origin of draft is shown at the phantom line that represents basic machine allowance for the top surface. Length at the top surface is 200 mm and at the bottom surface 200 mm plus draft.

11.16.3 Interpretation C shows the parting line at the center. Origin of draft is shown at the phantom lines representing basic machine allowance for both top and bottom surfaces. Length at both top and bottom surfaces is 200 mm and at the parting line 200 mm plus draft. Depth of draw is shown as the dimension between parting line and bottom cast surface.

11.17 Figure 25 illustrates use of datum targets and dimensioned phantom lines to control parting location and direction of draft.

11.17.1 Illustration A shows machining required symbols indicating that both the top and bottom surfaces are machined and the **DIM AND TOL** block provides 3 mm basic machine allowance. Datum X is shown on the phantom line that represents basic machine allowance for the bottom surface, Datum A is shown on the top machined surface, and a supplemental datum target is shown on the right side of the part dimensioned 10 mm from Datum X.

11.17.2 The interpretation for A shows the parting line at the phantom line that represents basic machine allowance for the top surface and origin of draft is shown at the phantom line that represents machine allowance for the bottom surface which is the end closest to the datum target. When a datum target is shown on a drafted surface, the origin of draft must be on the end closest to the datum target and location of the datum target must be calculated by adding basic draft to the existing dimensions [300 + (10 x tan 1 degree)= 300.17]. If a supplier determines that this is not practical he must negotiate a change to relocate origin of draft or the location of the datum target. Depth of draw is shown as the dimension between top and bottom cast surfaces.

11.17.3 Draft is shown pictorially and dimensioned on an unmachined surface in Illustration B to indicate that control of draft is necessary. A machining required symbol is shown to indicate that the bottom surface is machined and draft is shown pictorially on both ends. The **DIM AND TOL** block shows 3 mm machine allowance, 1 degree external, and 3 degree internal draft.

11.17.4 The interpretation for B shows the parting line at the top surface of the casting and origin of draft at the phantom line that represents basic machine allowance. Depth of draw is shown as the distance between the top cast surface and the bottom cast surface.

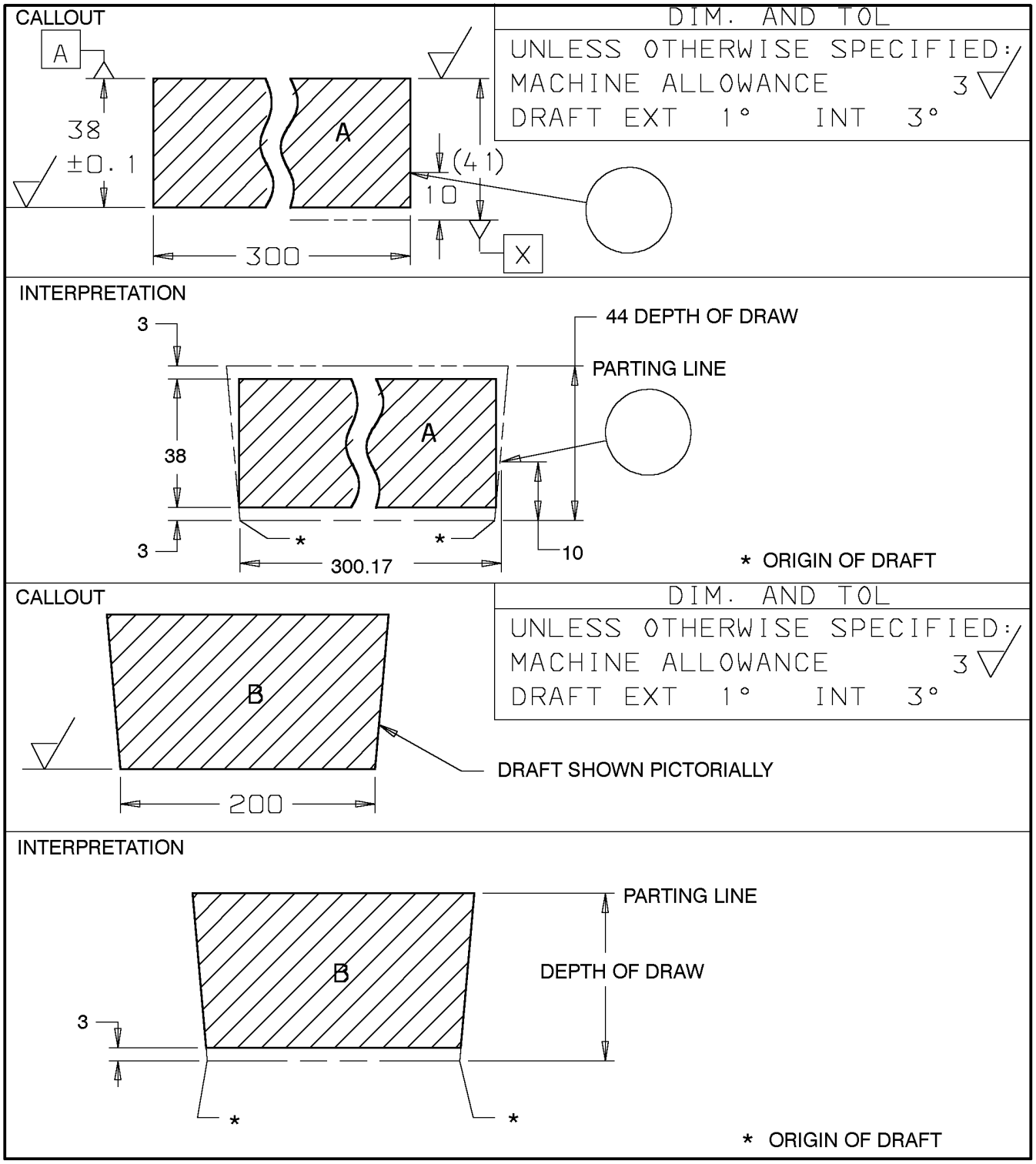


Figure 25 - Parting Location - Origin of Draft

11.18 Matching Draft - Where draft remains on the finished part, and the casting supplier elects to locate the parting line between nondrafted surfaces, on a plane other than through the center, as illustrated in Figure 26, the draft angle shown in the **DIM AND TOL** block must be applied to the long portion of the draft and additional draft (matching draft) applied to the short portion to match draft at the parting line, unless otherwise specified on the drawing.

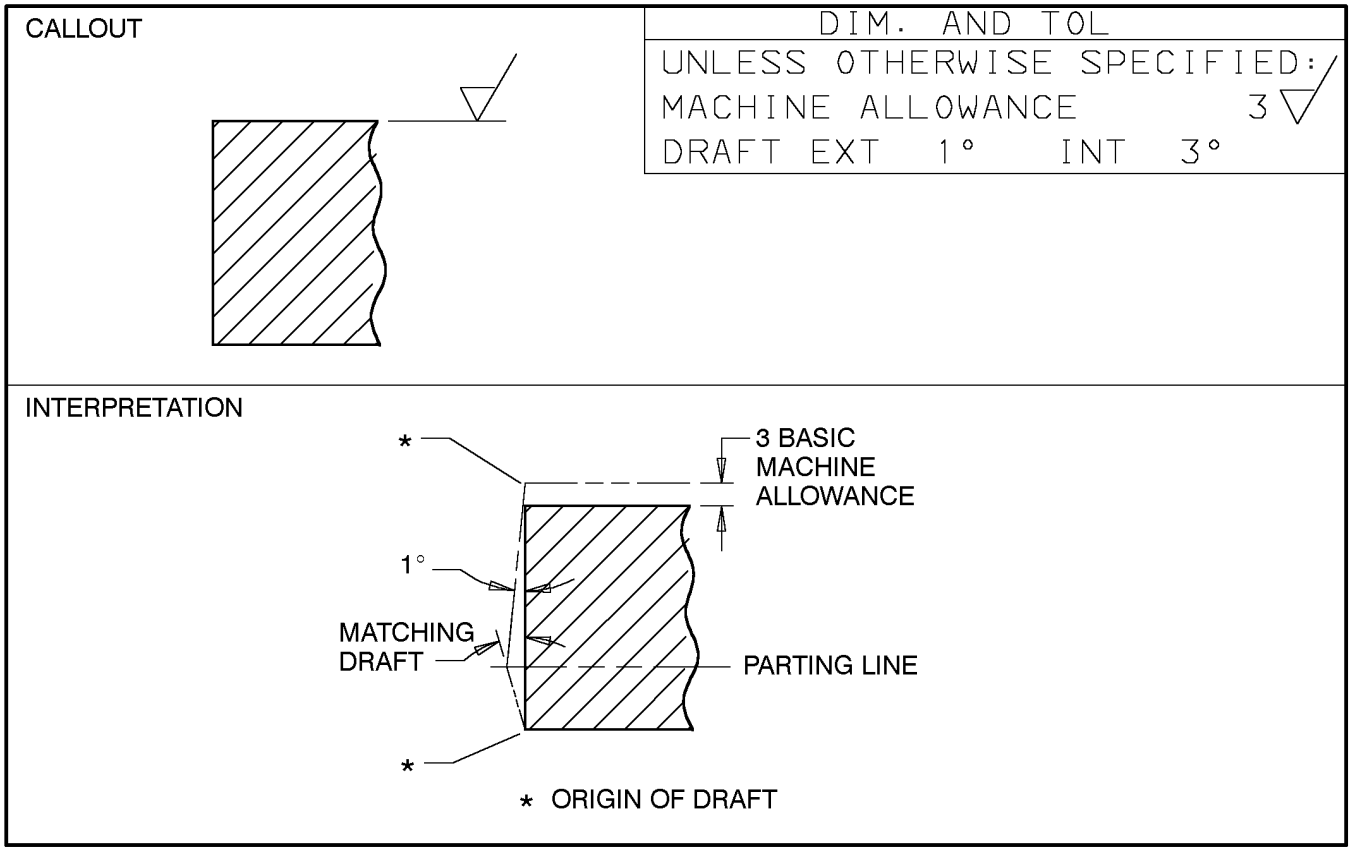


Figure 26 - Matching Draft - Unequal Length - Draft not Shown Pictorially

11.18.1 Figure 26 illustrates matching draft where draft with unequal lengths remains on the finished part. The callout shows a machining required symbol on the top horizontal surface and 3 mm basic machine allowance, 1 degree external, and 3 degree internal draft in the **DIM AND TOL** block.

11.18.2 The interpretation shows that the top cast surface which is represented by a phantom line (3 mm basic machine allowance) is not drafted. Angled phantom lines illustrate draft of unequal lengths. A 1 degree draft is shown on the long leg while a larger draft angle (X degrees) is shown on the short leg to match the draft at the parting line.

12.0 SPECIAL APPLICATIONS AND CONSIDERATIONS OF PROFILE TOLERANCE FOR CASTING

12.1 Gate, Riser, and Fin Removal Tolerance - An exception from the normal interpretation for profile tolerances that require all of the material to be within the profile tolerance zone is made in 1E0576 Foundry Instructions specification for cleaning. The exception controls removal of gates, riser, fins, etc. and permits a plus tolerance of 1.5 mm or a minus tolerance of 1 mm measured from the adjoining cast surface provided that machined surfaces from which gates and/or risers are removed clean up. The gates, risers, fins, etc., after removal may protrude mm beyond the maximum profile boundary when the adjoining cast surface is at the maximum profile boundary or the gates, risers, fins, etc. after removal may be 1 mm less than the minimum profile when the adjoining cast surface is at the minimum profile boundary.

12.2 Wall Thickness Tolerance (Castings) - The general profile tolerance shown in the casting **DIM AND TOL** block applies to all casting surfaces unless otherwise specified. This includes the two cast surfaces that form a wall, rib, gusset, or web unless thickness is restricted in some other manner such as shown in Figure 27.

12.2.1 The general profile tolerance ⌓ 4 X Y Z applies to each surface of Wall A. If each surface were at the maximum profile tolerance, wall thickness would be 16 + 2 + 2 = 20 and at the minimum profile tolerance, wall thickness would be 16 - 2 - 2 = 12.

12.2.2 Each surface of Wall B must lie within the profile tolerance zones established by ⌓ 4 X Y Z. Wall thickness at maximum profile tolerance is 5.5 + 2 + 2 = 9.5. Minimum wall thickness is restricted by the **4 MIN** dimension.

12.2.3 General note **WALL THICK 8 BSC 6 MIN** applies to Wall C. Each surface of Wall C must lie within the profile tolerance zones established by ⌓ 4 X Y Z. Wall thickness at maximum profile tolerance is 8 + 2 + 2 = 12. Minimum wall thickness is restricted to 6 by the general note.

12.2.4 Outer surface of Wall D must lie within the special profile tolerance zone established by ⌓ 3 X Y Z while inner surface of Wall D is covered by the general profile tolerance ⌓ 4 X Y Z.

Maximum wall thickness is 5.5 + 1.5 + 2 = 9. Minimum wall thickness is restricted by the **4 MIN** dimension.

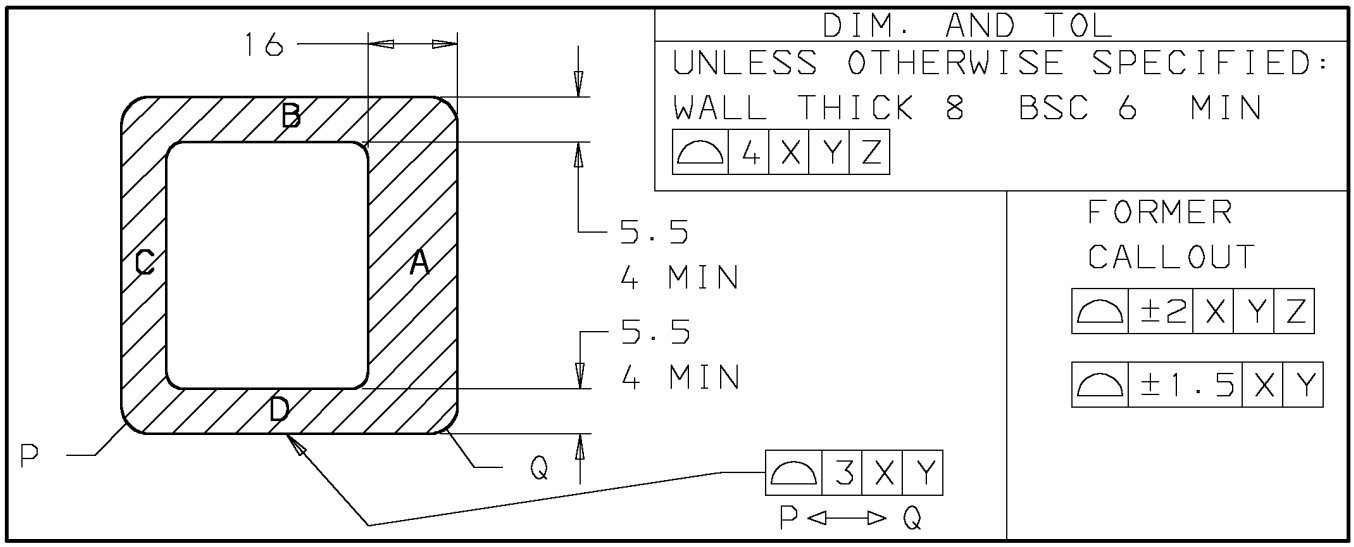


Figure 27 - Illustration - Wall Thickness

12.3 Cored Openings or Passages Tolerance - The general profile tolerance shown in the **DIM AND TOL** block applies to casting surfaces unless otherwise specified. This includes a surface or surfaces forming an opening or internal passage unless restricted in some other manner as shown in Figure 28.

12.3.1 All surfaces of the cored openings must lie within the profile tolerance zones established by ⌓ 4 X Y Z.

12.3.2 The maximum size of the basic 12.5 diameter opening is 12.5 diameter+ 2 + 2 = 16.5 diameter while the minimum diameter is restricted by the **11 MIN** dimension.

12.3.3 Size of the rectangular passage is covered by the special profile tolerance ⌓ 1.5 restricting maximum size to 12.5 +0.75 +0.75 = 14 by 22.5 +0.75 +0.75 = 24 and restricting minimum size to 12.5 -0.75 -0.75 = 11 by 22.5 -0.75 -0.75 = 21.

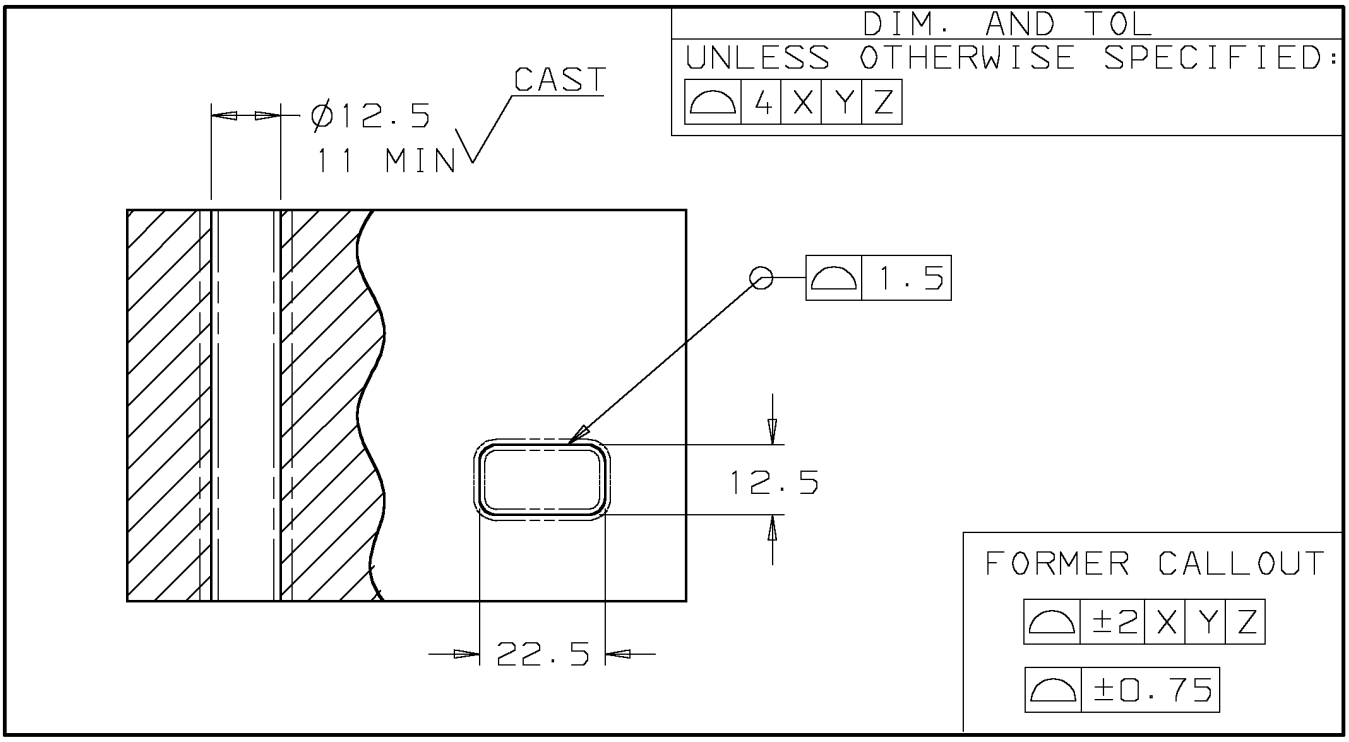


Figure 28 - Illustration - Cored Openings

13.0 SPECIAL APPLICATIONS AND CONSIDERATIONS OF PROFILE TOLERANCE FOR FORGINGS

13.1 Forging Trim - Trimmed Surface Remains on Finished Part - Trimming is a secondary operation used by the forging supplier to remove flash extension and draft from the forging. The amount of flash extension and draft removed is controlled by the type of trim used: normal trim; close trim; straight trim, pierce, or punch; and loose trim. When the trimmed surface and/or draft remains on the finished part, the type trim required is shown pictorially and dimensioned as follows:

13.1.1 Normal Trim - Normal trim is required when the surface is dimensioned to the origin of draft and full draft is shown pictorially as illustrated in Figure 29. Tolerance is provided by the general profile tolerance in the **DIM AND TOL** block. Refer to 1E0100 Forger Instructions specification for limits on flash extension due to forging shift during the trimming operation.

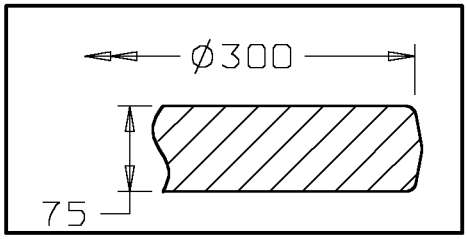


Figure 29 - Normal Trim/Finished Part

13.1.2 Close Trim - Close trim is required when the surface is dimensioned to the origin of draft and to the trim line and only half of the draft is shown pictorially as illustrated in Figure 30. Tolerance is provided by the general profile tolerance in the **DIM AND TOL** block.

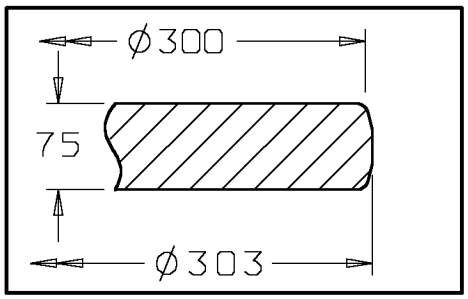


Figure 30 - Close Trim/Finished Part

13.1.3 Straight Trim, Pierce, or Punch - Straight trim, pierce, or punch is required when the surface is shown pictorially as a straight line without draft and a maximum radius is shown on the corner as illustrated in Figure 31. Tolerance is provided by the general profile tolerance in the **DIM AND TOL** block or by a smaller special profile tolerance as shown in Figure 31. The **MAX R** callout controls the radius caused by material pull down or die shift.

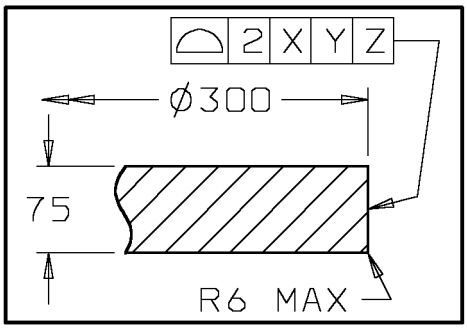
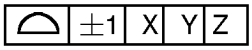


Figure 31 – Straight Trim/Finished Part

FORMER CALLOUT



13.1.4 Loose Trim – Loose trim is required when the surface is dimensioned to both the origin of default and the trim line and full draft extension is shown pictorially as illustrated in Figure 32. Tolerance is provided by the general profile tolerance in the **DIM AND TOL** block or by a special profile tolerance as shown in Figure 32.

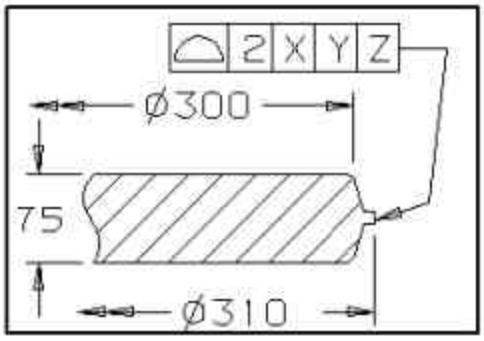
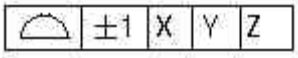


Figure 32 – Loose Trim/Finished Part

FORMER CALLOUT



13.2 Straight Punch Permitted for Stepped Holes - The callout in Figure 33 shows two machined internal diameters with special machine allowances and profile tolerances to permit a straight punch.

The interpretation shows minimum machine allowance, basic machine allowance, and maximum machine allowance for each diameter. The tolerance zones overlap. Any combination of punched or drafted surfaces within the specified tolerance zone is permitted.

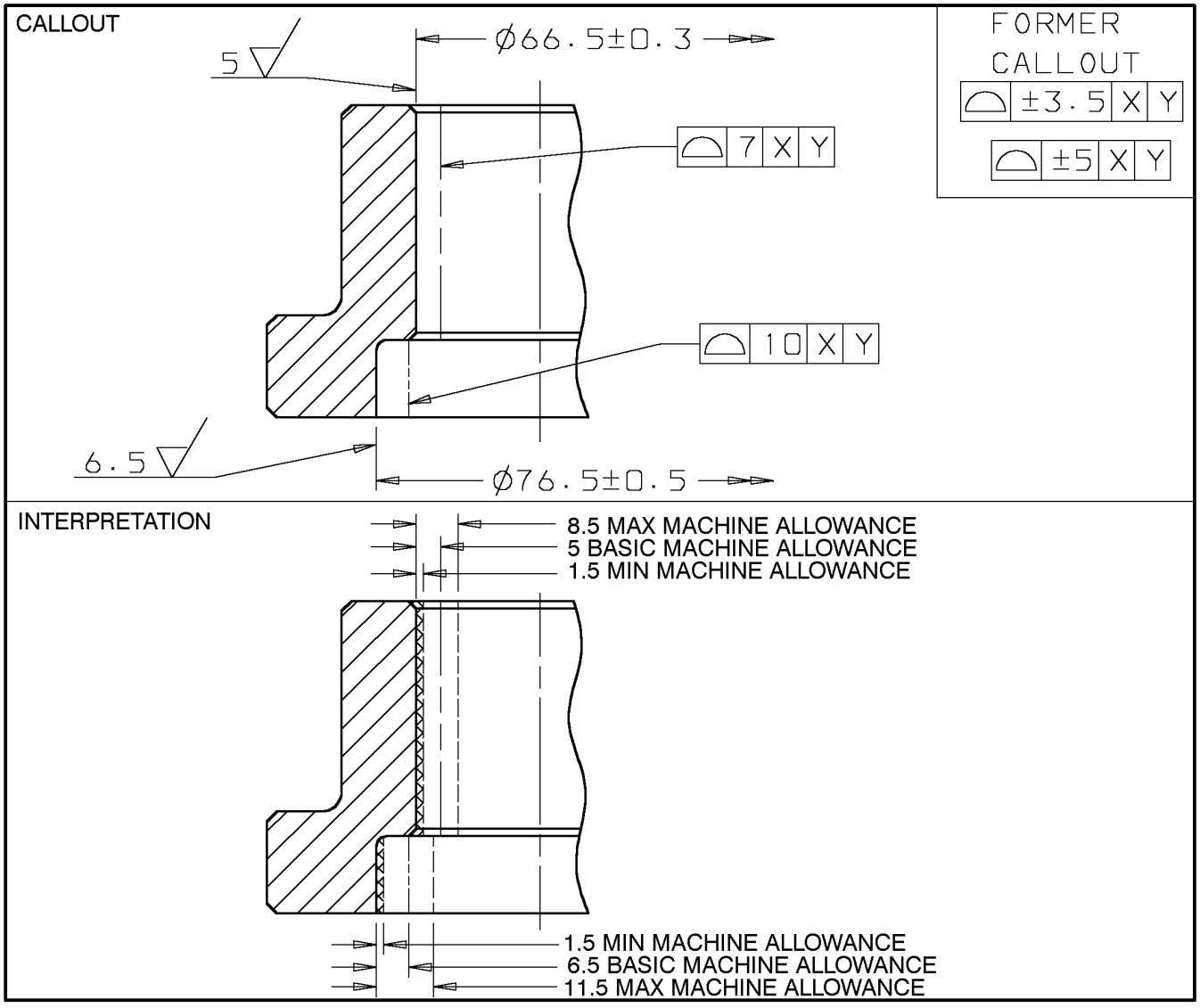


Figure 33 - Stepped Holes That Allow Straight Punch

13.3 Flash on Forging Produced on Upsetter Caused by Split Dies - 1E0100 Forger Instructions specification, control of flash removal on forgings produced on upsetter, permits a plus tolerance of 1 mm measured from the adjoining forged surface. The flash between the split dies and between the header die and the split dies after flash removal may protrude 1 mm beyond the maximum profile boundary when the adjoining forged surface is at the maximum profile boundary.

13.4 Application of 1E0100 Forger Instructions Tolerances to The Bar Portion of an Upset Forging

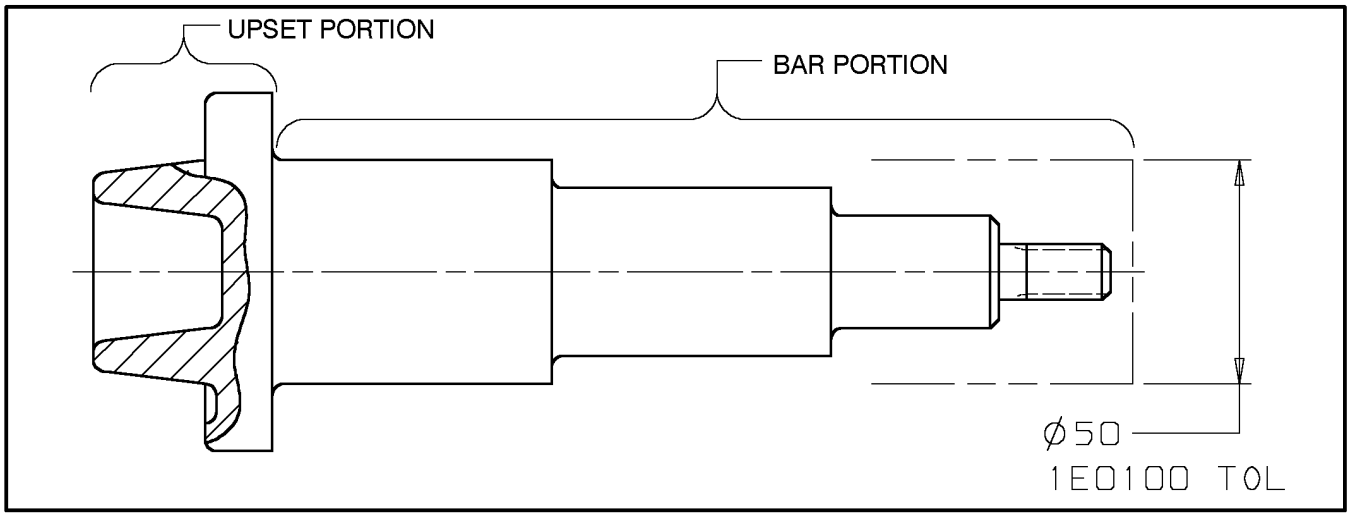


Figure 34 - 1E0100 Tolerance Upset Forging

13.4.1 Figure 34 illustrates use of 1E0100 Forger Instructions tolerance to control the bar portion of an upset forging. The tolerances in 1E0100 Forger Instructions include allowances to cover indentations and bulges caused by gripper dies in addition to normal wrought steel mill tolerances for bar. 1E2177 Mill Tolerances is specified when gripper die marks are not permitted on the bar portion. Normally the specified bar size is enough larger than the machined size to provide a machine allowance that covers the bar portion for both size and location and includes allowance for gripper die marks, straightness, perpendicularity, and axial alignment with the upset portion of the forging.

13.4.2 The upset portion of the forging that is formed by the forging dies is covered by the profile tolerance specified on the drawing.

14.0 SUPPLIER RESPONSIBILITY FOR FEATURE PROCESSING

14.1 Supplier Responsibility Indicated by Surface Texture Symbol. The supplier is responsible for producing the surface to which a surface texture symbol is attached when **CAST** or **FRG** is shown on the horizontal bar at the top of the symbol. The interpretations for supplier responsibility basically apply to casting and forging suppliers alike, however, they are used more frequently with castings because many forging suppliers do not perform machining operations. Interpretations for surface texture symbols showing supplier responsibility are shown in Figures 35 thru 38.

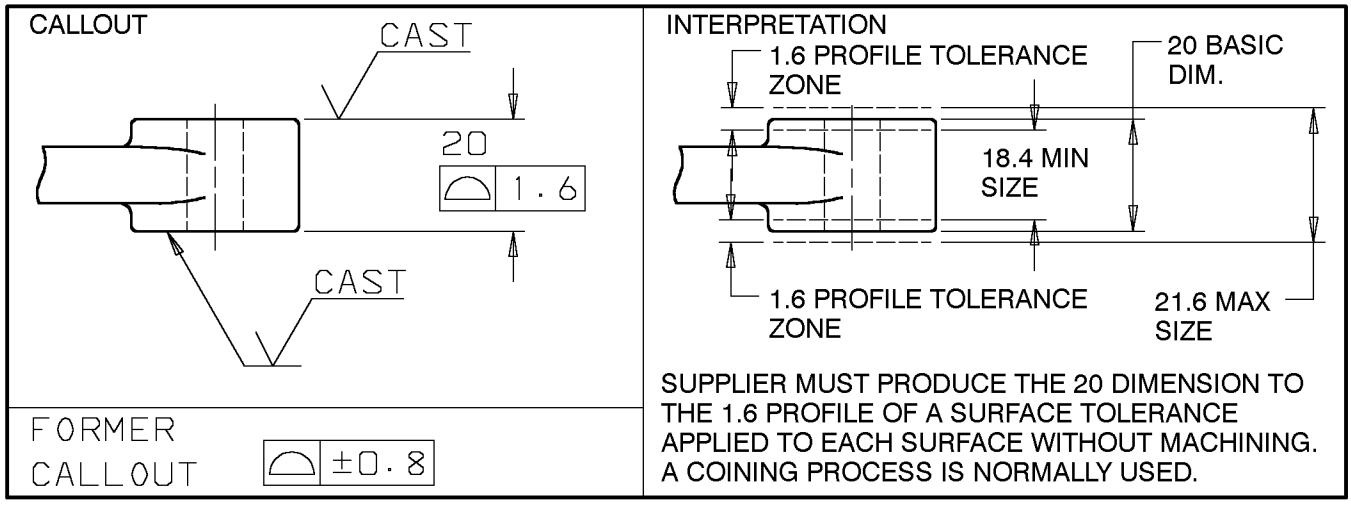
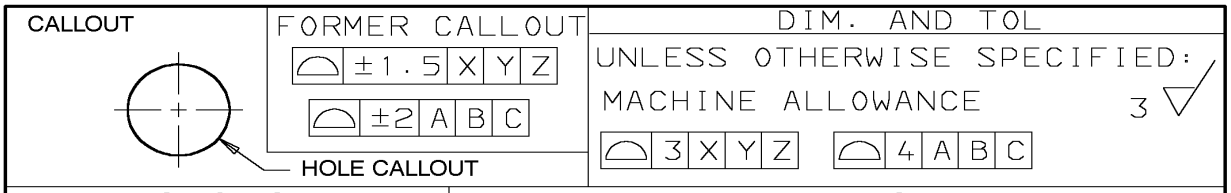


Figure 35 - Supplier Responsibility - Reduced Tolerance Without Mandatory Machining (Coining)



HOLE CALLOUT	INTERPRETATION
⌀30 H CAST	SUPPLIER MUST MACHINE HOLE TO H SURFACE TEXTURE AND THE 3mm GENERAL PROFILE TOLERANCE IN THE DIM. AND TOL BLOCK.
⌀30±0.1 H CAST	SUPPLIER MUST MACHINE HOLE TO H SURFACE TEXTURE AND TO THE ±0.1 SIZE TOLERANCE.
⌀30±0.1 H	SUPPLIER MUST PROVIDE 3mm BASIC MACHINE ALLOWANCE FROM THE DIM. AND TOL BLOCK. HOLE WILL BE MACHINED BY CATERPILLAR INC. TO THE H SURFACE TEXTURE.
⌀30±0.1 H ∞	SUPPLIER MUST CAST SOLID AND CATERPILLAR INC. WILL MACHINE HOLE FROM SOLID TO H SURFACE TEXTURE.
⌀30 CAST	SUPPLIER MUST CAST OR CORE HOLE TO 3mm GENERAL PROFILE TOLERANCE IN THE DIM. AND TOL BLOCK. MACHINING IS PROHIBITED.
⌀30 CAST	SUPPLIER MUST PROVIDE HOLE (CAST, CORE, MACHINE) TO 3mm GENERAL PROFILE TOLERANCE IN THE DIM. AND TOL BLOCK.
⌀30-B	SUPPLIER MUST CAST SOLID. HOLE WILL BE DRILLED FROM SOLID BY CATERPILLAR INC.

Figure 36 - Supplier Responsibility - Hole Callouts

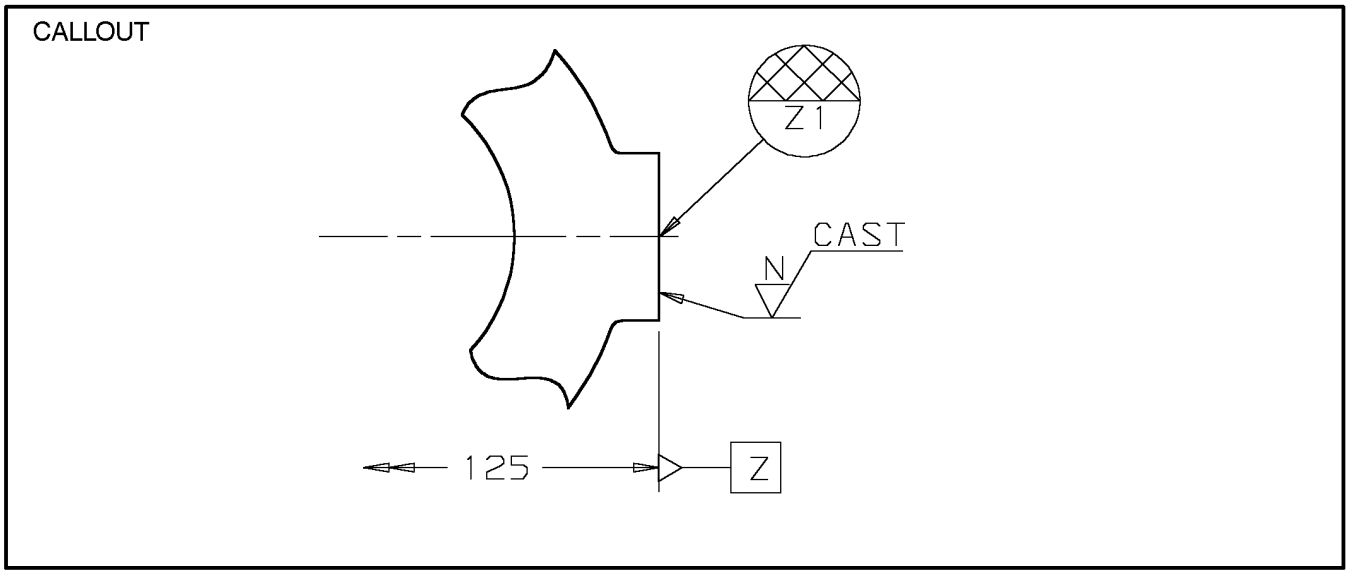


Figure 37 - Supplier Responsibility - Casting with Machining Required

14.2 The callout in Figure 37 indicates that the casting supplier must machine the flat surface within the N surface texture requirement. The datum target on the machine surface is used to establish Datum Plane Z.

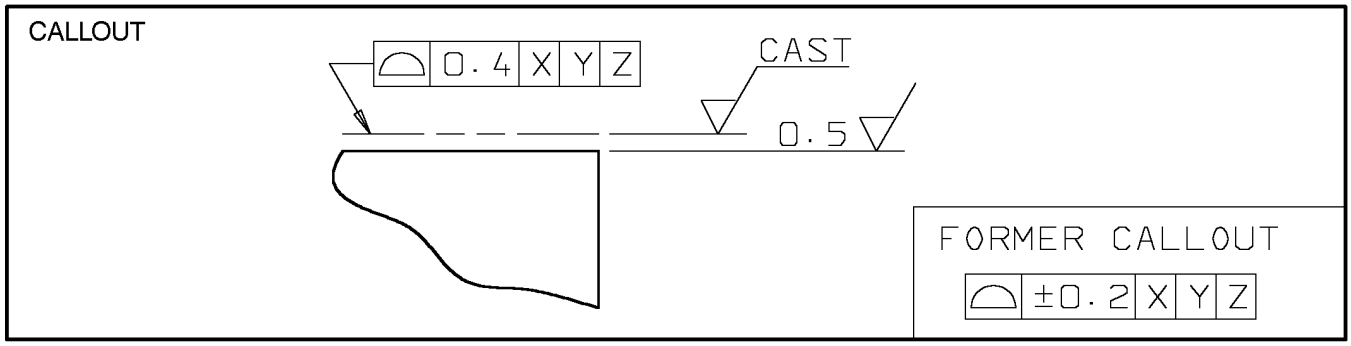


Figure 38 - Supplier Responsibility - Casting with Machining Required and Reduced Machine Allowance

14.3 The callout in Figure 38 indicates that the casting supplier is required to machine the surface represented by the phantom line to achieve the specified basic machine allowance of 0.5 mm within the profile of a surface tolerance of 0.4 mm with respect to Datums X, Y, and Z. This reduces the amount of machining required to complete the part after the casting is received.

15.0 INTERPRETATION OF EXAMPLES

15.1 Interpretation of Example Casting (Figure 39)

15.1.1 DIM AND TOL Block - Requirements shown in the **DIM AND TOL** block apply unless otherwise specified on the body of the drawing/model.
3 mm Basic machine allowance

10 Radius Fillets
15 Radius Corners
1 Degree External Draft

Applies to rough cast surface remaining on the completed part.

	3	X	Y	Z

A 3 mm profile tolerance on rough cast surfaces with respect to Datums X, Y, and Z as established by datum targets on the rough cast surface.

	4	A	B	C

A 4 mm profile tolerance for rough cast surfaces remaining on the completed part with respect to Datums A, B, and C.

A 3.5 mm machine allowance with a 4 mm profile tolerance with respect to Datums X, Y, and Z applies to surfaces designated 3.5 ∇ on the body of the drawing/model.

15.1.2 Datums and Datum Targets - Datums X, Y, and Z are established by datum targets on the rough cast surface; 3 datum targets for Datum X, 2 datum targets for Datum Y, and 1 datum target for Datum Z . Each datum target is 12 to 15 mm in diameter with its center located where the leader points on the drawing. Datum A is established by the flat machined surface on the opposite side of the casting from Datum X; Datum B is established by the long machined edge opposite Datum Y; and Datum C is established by the short machined edge opposite Datum Z . Datums X , Y, and Z are used to establish rough casting requirements and to locate the part for initial machining. Datums A, B, and C are for establishing completed part requirements.

15.1.3 Machine Allowance - The parting line has been assumed to be parallel to Datum X in the right side view. The casting supplier may locate the parting line at any point along the 41 mm thickness. Draft has been included in the 3.5 mm basic machine allowance for the edge surfaces designated 3.5 ∇ . A 4 mm profile tolerance for these surfaces is shown in the lower portion of the **DIM AND TOL** block. Minimum machine allowance for these surfaces is 1.5 mm and maximum machine allowance is 5.5 mm. Machine allowance for surfaces designated ∇ is controlled by the 3 mm basic machine allowance shown in the upper portion of the **DIM AND TOL** block and the 3 mm profile tolerance from Datums X, Y, and Z applies. Minimum machine

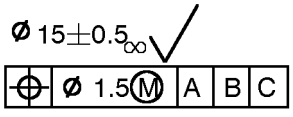
allowance for these surfaces is 1.5 mm and maximum machine allowance is 4.5 mm. Basic casting thickness is 44 mm (38 machined thickness +3 machine allowance +3 machine allowance). The 41 reference dimension from the phantom line (machine allowance) at Datum X to the machined surface, Datum A, indicates the reference machine tool setting from Datum X to machine the surface for Datum A.

15.1.4 Fillets - The fillets between the R15 boss and the body of the casting are not dimensioned on the body of the drawing. The R10 for fillets shown in the **DIM AND TOL** block is applicable and the 3 mm profile tolerance also applies to the fillets from Datums X, Y, and Z.

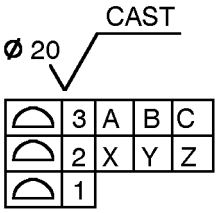
15.1.5 Corners - The radius at the lower left corner of the casting is not dimensioned on the body of the drawing/model. The R15 corner shown in the **DIM AND TOL** block is applicable and the 3 mm profile tolerance also applies to the corner from Datums X, Y, and Z.

15.1.6 Holes - Casting supplier responsibility for holes called out are as follows:

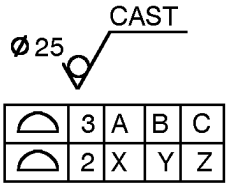
15.1.6.1 5 X 10.5-B Casting supplier does not provide the holes. The casting is made solid and the holes are machined later.

15.1.6.2 

Casting supplier must cast solid. The hole will be machined later. The 1.5 diameter position tolerance controls location from Datums A, B, and C for the completed part.

15.1.6.3 

Casting supplier must produce the hole. It may be cast, cored, or machined. The 1 mm profile tolerance controls size. The 2 mm profile tolerance controls location from Datums X, Y, and Z for the rough casting. The 3 mm profile tolerance controls location from Datums A, B, and C for the completed part.



15.1.6.4 Casting supplier must produce the hole with a casting process. Machining is prohibited. The 2 mm profile tolerance controls size and location from Datums X, Y, and Z for the rough casting. The 3 mm profile tolerance controls location from Datums A, B, and C for the completed part.

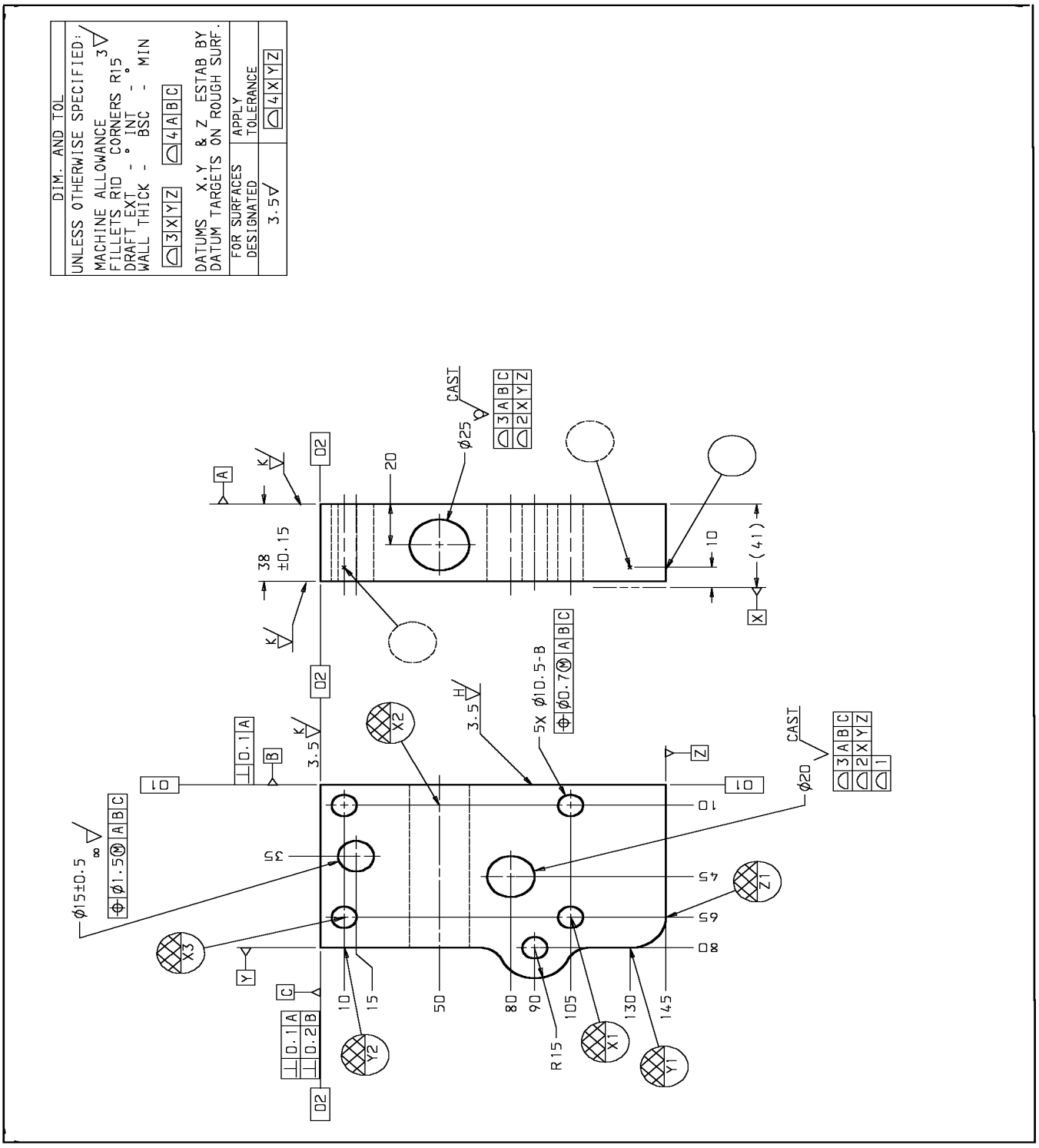


Figure 39 – Example Casting

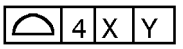
INTERPRETATION - PROFILE TOLERANCE-CASTING AND FORGING DRAWINGS	DATE	CHG NO	NUMBER
	18 MAY 1999	05	1E2600

15.2 Interpretation of Example Forging (Figure 40)

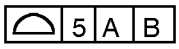
15.2.1 DIM AND TOL Block - Requirements shown in the **DIM AND TOL** block apply unless otherwise specified on the body of the drawing/model.

15.2.2 3.5 mm Basic Machine Allowance

10 Radius fillet
5 Degrees external draftApplies to rough forged surfaces remaining on
7 Degrees internal draftthe completed part.



A 4 mm profile tolerance on rough forged surfaces with respect to Datums X and Y as established by datum targets on the rough forged surface.



A 5 mm profile tolerance on rough forged surfaces remaining on the completed part with respect to Datums A and B.

15.2.3 Datums and Datum Targets - Datums X and Y are established by datum targets on the rough forged surface; 3 datum targets equally spaced in relation to the part number location for Datum X and 3 datum targets equally spaced in relation to the part number location contacted by a three-way equalizer to establish Datum Y. Each datum target is 12 to 15 mm diameter with it's center located where the leader points on the drawing/model. Datum A is established by the 82.55 ±0.08 diameter, and Datum B is established by the end face of the hub. Datums X and Y are used to establish rough forging requirements and to locate the part for initial machining. Datums A and B are for establishing completed part requirements.

15.2.4 Machine Allowance - Machine allowance for the surfaces designated K and F is controlled by the 3.5 mm basic machine allowance shown in the **DIM AND TOL** block and the 4 mm profile tolerance to Datums X and Y applies. Minimum machine allowance for these surfaces is 1.5 mm and maximum machine allowance is 5.5 mm. The 3.5 mm basic machine allowance (**DIM AND TOL** block) does not include draft and has been applied only to non-drafted surfaces. Allowances for draft and flash extension which will permit close trim have been included in the following machine allowances. Machine allowance for the surface designated H is 5 mm with a 7 profile tolerance. Minimum machine allowance is 1.5 mm and maximum machine allowance is 8.5 mm. Basic machine allowance for the surface designated 4.5 N is 4.5 mm with a 6 mm profile tolerance. Minimum machine allowance is 1.5 mm and maximum machine allowance is 7.5 mm. Basic machine allowance for the surface designated 5.5 1.6 is 5.5 mm with a 8 mm profile tolerance. Minimum machine allowance is 1.5 mm and maximum machine allowance is 9.5 mm. This requirement is based on continuation of draft from the 100 diameter. Basic machine allowance for the surface designated 100 mm.

Basic machine allowance for the surface designated 6.5 1.6 is 6.5 mm with a 10 mm profile tolerance. Minimum machine allowance is 1.5 mm and maximum machine allowance is 11.5 mm.

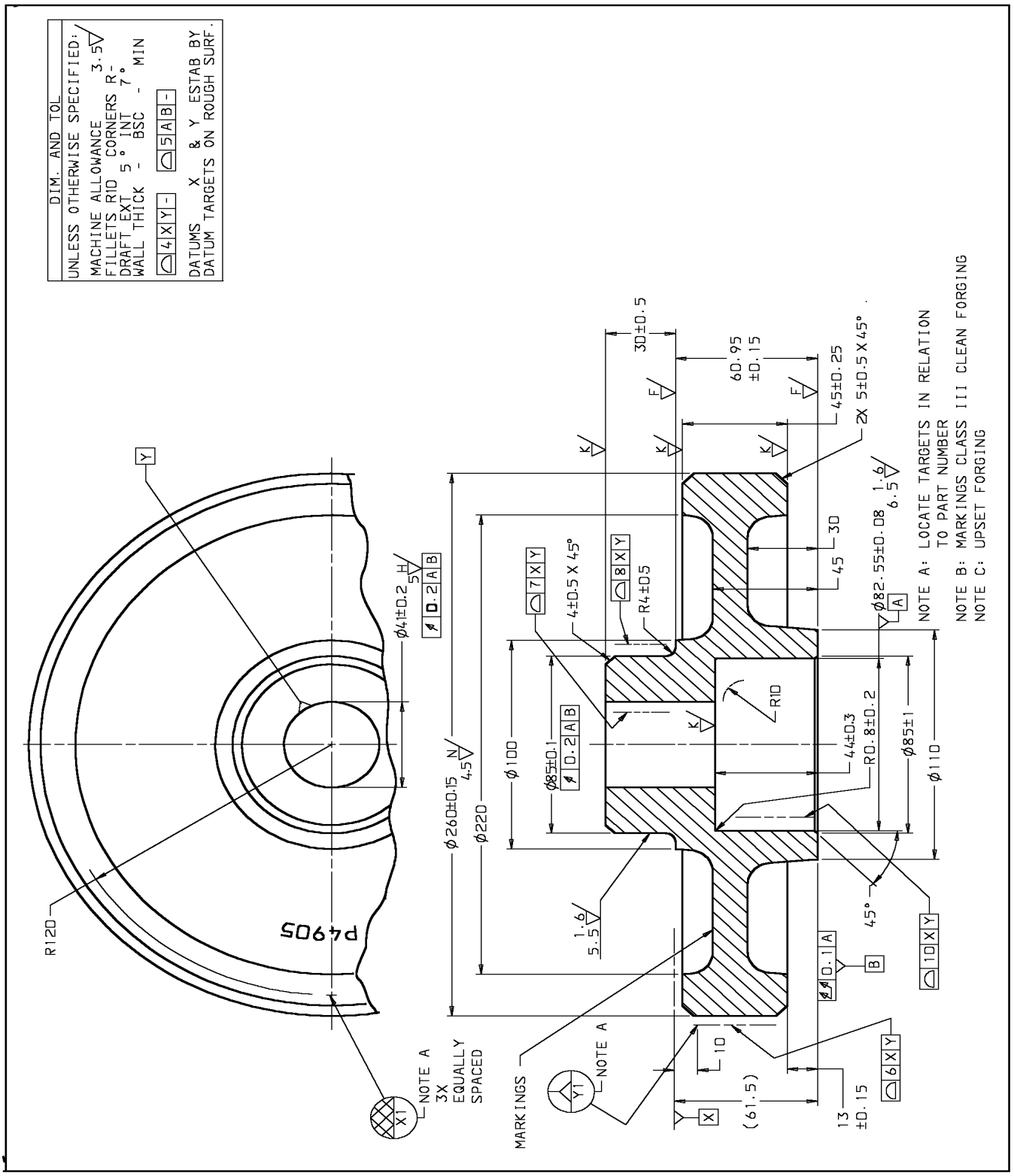
15.2.5 Fillets - The fillets between the web and outer rim and the web and hub are not dimensioned on the body of the drawing. The R10 for fillets shown in the **DIM AND TOL** block is applicable and controls size of the fillets. Tolerance for the fillets is also provided by the profile tolerances called out in the **DIM AND TOL** block; 4 mm profile tolerances with respect to Datums X and Y and 5 mm profile tolerance with respect to Datums A and B.

The R10 dimension for the fillet inside the hub overrides application of machine allowance to the 0.8 ±0.2 machined radius and establishes the R10 fillet as tangent to both the 3.5 mm machine allowance for the bottom of the hole and the 6 mm machine allowance for the sides of the hole. The fillet may be R8 (10-2 = 8) at maximum machine allowance or a sharp corner at minimum machine allowance. The forging surface may lie anywhere within the zone between maximum and minimum machine allowance.

15.2.6 Web Thickness - The web is not machined. Basic thickness is 15 mm (45-30). Tolerance on thickness and location is provided by the **DIM AND TOL** block; 4 mm profile tolerance with respect to Datums X and Y and 5 mm profile tolerance with respect to Datums A and B.

16.0 REFERENCES

Caterpillar Specifications 1E0011, 1E0012, 1E0100, 1E0576, 1E2122, 1E2177



INTERPRETATION - PROFILE TOLERANCE-CASTING AND FORGING DRAWINGS	DATE	CHG NO	NUMBER
	18 MAY 1999	05	1E2600

Figure 40 – Example Forging