TITLE
WORK INSTRUCTIONS, GENERAL PRINT AMENDMENT

Revision	EXPLANATION OF CHANGE	Approvals	DATE
A	Initial Release	WB, GR	02/26/1999
B	Add PMR caveat	WB, GR	05/13/1999
C	Add Surface Treatment Stipulation	MF, DL	10/20/2000
D	Clarify Concentricity paragraph	MF, DL, GR	11/09/2000
E	Update Hole Tolerance and add Stock Material sections	BH, GR, PK	10/15/2004
F	Revised & Expanded	BH, GR, PK	06/0620/06
G	Updated to included helicoil threaded holes	BH, GR, PK	06/21/2006
H	Updated section 7 and 8.1	BH, GR, BP	03/21/2007
I	Confirm all features not measured	BH, GR, BP	07/26/2007
J	Revised sections 1, 4, 8 and 9, added examples 9a, 9b and 12	BH, GR, BP	07/06/2012
K	Added ANSI tapered pipe thread detail	BH, GR, BP	11/26/2012
L	Clarified implied angular location tolerance sect 6.11. Rev'd example 3 to 3a, added example 3b	BH, GR, BP	01/31/2013
M	Sect 6.1 and 6.2 updated wording for consistency Sect 6.10 expanded to reflect current Inspection capability. Added examples 6.10a, and 6.10b. Revised example No.s to reflect location in text. Sect 8.5 added AS5202 port callout. Sect 8.6 remove obsolete MS-16142 add Sect 8.7 clarified thread-milled NPT vanishing threads.	KO, MJ, GF	09/13/2018

1. Purpose

To define general design considerations, common machining practices, and minimum inspection requirements for parts and assemblies designed and manufactured by Barber-Nichols, Inc. and subcontractors.

2. Scope

This work instruction applies to Barber-Nichols designed parts and assemblies whose specifications include DS-703 as an amendment. Additionally, BNI part and assembly drawings approved prior to 1999 are assumed to include DS-703 as an amendment unless otherwise declared.

3. References

FORM, PRELIMINARY MATERIAL REVIEW, BNQF-110
WORK INSTRUCTIONS, ROUTE CARDS, BNWI-9001

4. General Application

4.1 Finished parts must meet all drawing requirements and specifications unless otherwise exempted.

4.2 Route card instructions shall take precedence over drawing specifications and this document.

4.3 BNI drawings may contain additional tolerancing and inspection requirements not included here.

4.4 Application of DS-703 is limited to BNI designed parts and assemblies drawn on BNI title block or letterhead. DS-703 may be applied to parts designed and/or drawn by others with prior agreement and as stated on the route card.

4.5 Design and Engineering are responsible for the completeness and correctness of all released BNI drawings and models. Manufacturing is responsible for producing parts according to the drawing and Route Card provided. Inspection is responsible for verifying the completeness and correctness of the part. All part features regardless of inspection requirement must be accounted for.

5. Specific Application

All drawing specifications, dimensions and geometric tolerances are to be interpreted per ANSI/ASME Y14.5M-1994 unless otherwise noted in the titleblock. Default tolerances of size and angularity are specified in the drawing title block. Default surface finish is specified in the drawing title block. Default corner chamfer and fillet radius are specified in the drawing title block. Absent specification, parts less than 10.0 inches in diameter will have a maximum corner

chamfer of .02 inch and a maximum fillet radius of .04 inch. Parts 10.0 inches in diameter and greater will have .02/.06 inch corner chamfers and .02/.08 fillet radii. All may be considered "Handwork". Additional common practices for design, machining, and inspection are given below.

6. General Dimensions

6.1 Any suspect dimension or part condition may be verified at the discretion of Quality Assurance after visual inspection or if it has been previously identified as being out of tolerance.

6.2 Size tolerances for decimal dimensions of two places (or less) to the right of the decimal point need not be verified or recorded by QA. Fractional dimensions will be considered "two-place" dimensions and only require inspection where noted.

6.3 Size tolerances for decimal dimensions of three places, whether given specifically or by titleblock default, will be verified and documented by QA except for drilled holes as noted below.

6.4 Size tolerances of four-place (or greater) decimal dimensions will be given specifically and will be verified and documented by QA except for reamed holes as noted below.

6.5 Angular tolerances given by titleblock default will be visually inspected. Angular tolerances given specifically will be verified and documented by QA unless larger than title block default.

6.6 Locational tolerances (true position) equal to or greater than .010 inch need not be verified or documented by QA.

6.7 Orientation tolerances (perpendicularity, parallelism, angularity) equal to or greater than .010 inch need not be verified or documented by QA unless the tolerance is less than .001 inch per running inch of the controlled feature.

EXAMPLE 6.7a Two faces of a 20 inch diameter cylinder are required to be parallel within .010 inch or .0005 inch per running inch of feature. Verification and documentation by QA are required.

EXAMPLE 6.7b A 10 inch diameter cylinder 2.0 inch long is required to be perpendicular to a shoulder within .010 inch or .005 inch per running inch. Only functional inspection is required.

6.8 Run-out tolerances (circular and total) equal to or greater than .010 inch require only functional inspection unless the tolerance is less than .001 inch per running inch of the controlled feature (see examples above).

6.9 Form tolerances (flatness, straightness, circularity, cylindricity) require verification and documentation by QA.

6.10 Profile tolerances (of line or surface) only contained in a single plane, which could be inspected using a shadowgraph will be inspected. Profiles that will be inspected in this manner (or by an equivalent method) include:

Impeller Hub and Shroud profiles – see Example 6.10a
Axial turbine blades profiles – see Example 6.10a
Conical nozzle profiles using impressions
Conical diffuser profiles using impressions

Non-planar profile inspection at BNI should be limited to specified regions of features.

Non-planar profiles, like complex inducer/impellers, may be inspected on the machine during milling. For a truly independent inspection on the CMM at BNI 4 requirements must be met:

1. There must be complete fidelity between the part's solid model and the expected machined surfaces.
2. The solid model must be at a known material condition, nominal , maximum or minimum and that condition must be consistent throughout the model.
3. The part must be fixtured in the CMM so that the part origin is absolutely unambiguous. Angular orientation features like pins or keyways must be present at the time of inspection.
4. The surface being inspected must allow the probe or scanhead to contact the part normal to the surface. See Example 6.10c.

If more complete inspection is required, or if the above conditions cannot be met inspection at an outside service may be required.

In general profile tolerances equal to or greater than .010 do not require verification or documentation by QA.

6.11 Angular location (true position). Absent a datum controlling angular or rotational location of radial features a vertical or horizontal center mark on the drawing will be taken as datum. Features located on a common vertical or horizontal center mark, or dimensioned from a common center mark will be located relative to one another within the sum of their locational tolerance.

Angular location of features relative to drawing centermarks not controlled by true position shall be within the titleblock angle tolerance. Features controlled only by their location on a centermark shall have an angular location within the title block angular tolerance.

EXAMPLE 6.11a A radial hole passes through one wall of a cylinder at top dead center (no locational tolerance implied) a regular hole pattern is shown in the end of the cylinder with one hole at top dead center. (The two features share a common vertical center mark line) The radial hole through the cylinder wall will be considered the clocking control for the hole pattern. Any

additional features located angularly from the same vertical line would also use the radial hole as a clocking control. The implication is that all of the hole patterns controlled by true position will include the TDC hole as an angular datum. The notch shown in detail section B – B has no locational tolerance given other than the location on the horizontal centerline. The angular location of the notch relative to the bolt patterns and the TDC implied datum shall be within the titleblock angular tolerance of ± 2°.

EXAMPLE 6.11b Size tolerances for the patterned T-slot features are controlled by the profile tolerance to datums –A- and –B-. Angular location between pattern features is controlled by the "3 EQUALLY SPACED" note which is equal to declaring a 120° BASIC angle between pattern features also controlled by the Profile tolerance. The drawing implies an angular relationship between the pattern and the Ø.079 reamed hole that is controlled by the titleblock angular tolerance of ± 2°.

7. Features on a Common Axis

Uncontrolled diameters on a common axis or centerline as shown on the drawing shall fall within the locational tolerances described below. All typical inspection requirements apply. Uncontrolled features will be inspected at Inspection's discretion.

7.1 True position from the largest datum diameter to any diameter on a common axis shall be within a cylindrical tolerance zone equal to half the sum of the total size tolerance and regardless of feature size.

EXAMPLE 7.1 An uncontrolled shaft diameter has a size tolerance of ±.015 inch. The largest datum diameter has a size tolerance of ±.001 inch. The sum of the total size tolerance is .032 inch. The true position of the uncontrolled diameter must be within a cylindrical tolerance zone of .016 inch relative to the datum diameter and regardless of feature size. If other uncontrolled diameters exist on this shaft the tolerance zone must be calculated for each feature. For instance a different diameter on the common axis has a size tolerance of ±.005 inch. The sum of the total size tolerance is .012 inch. This diameter must be located within a .006 inch cylindrical tolerance zone relative to the datum diameter and regardless of feature size.

7.2 True position of an uncontrolled diameter on an axis solely defined by machined center datums shall be within a cylindrical tolerance zone equal to half the total size tolerance of the uncontrolled diameter and regardless of feature size.

EXAMPLE 7.2 An uncontrolled shaft diameter has a size tolerance of ±.015 inch. The only cylindrical datums are machined centers. The true position of the diameter must be within a cylindrical tolerance zone of .015 inch relative to the axis defined by the centers and regardless of feature size.

7.3 True position from the largest controlled diameter to any diameter on a common axis shall be within a cylindrical tolerance zone equal to half the sum of the total size tolerance and regardless of feature size.

EXAMPLE 7.3 A 1.50 inch diameter cylinder (±.015 inch size tolerance by titleblock default) is controlled perpendicular to a face. A 2.00 diameter (±.015 in. by default) is on a common axis but otherwise uncontrolled. The 1.50 inch diameter becomes the datum diameter since it is controlled. True position of the 2.00 inch diameter must be within a cylindrical tolerance zone of .030 inch relative to the 1.50 inch diameter and regardless of feature size.

7.4 In the case where two or more diameters share a common axis, and no controls are specified the largest or longest diameter will be considered datum. True position of any other diameter must be within a cylindrical tolerance zone equal to half the sum of the total size tolerances and regardless of feature size. (See Example 4)

8. General Holes, Drilled Holes, Reamed Holes, Threaded Holes

8.1 Holes (including drilled, countersunk and counterbored holes) intended to be produced with standard tools and absent any specified tolerance or surface finish requirement on the drawing will meet the following:

Holes with multiple features such as counterbores, countersinks, chamfers, drill points, pilot holes, etc. shall be considered one feature for all locational tolerances.

Unless otherwise specified drilled holes will have a 250 surface finish or better.

Unless otherwise specified, the size tolerances of all holes dimensioned to 3 places are:

.000 - .124 inch	+ .003/-.002 inch
.125 - .311 inch	+ .010/-.004 inch
.312 - .749 inch	+ .015/-.005 inch
.750 – 1.250 inch	+ .020/-.010 inch

Titleblock tolerances and inspection requirements apply to all other hole dimensions. Holes which require different size tolerance control, or which cannot be produced with standard tooling must be toleranced specifically. For example Ø.495/.505 thru, even though the ±.005 tolerance is standard for a three-place dimension. As a hole this must be toleranced specifically. Holes larger than Ø1.250 obey the titleblock tolerances.

Where practical drilled holes should include a machined lead-in chamfer of .010/.020 inch with a 90º/120º included angle. Both sides of thru holes should be chamfered. This chamfer should be considered "handwork" and need not be verified or documented by QA.

Drilled hole depth refers to the depth of the full diameter of the hole. If a hole is designated as "DRILL POINT OK" the exact drill point geometry is at the machinist's discretion unless specified on the drawing. If the material under the drill point is less than 0.10 inch the work piece will be supported to prevent dimpling.

Counterbores and spot faces are flat to within .005 from I.D. to O.D. and have .012 MAX corner fillets. Machinists must verify that the tool meets the requirement. QA need not verify or

document this except at the machinist's request.

Bolt clearance holes will have locational tolerance given at maximum material condition.

Bolt hole patterns located to a true position equal to or greater than .010 inch need not be verified or documented by QA. True position will be given at maximum material condition.

8.2 Reamed holes intended to be produced with standard tools and absent any specified tolerance or surface finish requirement on the drawing will meet the following:

Reamed holes will be given as four-place decimals and will be noted as reamed holes. For instance as .2500 diameter ream thru or .1875 diameter ream to .250 deep.

Reamed holes with multiple features such as counterbores, spot faces, chamfers and pilot holes shall be considered one feature for all locational tolerance.

Unless otherwise specified reamed holes will have a 32 surface finish or better.

Unless otherwise specified the size tolerances of reamed holes are:

.0313 - .5000 inch	+ .0002/-.0000 inch
.5313 - .6250 inch	+ .0003/-.0000 inch
.6563 - 1.5000 inch	+ .0005/-.0001 inch

Titleblock tolerances and inspection requirements apply to all other hole dimensions.

Reamed hole depth refers to the depth of the full diameter of the hole. If a hole is designated as "DRILL POINT OK" the exact drill point geometry is at the machinist's discretion unless specified on the drawing. If the material under the drill point is less than 0.10 inch thick the work piece will be fully supported.

Reamed holes located to a true position equal to or greater than .010 inch need not be verified or documented by QA. Match drilled reamed holes only require functional inspection.

Reamed holes shall include a lead-in chamfer as described for drilled holes.

8.3 Straight threaded holes intended to be produced with standard tools and absent any specifications to the contrary will meet the following:

All threaded holes must carry a complete description including thread size, pitch, series, minimum full thread depth and minor diameter depth.

Helicoil threaded holes must carry a complete description including nominal thread size, pitch, insert length and minor diameter depth, all preceded by the wording "Tap for Helicoil" and followed by the helicoil type.

EXAMPLE 8.3a .250–28UNF-2B THD
.38 min full THD depth
Minor diameter thru

EXAMPLE 8.3b TAP for.250–28UNF Helicoil
2.0 dia long, Type 3591-4CN
Minor diameter thru

EXAMPLE 8.3c .250–28UNF-3B THD
.43 min full THD depth
Minor diameter as required
DRILL POINT OK

EXAMPLE 8.3d TAP for.250–28UNF Helicoil
2.0 dia long, Type 1191-4CN
Minor diameter as required
FLAT BOTTOM DRILL

If the minor diameter depth is left to the machinist's discretion the minor diameter depth may be 7 thread pitches below the minimum full thread depth, or until the material thickness below the drill point is 0.10 inch.

Blind tapped holes that cannot include a minor diameter depth of 7 thread pitches will be specified. Specifying "BOTTOMING TAP REQ'D" implies a minor diameter depth 3 thread pitches below the minimum full thread depth. Flat bottom drills may also be specified. Any other relationship between the minor diameter depth and the minimum full thread depth must be specified be giving the minor diameter depth.

EXAMPLE 8.3e .250–28UNF-3B THD
.43 min full THD depth
BOTTOMING TAP REQ'D
Minor diameter as required
DRILL POINT OK

EXAMPLE 8.3f TAP for.250–28UNF Helicoil
2.0 dia long, Type 1191-4CN
Minor dia to depth shown
FLAT BOTTOM DRILL

If the material thickness under the drill point is less than 0.10 inch thick the work piece will be fully supported when drilled to prevent dimpling.

Where practical, fine threads will be used for all thread sizes. When coarse threads are specified oversized tap drills to 80% of thread height may be used in materials of hardness equal to or greater than RC30.

EXAMPLE 8.3g A 316L S.S. part (RC32) requires a .375 – 16UNC -2B thread. The nominal minor diameter is 0.316 inch. Nominal thread height is .0295 inch, 80% of nominal height is .024 inch so the minor diameter could be increased to .328 inch.

8.4 Straight threaded hole inspection requirements

Threaded holes with multiple features shall be considered one feature for all locational tolerance.

Threaded holes located to a true position equal to or greater than .010 inch only require functional inspection for location.

Class 2 threads only require functional inspection for size, pitch and thread depth. For multiple identical features functional inspection of one tapped hole and visual confirmation of all others is sufficient.

Class 3 threads require 100% inspection for size, pitch and thread depth. For multiple identical features on a common bolt pattern (either linear or circular) sampling of 25% of the holes per pattern (minimum of one) and visual confirmation of all others is sufficient.

8.5 AS5202/MS-33649 ports intended to be produced with standard tools and absent any specifications to the contrary will meet the following:

All AS5202/MS-33649 ports must carry a description that includes the following:

Port size, thread size, pitch, series and class, spot face depth, full thread depth, and minor diameter depth. All other port features including spot-face, counterbore, and o-ring chamfers are the responsibility of the machinist unless otherwise specified. Designating a drawing feature as an MS-33649 port is sufficient. A detailed pictorial representation is not required.

EXAMPLE 8.5 A part requires an MS-33649-05 or AS5202-05 port. It may be shown as a simple tapped hole. The drawing callout requires;

Port per MS-33649-05 or AS5202-05
.500 – 20UNJF-3B THD
.45 Min full thread depth
Minor diameter to depth shown
Spot face to clean up
Implied in this description are
A .750 diameter spot face, 125 surface finish flat within .005 I.D. to O.D.
A .640/.625 diameter, 120º included angle lead-in chamfer with 32 surface finish
A .522/.517 diameter counterbore .090/.075 deep with 32 surface finish and
A 50º/40º thread lead-in chamfer at the bottom of the counterbore
All of which are produced by the standard porting tool.

Minor diameter depths should extend 7 thread pitches below the required full thread depth. If this is not possible the drawing will note the minor diameter requirements. "BOTTOMING TAP" designations apply as given above.

Ports will only be functionally inspected. Visual inspection of spot-face and o-ring counterbore finishes are required.

Port location tolerances equal to or greater than .010 inch will be functionally inspected.

8.6 SAE J1926/1 ports intended to be produced with standard tools and absent any specifications to the contrary will meet the following:

All SAE J1926/1 ports must carry a description that includes the following:
Port size, thread size, pitch, series and class, spot face depth, full thread depth, and minor diameter depth. All other port features including spot-face, counterbore, and o-ring chamfers are the

responsibility of the machinist unless otherwise specified. Designating a drawing feature as an SAE J1926/1 port is sufficient. A detailed pictorial representation is not required.

EXAMPLE 8.6 A part requires an SAE J1926/1 port. It may be shown as a simple tapped hole. The drawing callout requires;

Port per SAE J1926/1 -05
.500 – 20UNF-2B THD
.55 Min full thread depth
Minor diameter to depth shown
Spot face to clean up
Implied in this description are
A .906 diameter spot face, 125 surface finish flat within .005 I.D. to O.D.
A .555/.551 diameter, 26º/22° included angle countersink .110/.094 deep
with a 32 surface finish.
A 50º/40º thread lead-in chamfer at the bottom of the countersink
All of which are produced by the standard porting tool.

Minor diameter depths should extend 7 thread pitches below the required full thread depth. If this is not possible the drawing will note the minor diameter requirements. "BOTTOMING TAP" designations apply as given above.

Ports will only be functionally inspected. Visual inspection of spot-face and o-ring countersink finishes are required.

Port location tolerances equal to or greater than .010 inch will be functionally inspected.

8.7 Tapered pipe (NPT) tapped ports intended to be produced with standard thread tapping tools per ANSI B2.1 and absent any specifications to the contrary will meet the following:

All NPT holes must carry a complete description including pipe size, thread pitch and minor diameter depth. Full thread depth is assumed to be the L1 depth per ANSI B2.1 with runout threads to the L3 depth.

For NPT threaded holes produced by thread milling or single-pointing the full thread depth shall be the L1 + L3 depth plus 2 ±0.5 full threads to compensate for the vanishing threads produced by conventional tapping.

If the minor diameter depth is left to the machinist's discretion the minor diameter depth may be 7 thread pitches below the nominal L1 thread depth, or until the material thickness below the drill point is 0.10 inch. There are no "BOTTOMING TAPS" for NPT threaded holes. The minimum practical minor diameter depth is 4.5 thread pitches below full thread depth.

A copy of the ANSI B2.1 thread chart is included with this document.

8.8 Tapered pipe (NPT) threaded hole inspection requirements:

NPT threaded holes with multiple features shall be considered one feature for all locational tolerance.

NPT threaded holes located to a true position equal to or greater than .010 inch only require functional inspection for location.

All NPT threads only require functional inspection for size, pitch and thread depth

9. Radii and Chamfers

All fillet radii and corner chamfers of two decimal places to the right of the decimal point or less will be visually inspected.

All fillet radii and corner chamfers noted as "hand" operations i.e. hand break, hand work, hand blend, etc., will be visually inspected. Hand operation dimensions are intended as guide. No tolerances apply unless specified.

All radii and chamfers of three decimal places or more will be verified and documented by QA.

10. Surface Treatment

Surface finishes of 16 or less require comparative inspection. Rougher surface finishes require visual inspection.

Unless otherwise stated all dimensions are considered to be prior to surface treatments such as irridite, anodize and inlox. Surface treatments that require specific machining after application will be noted on the drawing. Treated surfaces that require post-treatment inspection will be noted on the drawing, for instance anodized surfaces will frequently require pre- and post-anodize dimensional inspection.

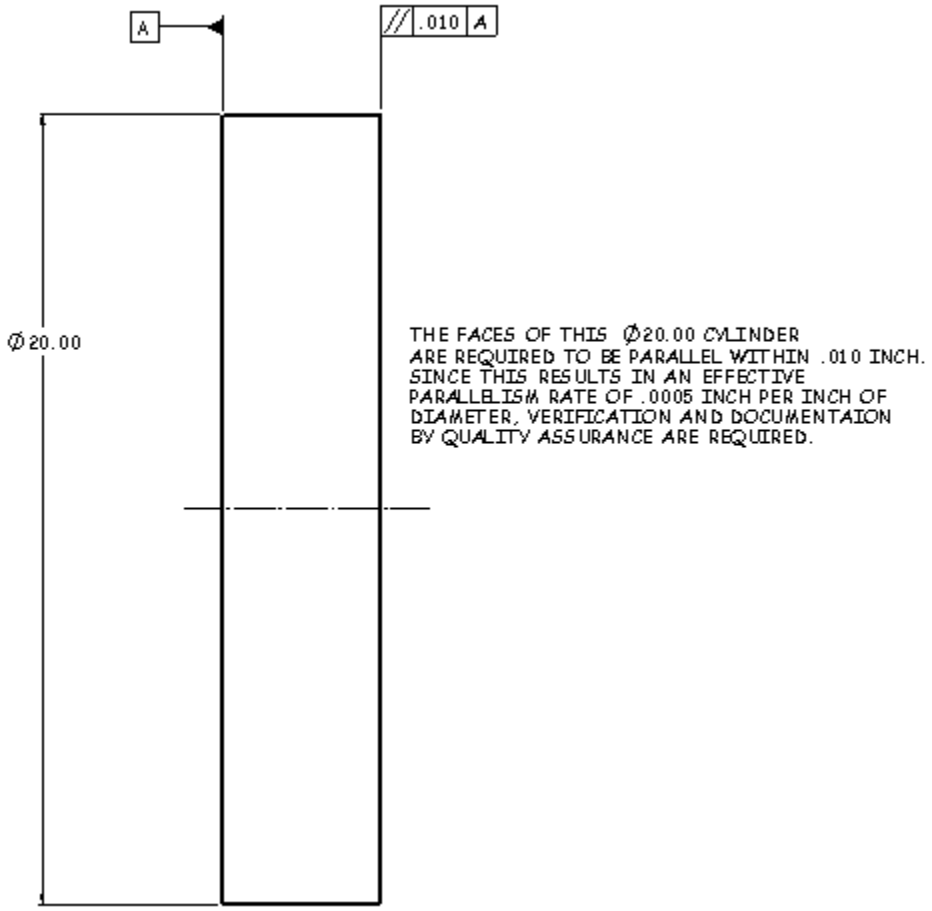
11. Stock Surfaces

Surfaces defined as "stock" or "stk" do not require any additional processing to achieve the indicated dimension. Default size tolerances do not apply to stock dimensions. It is incumbent upon the designer to know if the applicable size tolerance for the type of stock (i.e. bar, plate, tube, pipe, etc.) will produce the part required.

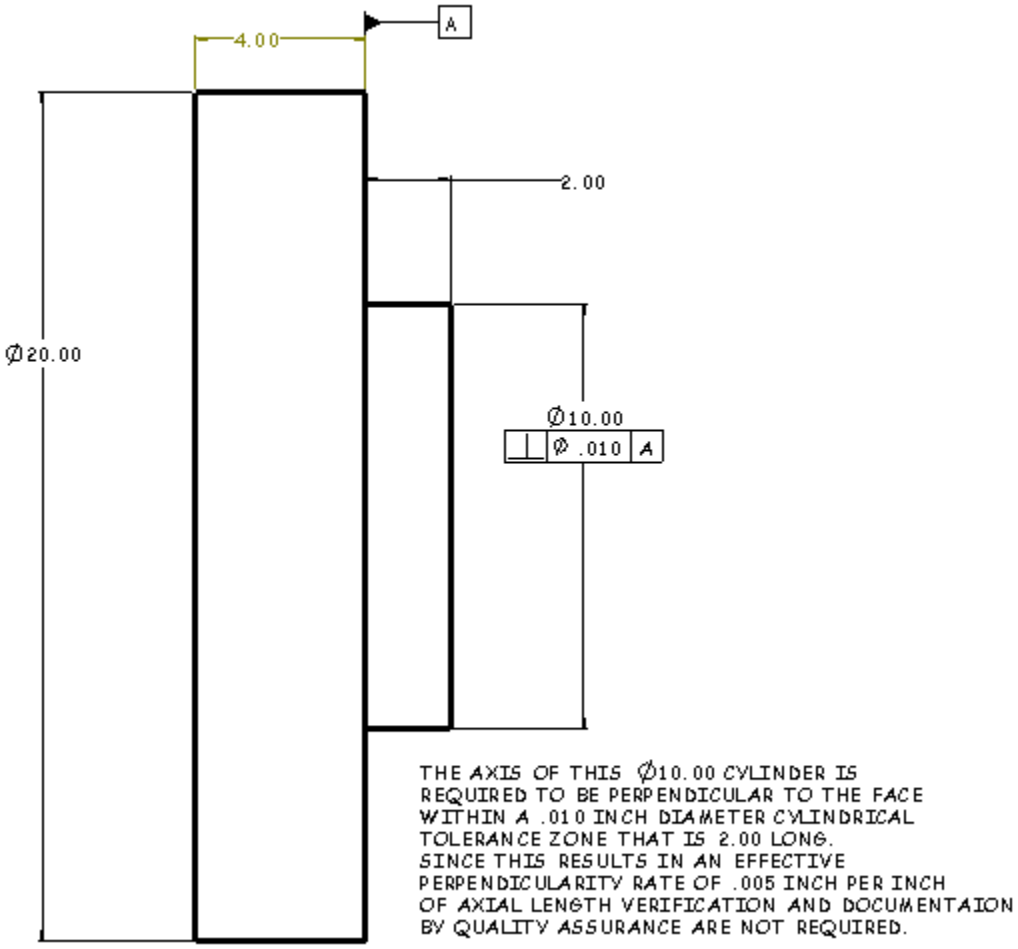
Minimum material may be removed for full clean up to improve aesthetic appearance of the part. This material removal must not exceed .030 inches per side (.060 inch on a diameter) without engineering approval. This clean up does not apply to pipe, tube, structural tubing, or structural shape such as angle, channel, I-beam, etc. Cosmetic clean up of pipe, tube and structural shapes must not remove significant amounts of material or alter the shape of the part. Abrasive cloth, abrasive grinding, bead blasting and sand blasting may be used.

12. Quality Documents

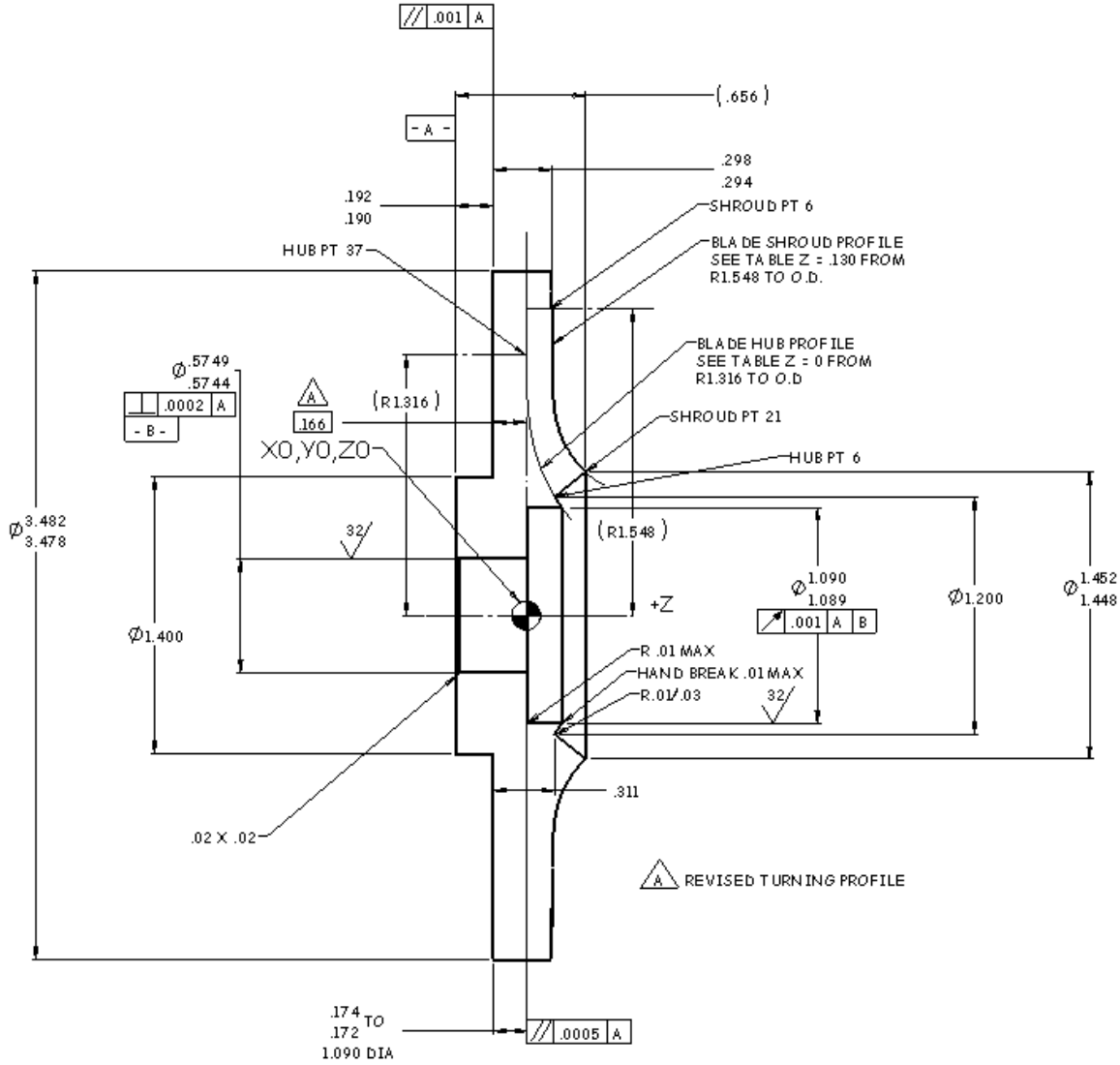
Completed routing package



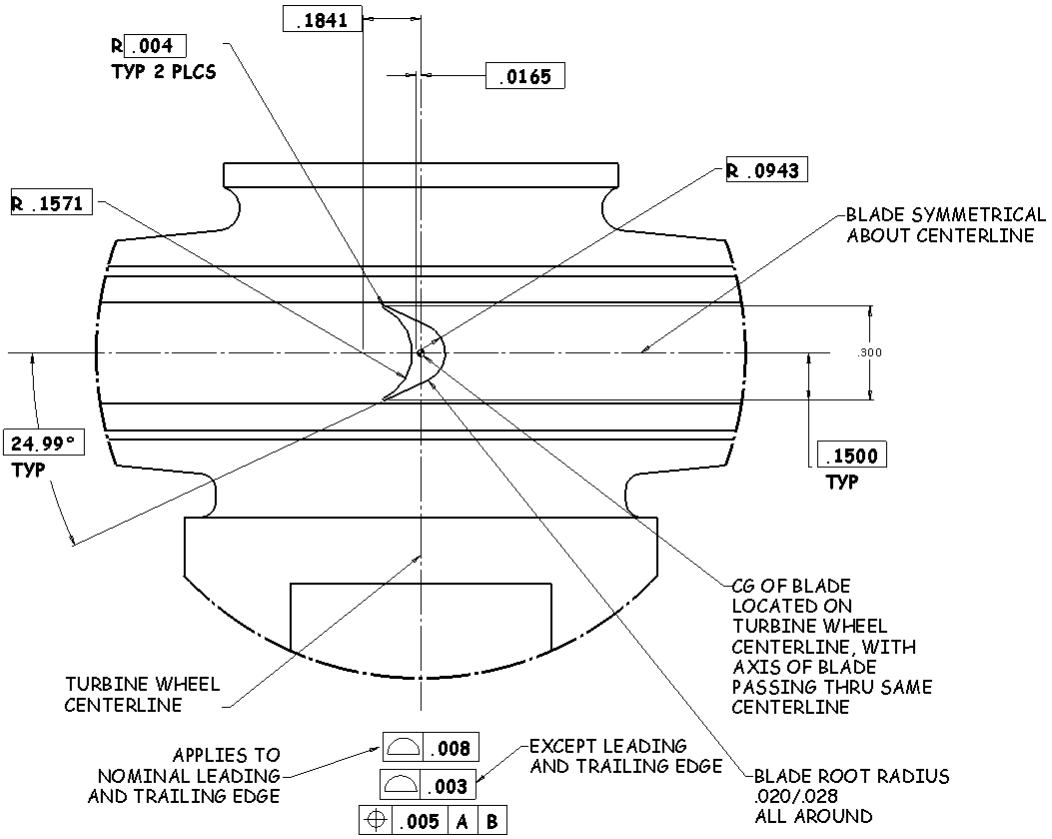
DS-703 EXAMPLE 6.7a EFFECTIVE RATE CONTROL OF PARALLELISM



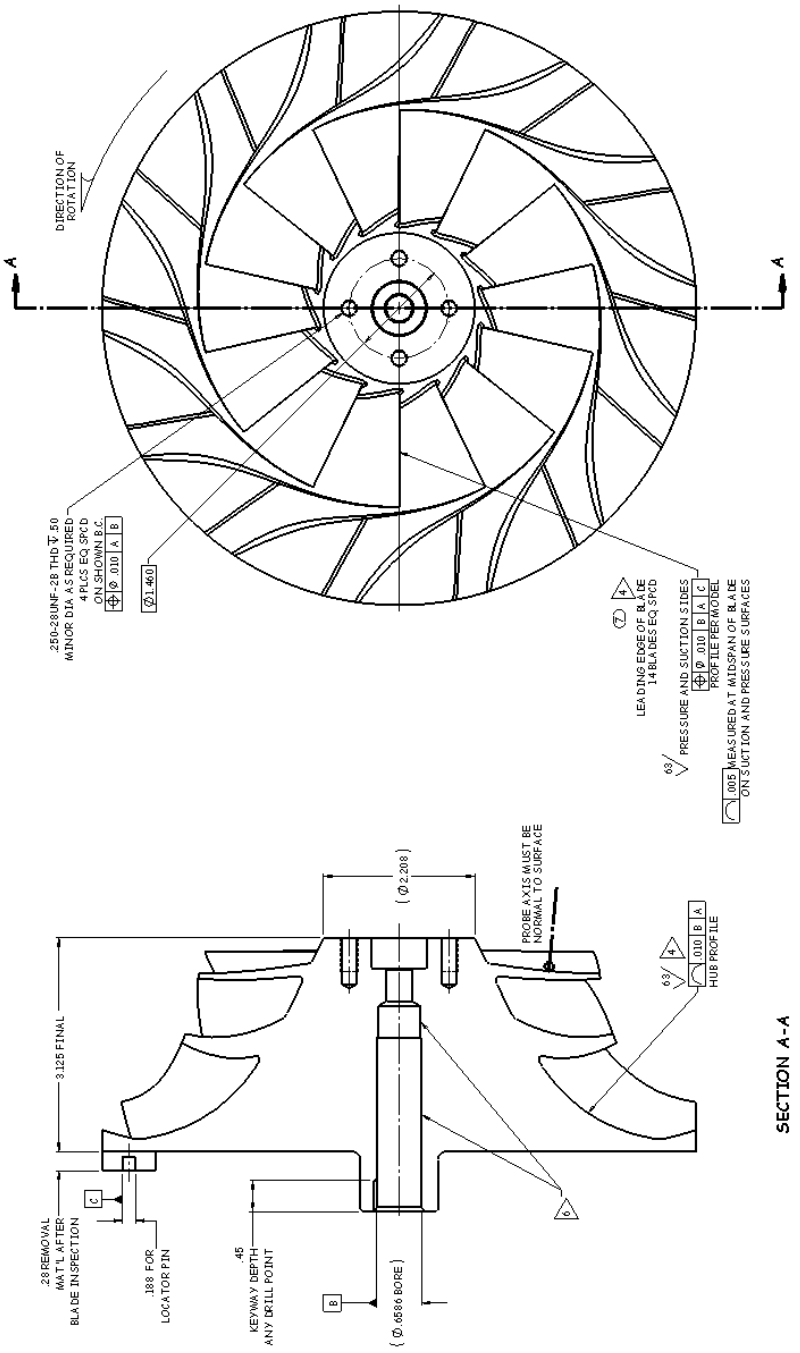
DS-703 EXAMPLE 6.7a EFFECTIVE RATE CONTROL OF PARALLELISM



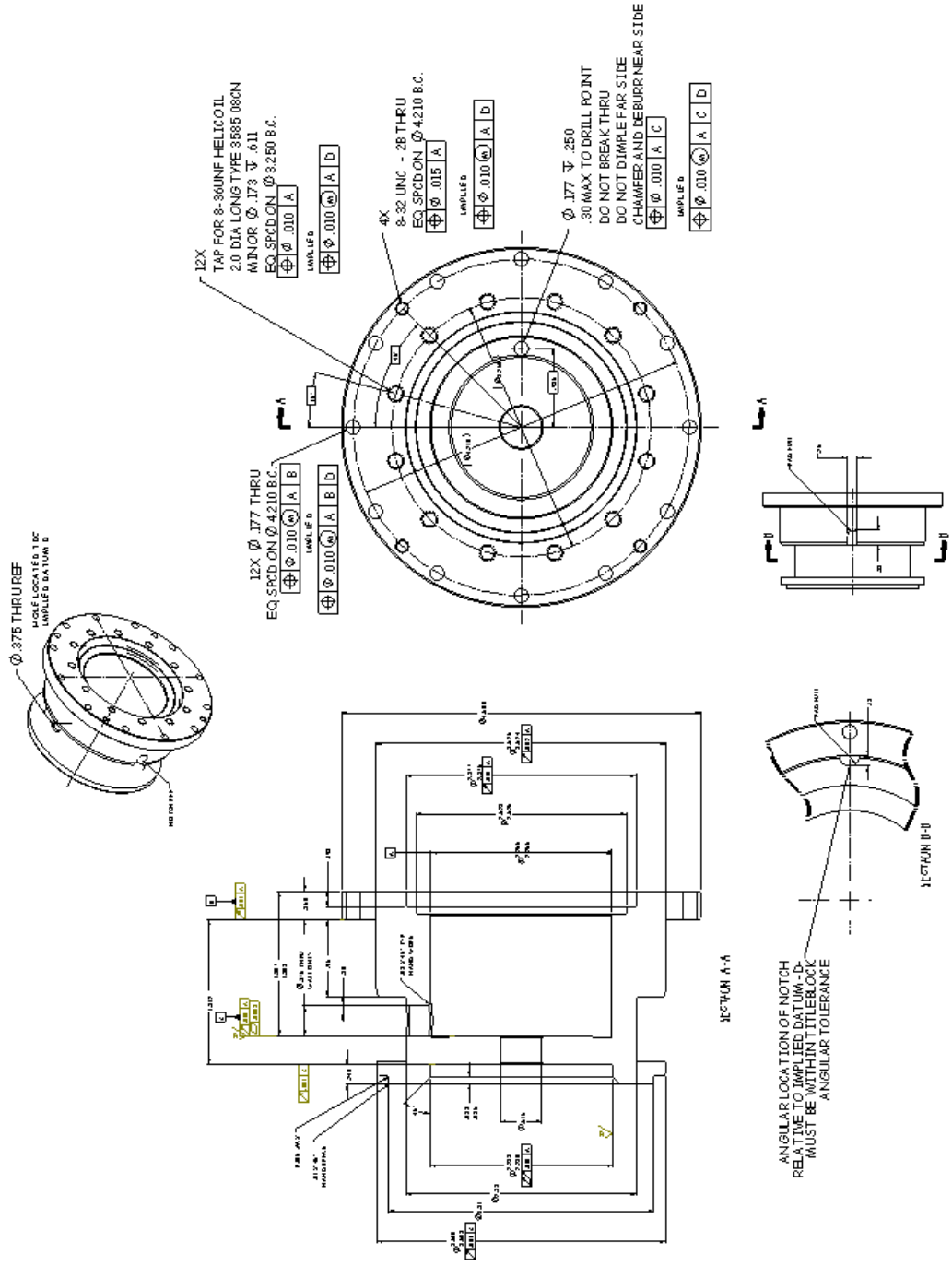
DS-703 EXAMPLE 6.10a PLANAR PROFILE INSPECTION ON CMM



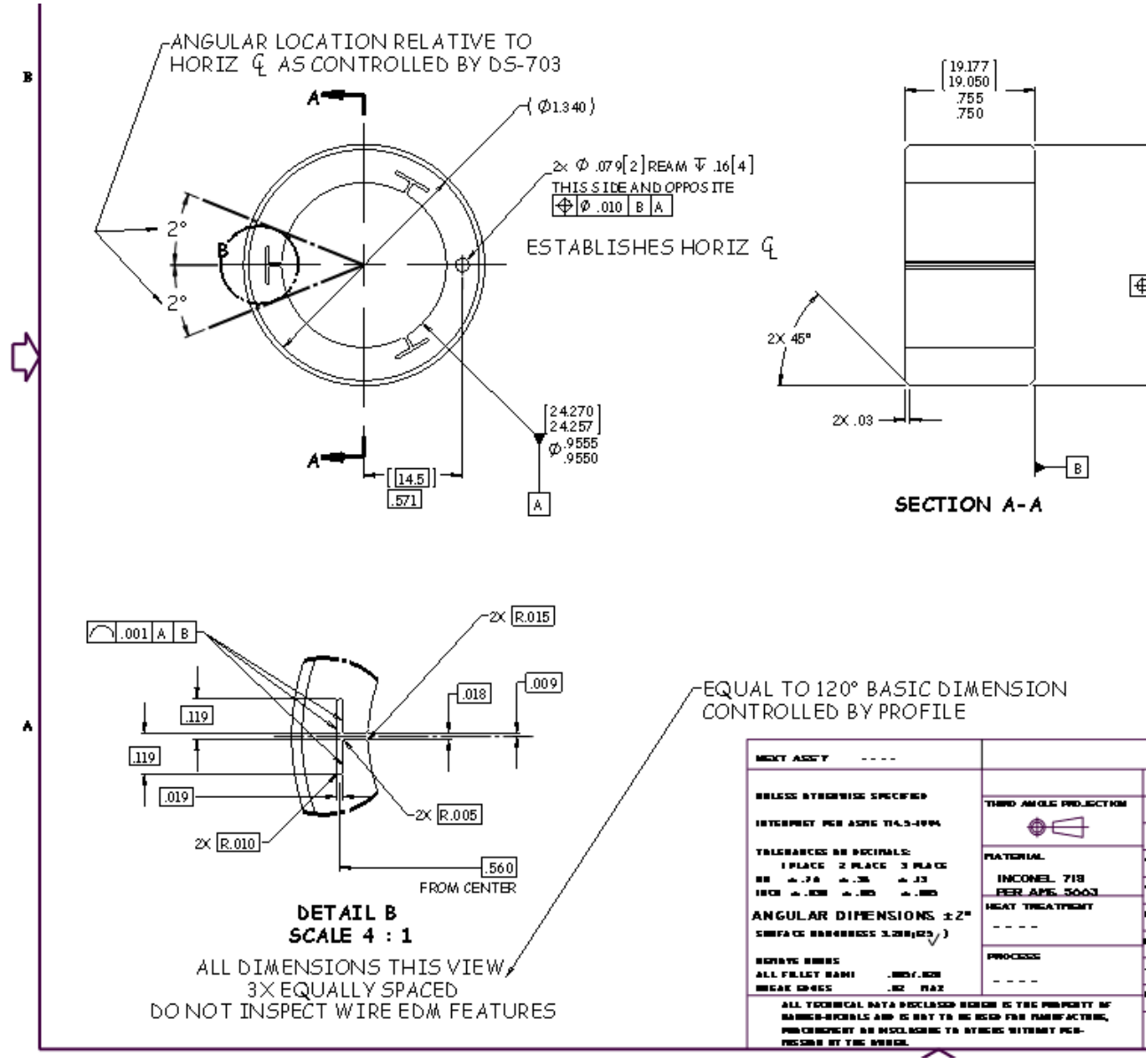
DS-703 EXAMPLE 6.10b PLANAR PROFILE INSPECTION ON CMM OR SHADOWGRAPH



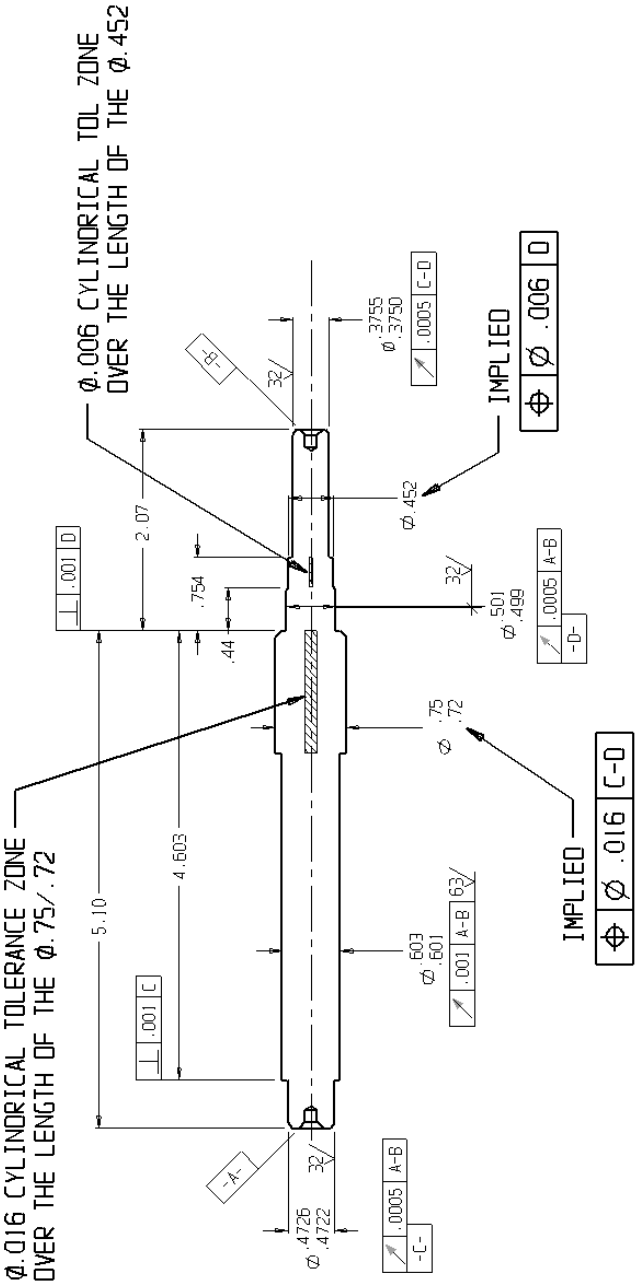
DS-703 EXAMPLE 6.10c NON-PLANAR PROFILE INSPECTION ON CMM



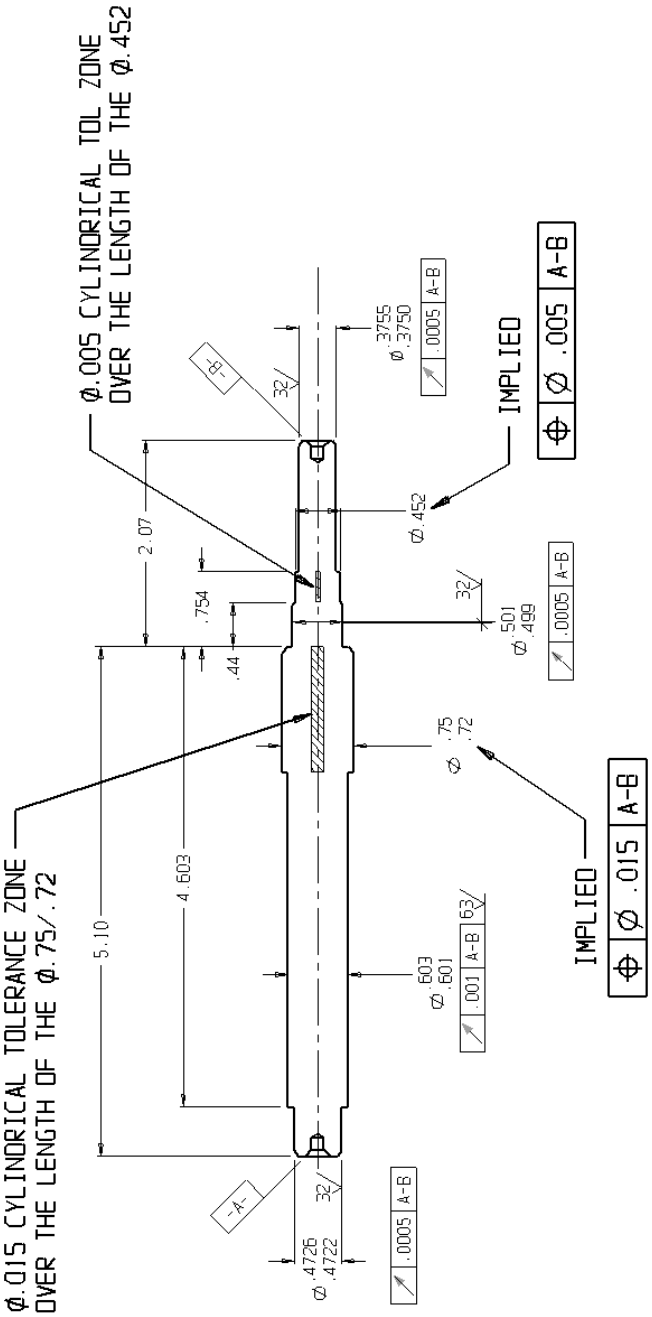
DS-703 EXAMPLE 6.11a IMPLIED ROTATIONAL LOCATION



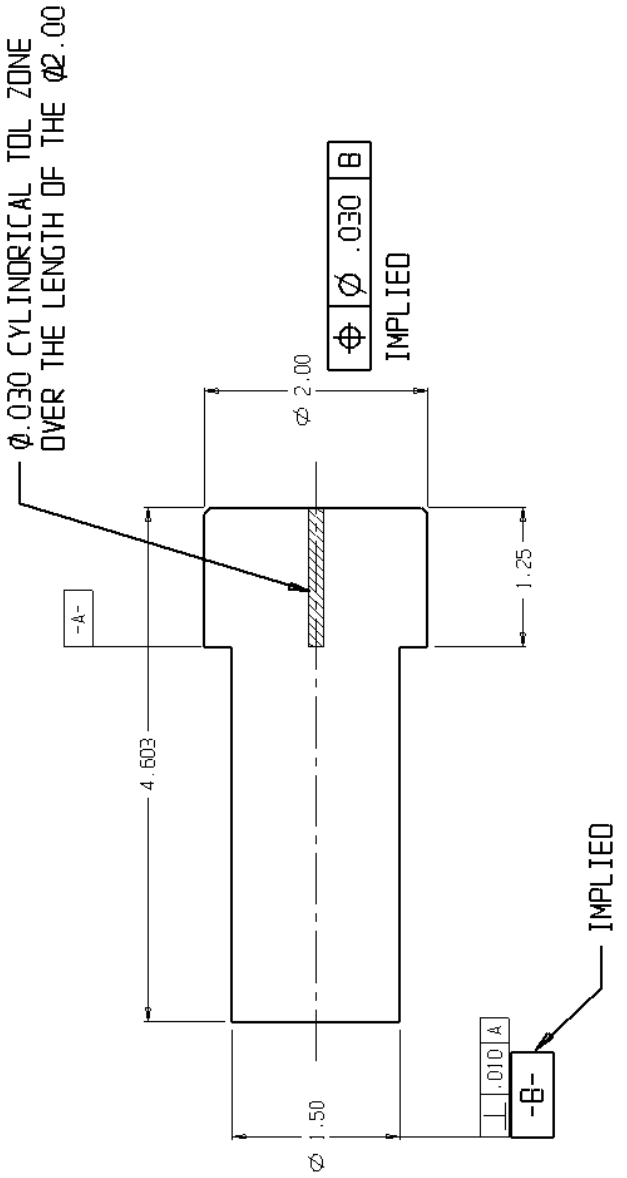
DS-703 EXAMPLE 6.11b IMPLIED ROTATIONAL LOCATION



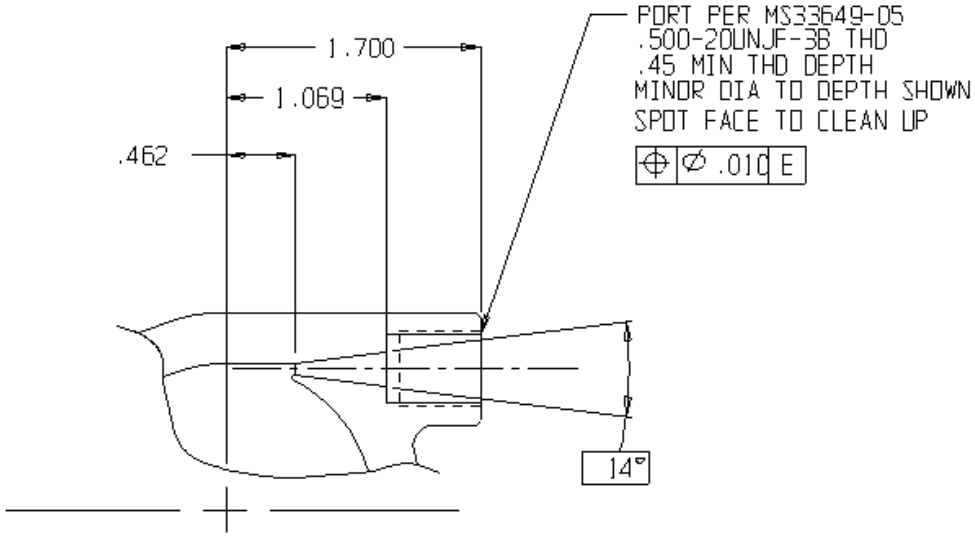
DS-703 EXAMPLE 7.1 IMPLIED LOCATIONAL TOLERANCE TO DATUM DIAMETERS



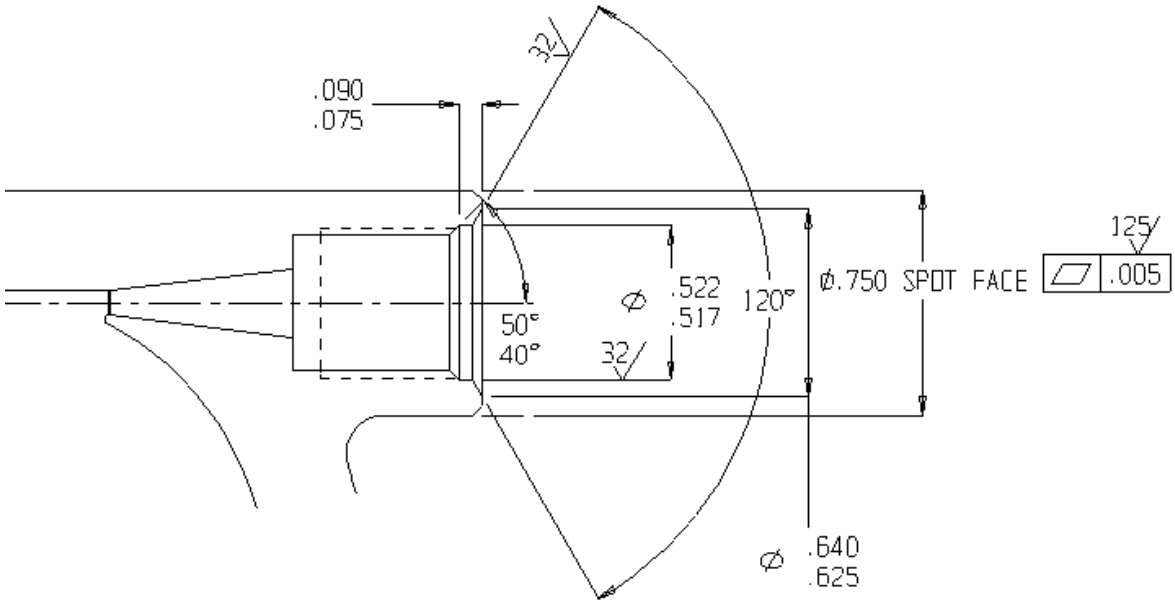
DS-703 EXAMPLE 7.2 IMPLIED LOCATIONAL TOLERANCE TO DATUM CENTERS



DS-703 EXAMPLE 7.3 IMPLIED LOCATIONAL TOLERANCE OF CYLINDERS ON A COMMON AXIS

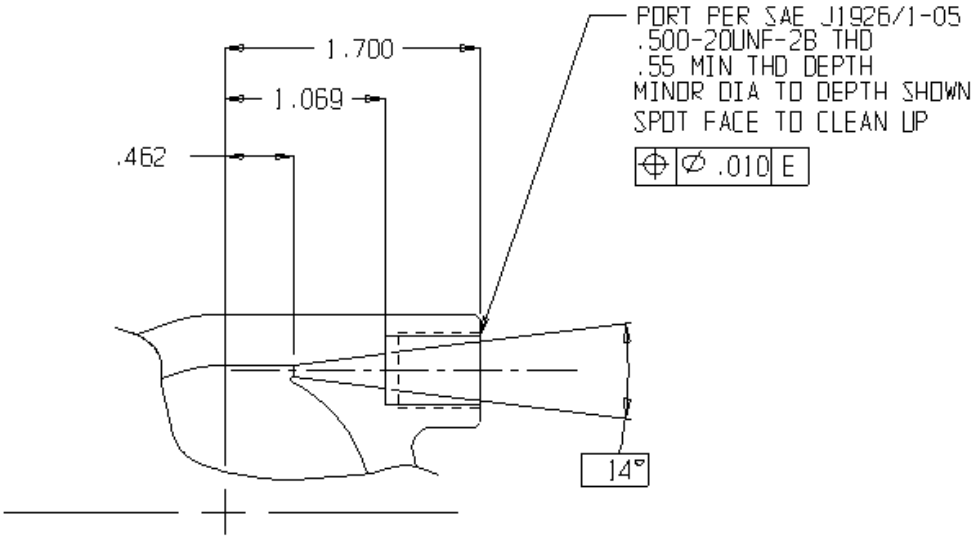


WHAT THE DRAWING SHOWS

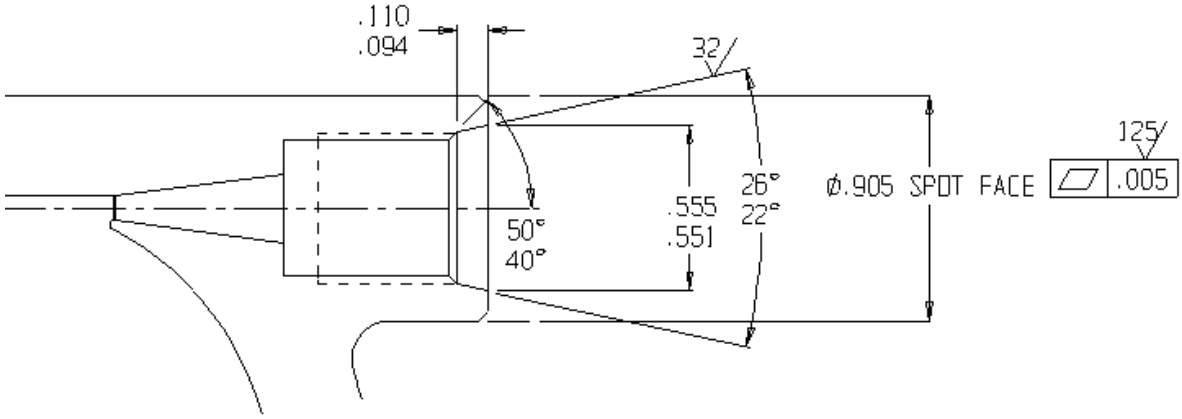


WHAT THE DRAWING IMPLIES

DS-703 EXAMPLE 8.5 MS33649 PORT IMPLIED DETAIL



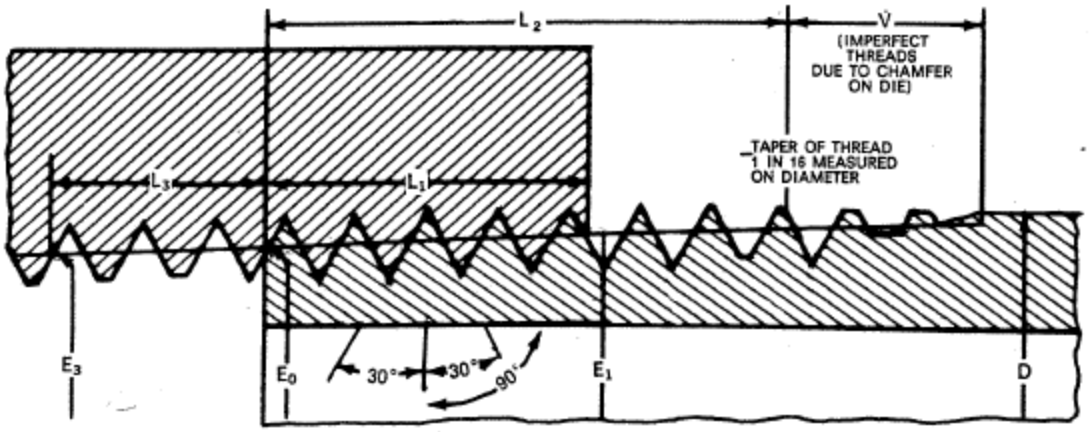
WHAT THE DRAWING SHOWS



WHAT THE DRAWING IMPLIES

DS-703 EXAMPLE 8.6 SAE J1926/1 PORT IMPLIED DETAIL

THREAD STANDARDS



BASIC DIMENSIONS, ANSI STANDARD TAPER PIPE THREAD[1]

Nominal Pipe Size	Outside Diameter of Pipe D	Number of Threads Per Inch	Pitch Diameter at Beginning of External Thread, E_0	Length[2] of Effective Thread, External L_3	Vanish Threads V	Handtight Engagement		Wrench Make-up Length for Internal Thread		Total Normal Engagement[5], $L_1 + L_3$
						Diam.[3], E_1	Length[4], L_1	Diam., E_2	Length, L_3	
1/8	.405	27	.36351	.2639	.1285	.37360	.1615	.35656	.1111	.2726
1/4	.540	18	.47739	.4018	.1928	.49163	.2278	.46697	.1667	.3945
3/8	.675	18	.61201	.4078	.1928	.62701	.240	.60160	.1667	.4067
1/2	.840	14	.75843	.5337	.2478	.77843	.320	.74504	.2143	.5343
3/4	1.050	14	.96768	.5457	.2478	.98887	.339	.95429	.2143	.5533
1	1.315	11½	1.21363	.6828	.3017	1.23863	.400	1.19733	.2609	.6609
1¼	1.660	11½	1.55713	.7068	.3017	1.58338	.420	1.54083	.2609	.6809
1½	1.900	11½	1.79609	.7235	.3017	1.82234	.420	1.77978	.2609	.6809
2	2.375	11½	2.26902	.7565	.3017	2.29627	.436	2.25272	.2609	.6969
2½	2.875	8	2.71953	1.1375	.4337	2.76216	.682	2.70391	.2500	.9320
3	3.500	8	3.34062	1.2000	.4337	3.38850	.766	3.32500	.2500	1.016
3½	4.000	8	3.83750	1.2500	.4337	3.88881	.821	3.82188	.2500	1.071
4	4.500	8	4.33438	1.3000	.4337	4.38712	.844	4.31875	.2500	1.094
5	5.563	8	5.39073	1.4063	.4337	5.44929	.937	5.37511	.2500	1.187
6	6.625	8	6.44609	1.5125	.4337	6.50597	.958	6.43047	.2500	1.208
8	8.625	8	8.43359	1.7125	.4337	8.50003	1.063	8.41797	.2500	1.313

DIMENSIONS ARE IN INCHES

[1]The basic dimensions of the ANSI Taper Pipe Thread are given in inches to four or five decimal places. While this implies a greater degree of precision than is ordinarily attained, these dimensions are the basis of gage dimensions and are so expressed for the purpose of eliminating errors in computations.

[2]Also length of plug gage.

[3]Also pitch diameter at gaging notch (handtight plane).

[4]Also length of thin ring gage and length from gaging notch to small end of plug gage.

[5]Dimensions given do not allow for variations in tapping or threading. Standard gaging tolerance equals plus or minus one thread on both male and female threads.

The above information taken from the American National Standards Institute, Inc., for Pipe Threads, ANSI-B2.1.