U.S. DEPARTMENT OF COMMERCE
National Institute of Standards and Technology

NISTIR 4744

National PDES Testbed Report Series

Comparison of ISO 10303 Part 47 Draft with ANSI and ISO Tolerancing Standards for Harmonization and Completion of Part 47

NATIONAL PDES TESTBED

NIST

U.S. DEPARTMENT OF COMMERCE
National Institute of Standards and Technology

NISTIR 4744

National PDES Testbed

Comparison of ISO 10303 Part 47 Draft with ANSI and ISO Tolerancing Standards for Harmonization and Completion of Part 47

Shaw C. Feng

U.S. DEPARTMENT OF COMMERCE
Robert A. Mosbacher, Secretary of Commerce

National Institute of Standards and Technology
John W. Lyons, Director

January 8, 1992

Contents

COMPARISON OF ISO 10303 PART 47 DRAFT WITH ANSI AND ISO TOLERANCING STANDARDS FOR HARMONIZATION AND COMPLETION OF PART 47

Shaw C. Feng
National Institute of Standards and Technology
Gaithersburg, MD 20899

This report provides an evaluation of the ISO 10303 Part 47 draft with respect to ANSI and ISO tolerancing standards. We expect that the information contained in this report will contribute toward the completion and harmonization of Part 47.

This document is the result of a comparison of the Part 47 draft with existing and developing U.S. and international dimensioning and tolerancing standards. Comparison results are summarized in tables in each section. The comparison of the Part 47 draft with other standards is made in basic tolerancing principles, size tolerance, datum and datum establishment, and geometric tolerances specified in standards. Recommendations for additions and modifications to Part 47 are at the end of each section. The objective is to have Part 47 completely cover the basic principles, concepts, and geometric tolerancing methods specified in ANSI and ISO standards.

Key Words: ANSI; DMIS; geometric dimensioning and tolerancing; geometric tolerances; harmonization; ISO; PDES; product data; standard; standards harmonization; STEP.

1 Introduction

The Standard for the Exchange of Product Model Data (STEP) is being developed as an international standard for the computer representation of product data throughout the product's life cycle [1]. One part of STEP, the Shape Tolerance Resource Model, contains the information model for dimensions and tolerances of products. The Shape Tolerance Resource Model, also known as STEP Part 47 [2], is in the form of a draft proposal within ISO/TC 184/SC 4/WG 3/Project 3. NIST has a strong interest in completing Part 47, while also ensuring that it is consistent with other U.S. and international standards.

STEP Part 47 will be used in computer integrated design and manufacturing. The purpose of harmonizing Part 47 with existing and developing standards is to ensure that Part 47 will completely cover all the tolerancing standards. The goal is that users should be able to smoothly integrate manufacturing operations that use these different standards. STEP Part 47 is critical to a number of activities in STEP, including development of Application Protocols for inspection planning and process planning. This report evaluates the content of Part 47 with respect to several national and international standards related to geometric tolerances (one of which is currently under development). These tolerancing standards are for specifying allowable shape variations of designed parts, except, one is used for the inspection of manufactured parts. These standards were selected because they all provide for the representation, in one form or another, of tolerance data.

The existing standards are ANSI Y14.5 [3], ISO 1101 [4] and its family standards, and DMIS [5]. The ISO 1101 family standards are ISO 286-1 [7], 1660 [8], 2692 [9], 5458 [10], 5459 [11], 8015 [11]. The standard under development is the Mathematical Definition of Y14.5 Dimensioning and Tolerancing Principles [6], being developed by the ASME Y14.5.1 committee. ANSI Y14.5 and the ISO 1101 family provide definitions and drawing representations for geometric tolerancing practice. Y14.5.1, based on Y14.5, will provide for rigorous and computer representable mathematical definitions of geometric tolerances. DMIS provides a vocabulary of terms for communicating tolerance data between computer aided design systems and dimensional measurement equipment. Part 47 will provide for computer representable geometric tolerances. The tolerances in Part 47 were created primarily based on ANSI Y14.5M-1982 Dimensioning and Tolerancing standard with some influence from the ISO 1101 standard.

Our work toward harmonization takes place in three phases. The first phase is to compare the Part 47 draft with all the selected standards. This work will allow us to tell whether Part 47 covers all the ANSI and ISO tolerancing specifications. The second phase is to recommend additions and modifications needed to complete Part 47. The third phase is to generate issues for the ISO/TC 184/SC 4/WG 3/Project 3, the IPO (IGES/PDES Organization) Tolerance Committee, and the Y14.5.1 committee for consideration in harmonizing the two draft standards. This report is the result of Phase I work.

In this report, tolerancing principles, tolerance definitions, and methods for datum establishment are listed and compared. Section 2 compares basic tolerancing principles, the material condition concept, and the size tolerance in Part 47 with those in other standards. Section 3 compares datum-related definitions and datum establishing methods in Part 47 with those in other standards. Section 4, 5, 6, 7 and 8 contain, respectively, comparisons of definitions of location, orientation, form, profile, and runout tolerance in Part 47 with those in other standards. The recommended actions for Part 47 to have a complete coverage of these tolerances are listed at the end of each section. Section 9 proposes that the free state variation specified in the Y14.5 standard should be added to Part 47. Section 10 contains concluding remarks. Section 11 presents all the standards cited in this report.

2 Tolerancing Principles and Size Tolerances

Standard geometric tolerances are interpreted according to several fundamental concepts and principles. They are: the size of a feature, the concept of material condition, the maximum material principle, the principle of independency, datum, and tolerance zones. Tolerances are applied to mechanical design to ensure that manufactured parts meet their functional requirements.

ANSI Y14.5 subscribes to the maximum material principle (also called Taylor principle). ISO tolerancing standards adopt both the maximum material principle and the Principle of Independency.

In this section, we review ANSI and ISO standard definitions related to size, size tolerancing, and basic tolerancing principles adopted by the two standards. We also compare size definitions in the Part 47 draft with those in ANSI and ISO standards in a table of comparison. From the comparison, we recommend additions and modifications for Part 47.

2.1 Definitions

In ANSI Y14.5, the following definitions relate to the tolerancing principle and the size:

Envelope principle:
: A boundary of perfect form at MMC of a feature is prescribed to control variations in form as well as size of the individual feature.

Maximum material condition (MMC):
: The condition in which a feature of size contains the maximum amount of material within the stated limits of size.

Least material condition (LMC):
: The condition in which a feature of size contains the least amount of material within the stated limits of size

Regardless of feature size (RFS):
: The term used to indicate that a geometric tolerance or datum reference applies at any increment of size of the feature within its size tolerance.

Feature of size:
: A feature associated with a size dimension. Size dimension can be the diameter of a cylinder, a sphere, or the width of two parallel (opposing) planes.

Feature:
: A physical portion of a part, which needs to be distinguished from the rest of the part.

Virtual condition:
: The boundary of a feature generated in the condition that the total effect of MMC combined with any applicable geometric tolerances is considered.

Actual size:
: The measured size.

Limits of size:
: The specified maximum and minimum sizes of a feature.

Dimension:

A numerical value expressed in appropriate units of measure and indicated on a drawing and in other documents along with lines, symbols, and notes to define the size or geometric characteristic, or both, of a part or part feature.

Basic dimension:

A numerical value used to describe the theoretically exact size, profile, orientation, or location of a feature or datum target. It is the basis from which permissible variations are established by tolerances on other dimensions, in notes, or in feature control frames.

The virtual condition establishes a boundary which combines the MMC effect with any applicable geometric tolerance. The surface or surfaces of the considered feature, specified at MMC, should not vary beyond this virtual condition boundary. Although the boundary of perfect form at LMC is not required, the actual size of the feature must be within the limit of the LMC size. Similarly, if a feature is specified at LMC, the virtual condition establishes a boundary which combines the effect of LMC with any applicable geometric tolerances. The surface or surfaces of the considered feature should not vary beyond this virtual condition boundary specified at LMC. Although the boundary of perfect form at MMC is not required, the actual size must be within the limit of MMC size. Specified at MMC, a feature gains geometric tolerance, such as position and orientation, when the actual size of the feature departs from MMC to LMC. Similarly, specified at LMC, a feature gains geometric tolerance when the actual size of the feature departs from LMC to MMC. The Y14.5 standard allows exceptions to the above rules through notes on the drawing, so that different kinds of variation can be controlled independently.

In ISO 286-1, 2692, and 8015, the following definitions relate to tolerancing principles and size:

Principle of Independency:

Each specified dimensional or geometrical requirement on a drawing shall be met independently, unless a particular relationship is specified. Therefore, where no relationship is specified, the geometrical tolerance applies regardless of feature size, and the two requirements of size and of geometrical tolerance are treated as being unrelated.

Envelope requirement:

The envelope of perfect form at MMC size of the feature shall not be violated. Note that the envelope requirement violates the Principle of Independency.

Maximum material principle - general:

This is a tolerancing principle which requires that the virtual condition for the toleranced feature(s) and, if indicated, the maximum material condition of perfect form for datum feature(s), shall not be violated.

Maximum material principle - applied to the tolerance feature(s):

When applied to the toleranced feature(s), the maximum material principle permits an increase in the stated geometrical tolerance when the toleranced feature concerned departs from its maximum material condition provided that the feature does not violate the virtual condition.

Maximum material principle - applied to the datum feature(s);

When the maximum material principle is applied to the datum feature(s), the datum axis or median plane may float in relation to the toleranced feature if there is a departure from the maximum material condition of the datum feature. The value of the float is equal to the departure of the mating size of the datum feature from its maximum material size.

Maximum material condition:

The state of the considered feature in which the feature is everywhere at that limit of size where the material of the feature is at its maximum.

Least material condition:

The state of the considered feature in which the feature is everywhere at that limit of size where the material of the feature is at its minimum.

Virtual condition:

The limiting boundary of perfect form permitted by the drawing data for the feature; the condition is generated by the collective effect of the maximum material size and the geometrical tolerances.

Size:

A number expressing, in a particular unit, the numerical value of a linear dimension.

Basic (Nominal) size:

The size from which the limits of size are derived by the application of the upper and lower deviations.

Limits of size:

The two extreme permissible sizes of a feature, between which the actual size should lie, the limits of size being included.

Size tolerance:

The difference between the maximum limit of size and the minimum limit of size, i.e. the difference between the upper deviation and the lower deviation.

Maximum limit of size:

The greatest permissible size of a feature.

Minimum limit of size:

The smallest permissible size of a feature.

Actual size:

The size of a feature, obtained by measurement.

Actual local size:

Any individual distance at any cross-section of a feature, i.e. any size measured between any two opposite points.

Maximum material size:

The dimension defining the maximum material condition of a feature.

Least material size:

The dimension defining the least material condition of a feature.

Virtual size:

The dimension defining the virtual condition of a feature.

Mating size for an external feature:

The dimension of the smallest perfect feature which can be circumscribed about the feature so that it just contacts the surface at the highest points.

Mating size for an internal feature:

The dimension of the largest perfect feature which can be inscribed within the feature so that it just contacts the surface at the highest points.

The effect of the Principle of Independency is to decouple the mutual dependency of size and form when controlling variations of a feature. Otherwise, for ensuring mutual dependency of size and form when controlling variations of a feature, the drawing may call for the envelope requirement and the maximum material principle according to ISO 8015.

In DMIS, sizes are defined in terms of diameter, radius, width, and angle with two limits, *uptol* and *lotol* (pp. 67,68,69,70 of DMIS v2.1). The size of an ellipse is expressed by two diameters, the long diameter and the short diameter.

In the Y14.5.1 draft, the following definitions relate to the size tolerancing:

Actual mating envelope for an external feature:

A similar perfect feature of the smallest size which can be circumscribed about the feature so that it just contacts the surface at the highest points.

Actual mating envelope for an internal feature:

A similar perfect feature of the largest size which can be inscribed within the feature so that it just contacts the surface at the highest points.

Actual mating size:

The dimension of the actual mating envelope.

In the Part 47 draft, the following entities relate to the dimension and size tolerancing:

Size dimension:

This entity include both nominal dimension and measurement of an individual feature of the shape of an object. The shape of an object has the center of symmetry, i.e. a feature of size. This is a supertype of the size characteristic dimension and size parameter dimension.

Size tolerance:

A numerical tolerance range that constrains the variation of size dimension.

Size characteristic dimension:

A size of a shape element that has dimensional characteristics to which a size dimension may be applied.

Size parameter dimension:

A type of size dimension that specifies a value of the parameter and a tolerance on the parameter. This parameter dimension specifies the nominal size of a feature.

2.2 Comparison Table

The ✓ mark used in this section means that the standard explicitly defines or otherwise uses the concept, the principle, size tolerance, or method indicated at the beginning of the row.

	Y14.5	ISO 286-1/2692/8015	Y14.5.1	DMIS	Part 47
Principle of Independency		✓			
Maximum material principle	✓	✓	✓	✓	✓
Envelope principle	✓	✓	✓	✓	
Virtual condition	✓	✓	✓	✓	
Maximum material condition	✓	✓	✓	✓	✓
Least material condition	✓	✓	✓	✓	✓
Regardless of feature size	✓	✓	✓	✓	✓
Dimension	✓	✓		✓	✓
Basic dimension	✓		✓	✓	✓

	Y14.5	ISO 286-1/2692/8015	Y14.5.1	DMIS	Part 47
Size		✓		✓	✓
Nominal (Basic) size	✓	✓	✓	✓	✓
Feature of size	✓			✓	✓
Actual size (local, measured)	✓	✓		✓	
Limits of size	✓	✓		✓	✓
Size tolerance		✓			✓
Maximum limit of size	✓	✓		✓	✓
Minimum limit of size	✓	✓		✓	✓
Maximum material size		✓			✓
Least material size		✓			✓
Virtual size		✓			
Mating size for an external feature		✓	✓		
Mating size for an internal feature		✓	✓		
Actual mating envelope for an external feature		✓	✓		
Actual mating envelope for an internal feature		✓	✓		

2.3 Recommendations for Part 47

We recommend that the following tolerancing principles and definitions be integrated into Part 47 for its harmonization with ISO 286-1, 2692, 8015 and ANSI Y14.5 standards.

- The principle of independency

  Allow independent control of feature size and feature form.

- Virtual condition

  Allow computers to generate the boundary which simultaneously controls the variation of both size and form of a feature specified at MMC (or LMC).

- Virtual size
  Indicate the size of the virtual condition boundary.

- Mating envelope for an external feature
  Define the smallest perfect form that circumscribes the feature.

- Mating size for an external feature
  Define the size of the mating envelope for an external feature.

- Mating envelope for an internal feature
  Define the largest perfect form that inscribes an internal feature.

- Mating size for an internal feature
  Define the size of the mating envelope for an internal feature.

In Part 47, a size dimension is defined as "a measure of an individual feature of the shape of an object". We recommend that Part 47 define the dimension in a manner similar to Y14.5. Occasions when a dimension applies are:

1. A numerical value specifies the distance between two point features. One of the features can be a datum feature. A point feature can be a point in space or the center of a circle or a sphere.
2. A numerical value specifies the distance between a point feature and a line feature. One of the features can be a datum feature. A line feature can be a line element or an axis of another feature.
3. A numerical value specifies the distance between a point feature and a planar feature. One can be a datum feature. The planar feature can be a plane or a median plane.
4. A numerical value specifies the distance between two line features which are parallel. One can be a datum feature.
5. A numerical value specifies the distance between a line feature and a planar feature which are parallel. One of these feature can be a datum feature.
6. A numerical value specifies the distance between two planar features which are parallel. One of these feature can be a datum feature.
7. The size of a feature. Size is being defined by the Y14.5.1 committee.

3 Datums

In this section, we review definitions of datums at any specified material condition and methods for establishing a datum and a datum reference frame.

3.1 Datum-Related Definitions

In ANSI Y14.5, datum-related definitions and datum establishing methods are as follows:

Datum:

A theoretically exact point, axis, or plane derived from the true geometric counterpart of a specified datum feature. A datum is the origin from which the location, orientation, profile, and/or runout of features are established and measured.

Datum feature:

A feature used to establish a datum point, axis, or plane. A datum feature is selected on the basis of its geometric relationship to the toleranced feature and the requirements of the design.

Datum reference frame:

Sufficient datum features are chosen to position the part in relation to a set of three mutually perpendicular planes, jointly called a datum reference frame. Only three mutually perpendicular planes can establish a datum reference frame. Other possibilities are not explored in the standard. A datum reference frame can be established by any combination (important to the design, manufactured, and measurement of a part) of points, lines, and planes.

Datum axis establishment method:

Specified at RFS- The datum axis established from an internal cylinder is the axis of the largest perfect cylinder that inscribes the feature.
or
The datum axis established from an external cylinder is the axis of the smallest perfect cylinder that circumscribes the feature.

Specified at MMC or LMC- The datum axis is the axis of the feature at its MMC (or LMC) size or its virtual condition as applicable.

Datum plane establishment methods:

The datum feature is a feature of size:

Specified at RFS- The datum plane established from an internal feature is the median plane of two parallel planes that have the maximum distance of separation and make contact with the datum feature.
or
The datum plane established from an external feature is the median plane of two parallel plans that have the minimum distance of separation and make contact with the datum feature.

Specified at MMC or LMC- The datum plane is the median plane of two parallel planes that is at the MMC (or LMC) or the virtual condition of the feature.

The datum feature is not a feature of size:

A theoretical plane that makes contact with the datum feature. If the datum feature is unstable (that is, it wobbles), the part may be oriented to an optimum position to make contact with the mating feature or a fixture.

Primary datum:

A datum plane that is established from the primary datum feature.

Secondary datum:

A datum plane that is established from the secondary datum feature and perpendicular to the primary datum plane.

Tertiary datum:

A datum plane that is established from the tertiary datum feature and angularly oriented in relation to the secondary plane, such as perpendicular to both primary and secondary datum planes.

Compound datum:

More than one datum feature is used to establish a single datum. The appropriate datum feature reference letters, separated by a dash, are entered in one compartment of the feature control frame.

Pattern of features as a datum:

Multiple features of size, such as a circular pattern of holes at MMC, may be used as a group to establish a datum when part function dictates. Individual datum axes are established at the theoretically true position of each hole which is simulated by its virtual condition.

Partial surfaces as datums:

Only part of the surface, instead of the entire surface, serves as a datum feature. The partial surface is indicated by means of a thick chained line drawn parallel to the surface profile.

Datum target:

A specified point, line, or area on a part used to establish a datum.

Equalizing datums:

Where a part configuration is such that rounded features on opposite ends are used to establish datums, pairs of datum target points or lines are indicated on part surfaces. These target points or lines coordinately dimensioned are intended for equalizing pin locations.

In ISO 5459, the datum related definitions and datum establishing methods are as follows:

Datum:

A theoretically exact geometric reference (such as axes, planes, straight lines, etc.) to which toleranced feature are related. Datums may be based on one or more datum features of a part.

Datum-system:

A group of two or more separate datums used as a combined reference for a toleranced feature.

Datum feature:

A real feature of a part (such as an edge, a surface, or a hole, etc.), which is used to establish the location of a datum.

Datum target:

A point, line, or limited area on the workpiece to be used for making contact with the manufacturing and inspection equipment, to define the required datums in order to satisfy the functional requirements.

Simulated datum feature:

A real surface of adequately precise form (such as a surface plate, a bearing, or a mandrel, etc.) contacting the datum feature(s) and used to establish the datum(s).

Establishing a datum plane:

The datum feature shall be arranged in such a way that the maximum distance between it and the simulated datum feature has the least possible value. If the datum feature is not stable with the contacting surface, suitable supports should be placed between them at a practical distance apart.

Establishing an axis of a cylinder:

The datum is the axis of the largest inscribed cylinder of a hole or the smallest circumscribed cylinder of a shaft, so located that any possible movement of the cylinder in any direction is equalized.

Establishing a datum point:

Center point of the smallest circumscribed sphere of a datum feature.

Establishing a common datum by two features:

A common datum is established simultaneously by two datum features and is indicated in the third compartment of the tolerance frame by two letters separated by a hyphen.

NOTE: The largest inscribed feature or the smallest circumscribed feature are not always uniquely found in datum features.

In Y14.5.1 draft, the datum related definitions are based on Y14.5. A new concept of datum establishment is proposed as follow:

Candidate datum set:
A candidate datum set for a datum feature is the set of all datums that can be associated with the datum feature according to RFS, MMC, and LMC.

In DMIS, a single datum and a set of datums can be specified as:

DATDEF:
Assign datum label to one or more previously measured feature(s).

DATSET:
Define and activate a datum set, or part coordinate system, and assign to it a label.

In Part 47, definitions related to datum are as follows:

Simple datum:
A datum composed of at least one and up to three dt (dimensional/tolerance) features. In the case of datum targets, up to three points on a surface can be specified to define a datum.

Dimension/Tolerance feature:
A categorization of things [sic] which may be the target or origin of dimension/tolerance. A D/T feature corresponds to the "feature" in Y14.5.

Compound datum:
A datum which has two [sic] references which define the datum: a primary and a co-reference. A compound datum is used when two datums are equally important in the specification of a tolerance.

Co-reference:
The second simple datum of a compound datum which is referenced by a tolerance.

Datum reference frame:
A collection of from one to three datums which establish a basis for measuring the toleranced entity. Each datum shall be orthogonal to the previous datums. A material condition modifier may be applied to each datum, but if the datum is not present, then the modifier shall also be absent.

3.2 Comparison Table

The ✓ mark used in this section means that the standard explicitly defines or otherwise uses the concept, datum feature, or datum establishing method indicated at the beginning of the row.

	Y14.5	ISO 5459	Y14.5.1	DMIS	Part 47
Datum	✓	✓	✓	✓	✓
Datum feature	✓	✓	✓	✓	✓
Pattern of features as a datum	✓				
Partial surfaces as datums	✓				
Equalizing datums	✓				
Datum target	✓	✓			✓
Datum reference frame	✓	✓	✓	✓	✓
Establishing datum axis - MMC, LMC	✓	✓			
Establishing datum axis - RFS	✓	✓			
Establishing datum plane - MMC, LMC	✓	✓			
Establishing datum plane - RFS	✓	✓			
Establishing datum point - MMC, LMC		✓			
Establishing datum point - RFS		✓			
Datum precedence (primary, secondary, tertiary)	✓	✓	✓	✓	✓
Simulated datum feature		✓			
Compound (Common) datum	✓	✓		✓	✓
Partial surface as datums	✓				
Candidate datum set			✓		

3.3 Recommendations for Part 47

We recommend that the following datum definitions and datum establishment methods be added to Part 47 for its harmonization with ISO 5459, ANSI Y14.5, and Y14.5.1 standards.

- Candidate datum set
  This set defines all the possible datums for establishing a datum using an unstable (wobble) datum feature.

- Datum axis establishment at MMC, LMC, and RFS
  This specification allow Part 47 to be consistent with ISO 5459 and ANSI Y14.5 standards.

- Datum plane establishment at MMC, LMC, and RFS
  This specification will make Part 47 to be consistent with ISO 5459 and ANSI Y14.5 standards.

- Datum point establishment at MMC, LMC, and RFS
  This specification will make Part 47 to have datum point establishment methods.

- Partial surface as datums
  This specification will allow Part 47 to be able to handle partial surface as datums.

- Pattern of features as a datum
  This specification will allow Part 47 to be able to handle pattern of features as a datum.

- Equalizing datums
  This specification will allow Part 47 to be able to handle equalizing datums.

We also recommend that Part 47 adopt the Y14.5 definition for a compound datum so that more than two references can be used for establishing a compound datum.

4 Location Tolerances

In this section, we review position tolerances of a single feature, position tolerances of a pattern of features, symmetry, concentricity, and coaxiality.

4.1 Location Tolerance Definitions

In ANSI Y14.5, the following definitions are related to location tolerances:

Position tolerance at RFS:

The axis of a feature (such as cylinder, cone) or the median plane of a feature (feature of width) to be located within the specified position tolerance regardless of the size of the feature.

Position tolerance at MMC or LMC:

The axis of a feature (such as cylinder, cone) or the median plane of a feature (feature of width) to be located within the specified tolerance at its MMC (or LMC). Where the actual size of the feature departs from the MMC (or LMC) to LMC (or MMC), additional position tolerance is allowed.

Composite position tolerancing (Feature pattern location tolerancing):

The composite tolerance controls both the larger tolerance of locations of the pattern of features as a group and the smaller positional tolerance for each feature within the pattern.

Projection tolerance zone:

An extended portion of positional tolerance zone for a feature (such as a hole), this projection tolerance zone passes through its mating part (such as a fastner).

Concentricity:

The condition where the axes of all cross-sectional elements of a surface of revolution are common to the axis of a datum feature. A concentricity zone whose axis coincides with a datum axis and within which all cross-sectional axes of the feature being controlled must lie.

Symmetry:

The condition in which a feature is symmetrically disposed about the center plane of a datum feature. The symmetrical relationship is controlled by specifying a positional tolerance at either MMC, LMC, or RFS.

In ISO 1101, 5458 and 8015, the fundamental position tolerancing principle is that theoretically exact dimensions and position tolerances determine the location of features, such as point, axes, and median planes, relative to each other or in relation to one or more datums. The tolerance zone is symmetrically disposed about the theoretically exact location. The maximum material principle stated in ISO 8015 applies to location tolerances. The location tolerancing methods and related definitions are:

[NOTE: In the ISO 1101 standard, t equals to the tolerance, and t_1 and t_2 are used in a bidirectional tolerancing situation where t_1 equals to the tolerance in one direction and t_2 equals to the tolerance in the other direction.]

Position tolerance of a point:

The tolerance zone is limited by a circle (or sphere) of diameter t, the center of which is in the theoretically exact position of the considered point.

Position tolerance of a line:

The tolerance is limited by one of the three types of tolerance zones:

- two parallel straight lines (in two dimensions) a distance t apart disposed symmetrically with respect to the theoretically exact position,
- a parallelpiped of sections of t_1 x t_2, the axis of which is in the theoretically exact position,
- a cylinder of diameter of t, the axis of which is in the theoretically exact position,

within one of these tolerance zones the axis of a feature or a line must lie.

Position tolerance of plane:

The tolerance zone is limited by two parallel planes a distance *t* apart and disposed symmetrically with respect to the theoretically exact position of the considered surface.

Tolerance combination:

A group of features is individually located by positional tolerancing and their pattern location is also located by positional tolerancing, each requirement shall be met independently.

Concentricity:

The tolerance zone is limited by a circle of diameter *t*, the center of which coincides with the datum point.

Coaxiality:

The tolerance zone is limited by a cylinder of diameter *t*, the axis of which coincides with the datum axis.

Symmetry:

The symmetry of a feature can be specified as:

Symmetry of a median plane:

The tolerance zone is limited by two parallel planes a distance *t* apart and disposed symmetrically to the median plane with respect to the datum axis or datum plane.

Symmetry of a line or axis:

The tolerance zone when projected in a plane is limited by two parallel straight lines a distance *t* apart and disposed symmetrically with respect to the datum axis (or a datum plane) if the tolerance is specified only in one direction.

or

The tolerance zone is limited by a parallelpiped of section t_1 x t_2, the axis of which coincides with the datum axis if the tolerance is specified in two directions perpendicular to each other.

In the Y14.5.1 draft, the position tolerancing methods are the same as in Y14.5. The shape, size, and location of position tolerance zones are currently being mathematically defined by the Y14.5.1 committee.

In DMIS, the position tolerancing methods are based on Y14.5, the following tolerances can be specified:

TOL/POS:

Specifies the position tolerance MMC, LMC, or RFS.

TOL/CORTOL:

Specifies the bidirectional position tolerance. *This position tolerance specification is not based on Y14.5, but it is consistent with ISO 1101.*

TOL/COMPOS:

Specifies the composite position tolerance at MMC, LMC, or RFS for use with patterns to assign pattern tolerance zones and individual tolerance zones.

TOL/CONCEN:

Specifies the concentricity tolerance.

In Part 47, the position tolerance, concentricity, and symmetry are defined as follows:

Position tolerance:

Position tolerance is specified at MMC, LMC, or RFS referred to datum reference frame with spherical and cylindrical tolerance zone shapes available. The datum reference frame can be composed of only one datum (the primary datum).

Concentricity:

The tolerance specifies the diameter of cylinder center on a datum point or axis within which the center or axis of toleranced circular or cylindrical feature shall lie.

Symmetry:

Symmetry tolerance defines the allowable deviation of an element of shape from a symmetric form about a point, curve, or surface. Tolerance zones are cylindrical and spherical.

4.2 Comparison Table

The ✓ mark used in this section means that the standard explicitly defines the position tolerance zone indicated at the beginning of the row.

	Y14.5	ISO 1101/5458/8015	Y14.5.1	DMIS	Part 47
Position tolerance of a point - Circular tolerance zone		✓		✓	
Position tolerance of a point - Rectangular tolerance zone					
Position tolerance of a point - Spherical tolerance zone	✓	✓			✓

	Y14.5	ISO 1101/5458/8015	Y14.5.1	DMIS	Part 47
Position tolerance of an axis or a line - Cylindrical tolerance zone	✓	✓		✓	✓
Position tolerance of an axis or a line - Two-parallel-lines tolerance zone		✓			
Position tolerance of an axis or a line - Parallelpiped tolerance zone		✓		✓	
Position tolerance of a plane or a median plane - Two-parallel-planes tolerance zone	✓	✓		✓	✓
Composite position tolerancing	✓	✓		✓	
Projection tolerance zone	✓				✓
Concentricity	✓	✓		✓	✓
Symmetry	✓	✓			✓
Dimension origin tolerance zone			✓		
Position tolerance zone defined by a cylinder			✓		
Parallelpiped position tolerance zone by two pairs of parallel planes			✓		
Position tolerance zone defined by two parallel planes			✓		

4.3 Recommendations for Part 47

We recommend the following actions for harmonizing Part 47 with ISO 5458 and ANSI Y14.5 standards.

- Add composite position tolerance

  In ISO 5458 and ANSI Y14.5, composite (combined) position tolerance of a pattern of similar features are specified. We suggest that an entity of composite position tolerance be created in Part 47.

- Add bidirectional position tolerance zone

  Specified in ISO 5458, bidirectional position tolerance zone allows designers to tolerance the position of a point or an axis bidirectionally by defining rectangular or parallelpiped tolerance zones. We suggest that this type of tolerance zone be added to the position tolerance entity in Part 47.

- Enhance symmetry tolerance

  We suggest that two-parallel-planes tolerance zone be added to the symmetry entity and two parallel planes be added to the tol_zone_shape TYPE in Part 47.

5 Orientation Tolerances

Orientation tolerances include angularity, parallelism, and perpendicularity. In this section, we review definitions of orientation tolerances and material conditions applied to them.

5.1 Orientation Tolerance Definitions

In ANSI Y14.5, the following definitions are related to orientation tolerances:

Angularity:

The condition of a surface or an axis at a specified angle (other than 90°) from a datum plane or axis.

Angularity tolerance zone:

A region bounded by two parallel planes with the specified distance apart at the specified angle (other than 90°) from a datum plane or axis, within which the surface or the axis of the considered feature must lie.

Parallelism:

The condition of a surface equidistant at all points from a datum plane or an axis equidistant along its length to a datum axis.

Parallelism tolerance zone - two parallel planes:

Two parallel planes or lines, with a specified distance apart and parallel to a datum plane or an axis, within which the line elements of the surface or axis of the considered feature must lie.

Parallelism zone - a cylinder:

A cylindrical zone whose axis is parallel to a datum axis within which the axis of the considered feature must lie.

Perpendicularity:

The condition of a surface, median plane, or axis at a right angle to the datum plane or axis.

Perpendicularity zone - two parallel planes:

A tolerance zone defined by two parallel planes perpendicular to a datum plane or axis, within which the surface or median plane of the considered feature must lie.

Perpendicularity zone - a cylinder:

A cylindrical tolerance zone perpendicular to a datum plane within which the axis of the considered feature must lie.

In ISO 1101, orientation tolerances are defined as follows:

Angularity of a line with reference to a datum line - line and datum line on the same plane:

A tolerance zone is limited by two parallel lines with a distance t apart on a cutting plane and inclined at the specified angle to the datum line.

Angularity of a line with reference to a datum line - line and datum line on different planes:

The tolerance zone is applied to the projection of the considered line on the plane containing the datum line and parallel to the considered line.

Angularity tolerance of a line with reference to a datum surface:

The tolerance zone when projected in a plane is limited by two parallel straight lines a distance t apart and inclined at the specified angle to the datum surface.

Angularity tolerance of a surface with reference to a datum line:

The tolerance zone is limited by two parallel planes a distance t apart and inclined at the specified angle to the datum line.

Angularity tolerance of a surface with reference to a datum surface:

The tolerance zone is limited by two parallel planes a distance t apart and inclined at the specified angle to the datum surface.

Parallelism tolerance of a line with reference to a datum line - two-parallel-lines zone:

The tolerance zone, when projected in a plane, is limited by two parallel straight lines a distance t apart and parallel to the datum line, if the tolerance is only specified in one direction.

Parallelism tolerance of a line with reference to a datum line - parallelpiped zone:

The tolerance zone is limited by a parallelpiped of section t_1 x t_2 and parallel to the datum line if the tolerance is specified in two planes perpendicular to each other.

Parallelism tolerance of a line with reference to a datum line - cylindrical zone:

The tolerance zone is defined by a cylinder of diameter t parallel to the datum line.

Parallelism tolerance of a line with reference to a datum surface:

The tolerance zone is limited by two parallel planes a distance t apart and parallel to the datum surface.

Parallelism tolerance of a surface with reference to a datum line:

The tolerance zone is limited by two parallel planes a distance t apart and parallel to the datum line.

Parallelism tolerance of a surface with reference to a datum surface:

The tolerance zone is limited by two parallel planes a distance t apart and parallel to the datum surface.

Perpendicularity tolerance of a line with reference to a datum line:

The tolerance zone when projected in a plane is limited by two parallel straight lines with distance t apart and perpendicular to the datum line.

Perpendicularity tolerance of a line with reference to a datum surface - two-parallel-lines zone:

The tolerance zone when projected in a plane is limited by two parallel straight lines a distance t apart and perpendicular to the datum plane if the tolerance is specified only in one direction.

Perpendicularity tolerance of a line with reference to a datum surface - parallelpiped zone:

The tolerance zone is limited by a parallelpiped of section t_1 x t_2 and perpendicular to the datum plane if the tolerance is specified in two directions perpendicular to each other.

Perpendicularity tolerance of a line with reference to a datum surface - cylindrical zone:

The tolerance zone is limited by a cylinder of diameter t perpendicular to the datum plane.

Perpendicularity tolerance of a surface with reference to a datum line:

The tolerance zone is limited by two parallel planes with a distance t apart and perpendicular to the datum line.

Perpendicularity tolerance of a surface with reference to a datum surface:

The tolerance zone is limited by two parallel planes with a distance t apart and perpendicular to the datum surface.

In the current Y14.5.1 draft, angularity, perpendicularity, and parallelism tolerances are all included in orientation tolerance. The orientation tolerance zones are defined in a similar manner as in Y14.5. There are three types of zones, as follows:

- Orientation tolerance zone bounded by two parallel planes
- Orientation tolerance zone bounded by a cylinder
- Orientation tolerance zone bounded by two parallel lines.

In DMIS, the orientation tolerances are specified by the angle between two features, angle between a feature and a datum, parallelism, and perpendicularity as follows:

TOL/ANGLB:

Specifies an angle between two features.

TOL/ANGLR:

Specifies the angular relationship between a feature and a datum and the tolerance zone, within which the feature must lie, is bounded by two parallel lines, two parallel planes, or a cylinder.

TOL/PERP:

Specifies the perpendicularity tolerance of a zone, within which the feature must lie, bounded by two parallel lines, two parallel planes, or a cylinder.

TOL/PARLEL:

Specifies the parallelism tolerance zone, within which the feature must lie, bounded by two parallel lines, two parallel planes, or a cylinder parallel to the datum.

TOL/DISTB:

specifies a distance and the distance tolerance between two features.

In Part 47, the tolerance definitions of angularity, parallelism, and perpendicularity are similar to Y14.5 definitions. The angularity tolerance zone is bounded by two parallel planes. The parallelism tolerance zone is bounded by two parallel lines, two parallel planes, or a cylinder. The perpendicularity tolerance zone can be bounded by a cylinder (within which a line or an axis should lie) or a sphere (within which the locating point should lie).

5.2 Comparison Table

The ✓ mark used in this section means that the standard explicitly defines orientation tolerance zones indicated at the beginning of the row.

	Y14.5	ISO 1101	Y14.5.1	DMIS	Part 47
Angularity of a line to a datum line (both are on the same plane) - Two-parallel-lines tolerance zone		✓		✓	
Angularity of a line to a datum line (not on the same plane) - Two-parallel-lines tolerance zone		✓			
Angularity of a line to a datum surface - Two-parallel-planes tolerance zone		✓		✓	
Angularity of a surface to a datum line - Two-parallel-planes tolerance zone	✓	✓		✓	✓
Angularity of a surface to a datum surface - Two-parallel-planes tolerance zone	✓	✓		✓	✓
Parallelism of a line to a datum line - Two-parallel-lines tolerance zone		✓		✓	✓
Parallelism of a line to a datum line - Cylindrical tolerance zone	✓	✓		✓	✓
Parallelism of a line to a datum line - Parallelpiped tolerance zone		✓			
Parallelism of a line to a datum surface - Two-parallel-planes tolerance zone	✓	✓		✓	✓
Parallelism of a surface to a datum line - Two-parallel-planes tolerance zone	✓	✓		✓	✓
Parallelism of a surface to a datum surface - Two-parallel-planes tolerance zone	✓	✓		✓	✓

	Y14.5	ISO 1101	Y14.5.1	DMIS	Part 47
Perpendicularity of a line to a datum line - Two-parallel-lines tolerance zone		✓		✓	
Perpendicularity of a line to a datum surface - Two-parallel-lines tolerance zone		✓		✓	
Perpendicularity of a line to a datum surface - Parallelpiped tolerance zone		✓			
Perpendicularity of a line to a datum surface - Cylindrical tolerance zone	✓	✓		✓	✓
Perpendicularity of a surface to a datum line - Two-parallel-planes tolerance zone	✓	✓		✓	
Perpendicularity of a surface to a datum surface - Two-parallel-planes tolerance zone	✓	✓		✓	
Orientation tolerance zone - Two-parallel-lines			✓		
Orientation tolerance zone - Two-parallel-planes			✓		
Orientation tolerance zone - Cylindrical			✓		

5.3 Recommendations for Part 47

We recommend that the following orientation tolerance zones and orientation tolerancing methods be added to Part 47 for its harmonization with ISO 1101 and Y14.5.1.

- Angularity tolerance zone bounded by two parallel lines in the angularity entity
  This tolerance zone is specified in ISO 1101.

- Parallelism tolerance zone bounded by parallelpiped in the parallelism entity
  This tolerance zone is specified in ISO 1101.

- Perpendicularity tolerance zone bounded by two parallel lines in the perpendicularity entity
  These tolerance zone is specified in ISO 1101.

- Perpendicularity tolerance zone bounded by parallelpiped in the perpendicularity entity
  This tolerance zone is specified in ISO 1101.

- Perpendicularity tolerance zone bounded by two parallel planes in the perpendicularity entity
  This tolerance zone is specified in ISO 1101.

- Orientation tolerancing methods
  These methods specified in Y14.5.1. We recommend that a new entity be created in Part 47 for the orientation tolerancing methods.

6 Form Tolerances

Form tolerances include straightness, flatness, circularity (roundness), and cylindricity. A form tolerance is used as a refinement of the size tolerance. Therefore, the form tolerance of a feature should be less than the size tolerance of that feature. Form tolerances usually do not depend on the actual size of feature except that the straightness of an axis or the flatness of a median plane are subject to material condition.

6.1 Form Tolerance Definitions

In Y14.5, the definitions related to form tolerances are as follows:

Straightness:
: A condition where an element of a surface or an axis of a feature is straight. The straightness tolerance zone within which a line element on a surface must lie is bounded by two parallel lines, with a specified width, on a cutting plane that contains the line element. The straightness tolerance zone of an axis is bounded by a cylinder, with a specified size, within which the axis should lie.

Flatness:
: A condition of a surface having all elements in one plane. The flatness zone is bounded by two parallel planes, with a specified distance, within which a surface should lie.

Circularity:
: A condition of a surface of revolution where for a cylinder or cone, all points of the surface intersected by any plane perpendicular to a common axis are equidistant from that axis, for a sphere, all points of the surface intersected by any plane passing through a common center are equidistant from that center.

Cylindricity:

A condition of a surface of revolution in which all points of the surface are equidistant from a common axis.

In ISO 1101, the definitions related to form tolerances are as follows:

Straightness tolerance defined by two parallel lines:

The tolerance zone when projected in a plane is limited by two parallel straight lines a distance *t* apart.

Straightness tolerance defined by a parallelpiped:

The tolerance zone is limited by a parallelpiped of section t_1 x t_2 if the tolerance is specified in two directions perpendicular to each other.

Straightness tolerance defined by a cylinder:

The tolerance zone is limited by a cylinder of diameter *t*.

Flatness tolerance:

The tolerance zone is limited by two parallel planes a distance *t* apart.

Circularity tolerance:

The tolerance zone in the considered plane [NOTE: not defined] is limited by two concentric circles a distance *t* apart.

Cylindricity tolerance:

The tolerance zone is limited by two coaxial cylinders a distance *t* apart.

In current Y14.5.1 draft, the mathematical definitions for straightness, flatness, and cylindricity tolerances are defined. The circularity tolerance is defined in the same manner as defined in Y14.5.

In DMIS, similar to Y14.5, form tolerances are specified as follows:

TOL/CIRLTY:

Specifies the circularity.

TOL/CYLCTY:

Specifies the cylindricity.

TOL/FLAT:

Specifies the flatness.

TOL/STRGHT:

Specifies the straightness of tolerance zones bounded by two parallel lines with a specified width, two parallel planes (for the center plane) with a specified separation, or a cylinder with a specified diameter.

In Part 47 draft, straightness, circularity, cylindricity, and flatness tolerances are defined in the same manner as defined in the Y14.5, except the straightness tolerance zone can be bounded by two parallel lines with a specified width, two parallel planes with a specified separation, and a cylinder with a specified diameter.

6.2 Comparison Table

The ✓ mark used in this section means that the standard explicitly defines the form tolerance zone indicated at the beginning of the row.

	Y14.5	ISO 1101	Y14.5.1	DMIS	Part 47
Straightness - two-parallel-lines zone	✓	✓	✓	✓	✓
Straightness - parallelpiped zone		✓			
Straightness - two-parallel-planes zone		✓		✓	✓
Straightness - cylindrical zone	✓	✓	✓	✓	✓
Flatness	✓	✓	✓	✓	✓
Circularity	✓	✓	✓	✓	✓
Cylindricity	✓	✓	✓	✓	✓

6.3 Recommendations for Part 47

We recommend that the following form tolerance zone be added to Part 47 for its harmonization with ISO 1101.

- Add parallelpiped straightness tolerance zone in the straightness entity

  This tolerance zone is used for bidirectionally tolerancing an axis of a feature or a line element of a surface.

7 Profile Tolerances

Profiles are formed by projecting a three dimensional feature onto a plane or by taking cross-sections through the feature. A profile is the two dimensional view of the shape in a specified direction to an object. The profile tolerance specifies a uniform boundary along the true profile within which the elements of the surface must lie. It is used to control form, or combination of size, form, and orientation of a feature.

7.1 Profile Tolerance Definitions

In Y14.5, definitions of profile tolerance zones are as follows:

Profile of a line:

A profile of a line tolerance zone is a two dimensional zone offset from the nominal surface in the normal direction, extending along the length of the considered feature or all around of the part.

Profile of a surface:

A profile of a surface tolerance zone is a three dimensional zone offset from all points in the surface in the normal direction, extending through the length and width (or circumference) of the considered feature or features.

There are two types of profile tolerance zones: unilateral and bilateral. A unilateral tolerance zone can be either in the inside or in the outside of the feature. Profile of a surface tolerance requires reference to datums in order to provide proper orientation of the profile. With profile of a line, datums may be used to define a cross-section in the part for taking a profile. Partial profile control is allowed, i.e. a portion of the profile is controlled by an individual profile of tolerance independent of other portion of the profile.

In ISO 1101 and 1660, definitions of profile tolerance zone are as follows:

Profile of a line:

The tolerance zone is limited by two lines (curves) enveloping circles of diameter t, the centers of which are situated on a line having the true geometric form.

Profile of a surface:

The tolerance zone is limited by two surfaces enveloping spheres of diameter t, the centers of which are situated on a surface having the true geometrical form.

In Y14.5.1, the development of the nature of the function describing boundaries of the profile tolerance zone is in progress.

In DMIS, profile of a line tolerance and profile of a surface tolerance are specified as follows:

TOL/PROFL:

The profile of a line tolerance specifies a tolerance zone which is a two dimensional band, extending along the length of the feature. If the tolerance is unbounded, it is applied along the length of the feature (implies all around the part). The tolerance zone may be bounded when it is desired to apply it to only a portion of the feature.

TOL/PROFS:

The profile of a surface tolerance specifies a three dimensional tolerance zone which extends along the length and width or circumference of the feature. As with the profile of a line tolerance, it may be bounded or unbounded.

In part 47, definitions of profile tolerances are as follows.

Profile of a line:

The tolerance zone is formed by a circle, with the specified diameter, or its center moves along the design nominal curve, sweeps the region in which the feature shall lie. The tolerance zone can be applied to the inside, the outside, or both sides of a feature.

Profile of a surface:

The tolerance zone is a region bounded by ball-offset surface(s) located equally on either side or totally on one side of the design nominal feature, within the region the feature shall lie.

7.2 Comparison Table

The ✓ mark used in this section means that the standard explicitly defines the profile tolerance indicated at the beginning of the row.

	Y14.5	ISO 1660	Y14.5.1	DMIS	Part 47
Profile of a line - unilateral	✓				✓
Profile of a line - bilateral	✓	✓		✓ (assumed)	✓
Profile of a surface - unilateral	✓				✓
Profile of a surface - bilateral	✓	✓		✓ (assumed)	✓
Partial profile control	✓				

7.3 Recommendations for Part 47

In Part 47, the view direction of the profile of a line is defined by a vector. This vector also defines the normal of the cross sectional plane which locates the profile of a part. We propose that Part 47 explicitly specify that the vector must be a unit vector and a point must be added for locating the cross sectional plane in the profile entity so that the direction and the location of the cross sectional plane are clearly defined.

We recommend that the partial profile control be added to Part 47 for its harmonization with ANSI Y14.5.

8 Runout Tolerances

Runout tolerances are used to control the relationship of one or more features of a part to a datum axis. Circular runout and total runout are the two types of runout tolerance.

8.1 Runout Tolerance Definitions

In Y14.5, circular runout tolerance and total runout tolerance are defined as follows:

Circular Runout:

> Circular runout provides control of circular elements of a surface. The tolerance is applied independently at any circular measuring position as the part is rotated 360°. At any measuring position, each circular element of these surfaces must be within the specified runout tolerance when the part is rotated 360° about the datum axis. Where applied to surfaces constructed around a datum axis, circular runout may be used to control the cumulative variations of circularity and coaxiality. Where applied to surfaces constructed at right angles to the datum axis, circular runout controls wobble of a plane surface.

Total Runout:

> Total runout is applied simultaneously to all circular and profile measuring positions as the part is rotated 360°. The entire surface must lie within the specified runout tolerance zone when the part is rotated 360° about the datum axis. Where applied to surfaces constructed around a datum axis, total runout is used to control cumulative variations of circularity, straightness, coaxiality, angularity, taper, and profile of a surface. Where applied to surfaces constructed at right angles to a datum axis, total runout controls cumulative variations of perpendicularity and flatness.

In ISO 1101, circular runout tolerance and total runout tolerance are defined as follows:

Circular Runout - radial direction:

The tolerance zone is limited within any place of measurement perpendicular to the axis by two concentric circles a distance t apart, the center of which coincides with the datum axis.

Circular Runout - axial direction:

The tolerance zone is limited at any radial position by two circles a distance t apart lying in a cylinder of measurement, the axis of which coincides with the datum axis.

Circular Runout - in any direction:

The tolerance zone is limited within any cone of measurement, the axis of which coincides with the datum axis by two circles a distance t apart.

Circular Runout - in a specific direction:

The tolerance zone is limited within any cone of measurement of the specific angle, the axis of which coincides with the datum axis by two circles a distance t apart.

Total Runout - radial direction:

The tolerance zone is limited by two coaxial cylinders a distance t apart, the axes of which coincide with the datum axis.

Total Runout - axial direction:

The tolerance zone is limited by two parallel planes a distance t apart and perpendicular to the datum axis.

In Y14.5.1, circular runout tolerance and total runout tolerance are defined as follows:

[NOTE: t_0 equals to the tolerance]

Circular Runout:

Circular runout tolerance zone is a surface of revolution generated by revolving a line segment about the datum axis. The line segment is normal to the nominal surface and is of length t_0.

Total Runout:

Total runout tolerance is a volume of revolution generated by revolving an area about the datum axis. The area is defined by two lines a distance t_0 apart, normally offset from the nominal geometry and bounded on each side by the extent of the feature.

In DMIS, circular runout tolerance is specified in TOL/CRNOUT and the total runout tolerance is specified in TOL/TRNOUT as follows:

TOL/CRNOUT:

Applies to circular features which must lie within a circular area of a specified width around the datum axis at a specified cross section. The tolerance is applied independently at any circular cross-section as the part is rotated 360°. Both axial and radial direction are applicable to a feature the same manner as specified in the Y14.5.

TOL/TRNOUT:

Applies to surfaces which must lie within a tolerance zone defined by two parallel lines with a specified width, two parallel planes separated by a specified distance, or two coaxial cylinders with a specified distance. Total runout is used to control cumulative variations of a surface. Where applied to surface constructed at right angles to a datum axis, total runout controls cumulative variations of perpendicularity (wobble) and flatness (concavity or convexity).

In Part 47, circular runout tolerance and total runout tolerance are defined as follows:

Circular Runout:

Circular runout tolerance defines the maximum allowable deviation of position of a point on the toleranced feature during one complete revolution of the toleranced feature about the datum axis, without relative axial displacement of the measuring position. Circular runout can be applied in both axial and radial directions.

Total Runout:

Total runout specifies the maximum allowable deviation of position for all positions on the toleranced feature during one complete revolution of the feature about the datum axis, with relative axial displacement of the measuring position. Circular runout can be applied in both axial and radial directions.

8.2 Comparison Table

The ✓ mark used in this section means that the standard explicitly defines the runout tolerance zone indicated at the beginning of the row.

	Y14.5	ISO 1101	Y14.5.1	DMIS	Part 47
Circular Runout - radial	✓	✓		✓	✓
Circular Runout - axial	✓	✓		✓	✓
Circular Runout - in any direction		✓	✓		
Circular Runout - in a specific direction		✓			

	Y14.5	ISO 1101	Y14.5.1	DMIS	Part 47
Total Runout - radial	✓	✓	✓	✓	✓
Total Runout - axial	✓	✓		✓	✓
Total Runout - surface normal			✓		

8.3 Recommendations for Part 47

For more rigorous, more general, and computer representable definitions of circular runout and total runout, we propose new definitions as:

Circular runout:

Circular runout tolerance zone within which a circular element of a part must lie is bounded by an area generated by revolving a line segment with a specified length about the datum axis. The orientation of the line segment can be perpendicular to the datum axis (in the radial direction), parallel to the datum axis (in the axial direction), or any other orientation with respect to the datum axis.

Total runout - Adopt the definition in Y14.5.1 as:

Total runout tolerance zone within which a feature must lie is bounded by a volume generated by revolving an area about the datum axis. The area is bounded by two lines (curves) unilaterally or bilaterally generated from the normal offset of the nominal geometry extended along the axial direction.

These new definitions are compatible with ANSI Y14.5, Y14.5.1, ISO 1101, and DMIS.

9 Free State

Free state variation is used to describe the form distortion of a flexible part after removal of applied forces. These forces are primarily internal forces due to stresses resulting from manufacturing processing, the gravitational force, and the force due to assembly. In some cases, a flexible part is required to meet its tolerance requirements while the part is in free state. In other cases, a flexible part is required to meet its tolerance requirements while it is assembled with its mating part under certain conditions (usually the specified maximum force).

9.1 Free State Definitions

Free State Variation Tolerance:

Specified in Y14.5, the term FREE STATE may be placed beneath the frame to clarify a free state requirement on a drawing containing restrained feature notes or to separate a free state requirement from associated features having restrained requirements.

Specifying Tolerance on Features to be restrained:

The restraint condition under which the tolerance is applied should be noted on drawing.

There is no free state specification in ISO, DMIS, and Part 47.

9.2 Recommendations to Part 47

We recommend that an entity for free state variation control be added to Part 47 for its harmonization with ANSI Y14.5.

- Add the free state variation control into the tolerance model

  Create a new entity for specifying free state variation control.

- Allow a specification of restraint condition to a feature subject to free state

  Include a field for specifying restrain conditions of flexible features in the new free state entity.

10 Concluding Remarks

We have compared the current Part 47 draft with ISO 1101 family standards, ANSI Y14.5, ANSI Y14.5.1 draft, and DMIS. We have recommended actions for the ISO and IPO tolerance committee to consider for the completeness and harmonization of Part 47 with these standards.

Different tolerancing principles have different effects on how features are toleranced. We have proposed that the ISO Principle of Independency be added to Part 47 for the independent control of the size and the form of a feature. Various size definitions specified in ISO standards are missing from Part 47. We have proposed that these size definitions be added to Part 47 for the purpose of determining actual position, orientation, or form tolerances specified at maximum or least material conditions.

We have identified all the datum establishing methods in ISO and ANSI standards that should be included in Part 47.

We have proposed that the composite (combined) position tolerance be added to Part 47. This is necessary for tolerancing a pattern of similar features, which is missing from the current Part 47 draft. For other geometric tolerances, we have proposed that various shapes of tolerance zones specified in ISO standards be added to Part 47 for complete coverage of current geometric tolerances.

We plan to formalize these recommendations in another report and generate issues for the ISO and IPO tolerance committees and the ANSI/ASME Y14.5.1 committee for harmonization of these two standards.

11 References

[1] ISO 10303-1, *STEP Overview and Fundamentals*, Draft, ISO TC 184/SC 4/WG 6, February 1, 1991.

[2] ISO 10303-47, *Shape Variation Tolerances*, Draft, ISO TC 184/SC 4/WG 3/Project 3, version 2B, November 1990.

[3] ANSI Y14.5M-1982, *Dimensioning and Tolerancing*, American Society of Mechanical Engineers, New York, NY, 1982.

[4] ISO 1101, *Technical drawings - Geometrical tolerancing - Tolerancing of form, orientation, location and run-out - Generalities, definitions, symbols, indications on drawings*, International Organization for Standardization, 1983.

[5] DMIS, *Dimensional Measuring Interface Standard*, ANSI/CAM-I 101, version 2.1, Computer Aided Manufacturing - International Inc., Arlington, TX, 1990.

[6] ANSI Y14.5.1 Draft, *Mathematical Definition of Y14.5 Dimensioning and Tolerancing Principles*, ASME Y14.5.1 committee, April 1991.

[7] ISO 286-1, *Bases of tolerances, deviations and fits*, International Organization for Standardization, 1988.

[8] ISO 1660, *Technical drawings - Dimensioning and tolerancing of profiles*, International Organization for Standardization, 1988.

[9] ISO 2692, *Technical drawings - Geometrical tolerancing - Maximum material principle*, International Organization for Standardization, 1988.

[10] ISO 5458, *Technical drawings - Geometrical tolerancing - Positional tolerancing*, International Organization for Standardization, 1987.

[11] ISO 5459, *Technical drawings - Geometrical tolerancing - Datums and datum-systems for geometrical tolerances*, International Organization for Standardization, 1981.

[12] ISO 8015, *Technical drawings - Fundamental tolerancing principle*, International Organization for Standardization, 1985.

NIST-114A
(REV. 3-90)

U.S. DEPARTMENT OF COMMERCE
NATIONAL INSTITUTE OF STANDARDS AND TECHNOLOGY

BIBLIOGRAPHIC DATA SHEET

1. PUBLICATION OR REPORT NUMBER
NISTIR 4744

2. PERFORMING ORGANIZATION REPORT NUMBER

3. PUBLICATION DATE
JANUARY 1992

4. TITLE AND SUBTITLE

Comparison of ISO 10303 Part 47 Draft With ANSI and ISO Tolerancing Standards for Harmonization and Completion of Part 47

5. AUTHOR(S)

Shaw C. Feng

6. PERFORMING ORGANIZATION (IF JOINT OR OTHER THAN NIST, SEE INSTRUCTIONS)

U.S. DEPARTMENT OF COMMERCE
NATIONAL INSTITUTE OF STANDARDS AND TECHNOLOGY
GAITHERSBURG, MD 20899

7. CONTRACT/GRANT NUMBER

8. TYPE OF REPORT AND PERIOD COVERED

9. SPONSORING ORGANIZATION NAME AND COMPLETE ADDRESS (STREET, CITY, STATE, ZIP)

National Institute of Standards and Technology

10. SUPPLEMENTARY NOTES

11. ABSTRACT (A 200-WORD OR LESS FACTUAL SUMMARY OF MOST SIGNIFICANT INFORMATION. IF DOCUMENT INCLUDES A SIGNIFICANT BIBLIOGRAPHY OR LITERATURE SURVEY, MENTION IT HERE.)

This report provides an evaluation of the ISO 10303 Part 47 draft with respect to ANSI and ISO tolerancing standards. We expect that the information contained in this report will contribute toward the completion and harmonization of Part 47.

This document is the result of a comparison of the Part 47 draft with existing and developing U.S. and international dimensioning and tolerancing standards. Comparison results are summarized in tables in each section. The comparison of the Part 47 draft with other standards is made in basic tolerancing principles, size tolerance, datum and datum establishment, and geometric tolerances specified in standards. Recommendations for additions and modifications of Part47 are at the end of each section. The objective is to have Part 47 completely cover the basic principles, concepts, and geometric tolerancing methods specified in ANSI and ISO standards.

12. KEY WORDS (6 TO 12 ENTRIES; ALPHABETICAL ORDER; CAPITALIZE ONLY PROPER NAMES; AND SEPARATE KEY WORDS BY SEMICOLONS)

ANSI; DMIS; geometric dimensioning and tolerancing; geometric tolerances; harmonization; ISO; PDES; product data; standard; standards harmonization; STEP

13. AVAILABILITY

- [X] UNLIMITED
- [ ] FOR OFFICIAL DISTRIBUTION. DO NOT RELEASE TO NATIONAL TECHNICAL INFORMATION SERVICE (NTIS).
- [ ] ORDER FROM SUPERINTENDENT OF DOCUMENTS, U.S. GOVERNMENT PRINTING OFFICE, WASHINGTON, DC 20402.
- [X] ORDER FROM NATIONAL TECHNICAL INFORMATION SERVICE (NTIS), SPRINGFIELD, VA 22161.

14. NUMBER OF PRINTED PAGES
40

15. PRICE
A03

ELECTRONIC FORM