PDHonline Course M347 (5 PDH)

Geometric Dimensioning & Tolerancing (GD&T) and Design For Six Sigma (DFSS)

Instructor: Robert P. Jackson, PE

2012

PDH Online | PDH Center

5272 Meadow Estates Drive
Fairfax, VA 22030-6658
Phone & Fax: 703-988-0088
www.PDHonline.org
www.PDHcenter.com

An Approved Continuing Education Provider

TABLE OF CONTENTS

PART I

INTRODUCTION	5
DESIGN FOR SIX SIGMA (DFSS)	5
DFSS TOOLS	7
IDENTIFY 0R DEFINE	8
IMPORTANCE OF SIX SIGMA GOALS	9
MEASURE AND ANALZSE	11
DESIGN	12
APPLICATION SOFTWARE	12
DESIGN TO OPTIMIZE	13
VALIDATE	14

PART II

GEOMETRIC DIMENSIONING AND TOLERANCING (GD&T)	14
GD&T STANDARDS	15
BENEFITS	16
RULES	17
SYMBOLS	18
MODIFYING TOLERANCES SYMBOLS	19
GD&T EXAMPLES	19
FEATURE CONTROL FRAME	23
READING BASIC SYMBOLS	25

PART III

DFSS & GD&T	28
TOLERANCE ALLOCATION	28
STATISTICAL ALLOCATION	30
EXAMPLES	32

LIST OF FIGURES

Figure 1	Focus for Six Sigma (DFSS)	7
Figure 2	Reliability Goal Setting	9
Figure 3	Process Capability Between 3 Sigma and 6 Sigma	10
Figure 4	DFSS Process	14
Figure 5	Tolerance Classes	19
Figure 6	Spline Assembly	20
Figure 7	Handle Using GD&T	20
Figure 8	Linear Dimensioning	21
Figure 9	Collar (GD&T)	22
Figure 10	Collar in English Only	23
Figure 11	Feature Control Frame	23
Figure 12	Flatness, Runout, Perpendicularity	26
Figure 13	MMC, LMC	26
Figure 14	Basic Dimension	27
Figure 15	Steps for Evaluating "Gap" Allocation	31
Figure 16	GE Lower Drawer	32
Figure 17	Loop Diagram	33
Figure 18	Motor Assembly	36
Figure 19	Refrigerator Door Trim	36
Figure 20	Refrigerator Door Trim Enlargement	37

LIST OF TABLES

Table 1	Differences Between Six Sigma and DFSS	6
Table 2	Six Sigma vs DPMO & RTY	7
Table 3	DFSS Tools	8
Table 4	GD&T Characteristics	24
Table 5	Modifying Tolerance Symbols	25
Table 6	Standard Deviation vs Process	29
Table 7	Standard Deviation vs Process	29

Table 8	Tolerance Allocation Spreadsheet	30
Table 9	Dimensions and Mean Values	32

CONCLUSIONS: 38

REFERENCES: 39

Geometric Dimensioning & Tolerancing (GD&T) and Design For Six Sigma (DFSS)

Robert P. Jackson, P.E.

INTRODUCTION:

This course is basically intended to address two distinct, but related, areas of engineering design: 1.) DESIGN FOR SIX SIGMA **(DFSS)** and 2.) GEOMETRIC DIMENSIONING AND TOLERANCING **(GD&T).** Both concepts have been viable approaches to design and detailing for some years and both are extremely valuable and useful tools for the practicing engineer. **DFSS** is a **statistical** method of design that can serve as a **predictive tool** to greatly improve quality control, if used properly and consistently. **GD&T** is a well accepted methodology of detailing the characteristics, dimensions and tolerances of a component or assembly of components. The GD&T methodology is prescribed by ASME / ANSI Standards Y 14.5M-1994 and Y14.5-2009. This course uses the tenants of DFSS **AND** GD&T to fully define a mechanical component, or assembly of components, so that no more than 3.4 defects per one million parts will result when in use. DFSS and GD&T are usually taught as separate subjects but certainly complement each other as far as design tools. It is much more difficult to achieve six sigma (6σ) results without using the GD&T approach. By using standard linear dimensioning instead of GD&T, huge errors can be made that leave room for doubt when designing tools and dies for fabrication. This will become apparent as we address GD&T. For this reason, I am structuring the course to include, and integrate, both methodologies. I would like to state that the treatment of DFSS and GD&T will be somewhat general and not in depth as far as mathematical modeling, which sometimes accompanies courses of this nature. There are excellent texts available on both subjects but none that I have found integrating both disciplines. The combination of these two is definitely a logical presentation for "blue-collar," goal-oriented, working engineers and engineering managers.

The course is divided into four distinct divisions; i.e. 1.) DFSS Survey, 2.) GD&T Survey, 3.) Problem solving that shows how DFSS and GD&T interact and compliment each other **AND** 4.) Teachable QUIZ. Please note that I have chosen to construct the ***Quiz*** at the end of the course to be a learning experience. Several of the questions have descriptive information important to understanding the basic tenants of Six Sigma and GD&T. ***This descriptive information may not be in the body of the text itself.***

I hope to achieve an interest that will provide impetus for engineers and engineering managers to adopt both disciplines for their companies. I would also state that by using DFSS, GD&T **AND** the tenants of Reliability Engineering and Reliability Testing, a product can be designed and manufactured to satisfy the most critical end user, for either consumer or commercial products.

PART I

DESIGN FOR SIX SIGMA (DFSS):

DFSS is the application of Six Sigma principles to the design of components, subassemblies, completed products and their manufacturing and support processes. Six Sigma, by definition, focuses on the production phase of a product. DFSS focuses on the research, design and development phase of a project and is truly a design tool that can and should be used by the design engineer to meet consumer expectations and demands. **The goal of DFSS is to implement the Six Sigma methodologies as early in the product or service life cycle as possible, thereby guaranteeing the maximum return-on-investment (ROI).** The methods used to insure rigor in both processes are somewhat different. Six Sigma uses the **DMAIC** approach (Define, **M**easure, **A**nalyze, **I**mprove, **C**ontrol). DFSS uses DMADV (Define, **M**easure, **A**nalyze, **D**esign and **V**erify). Please note that DFSS strives to meet the same goals as Six Sigma, that being no more than 3.4 defects per million. In this respect, the end results are the same. The table below will give a very brief description of the differences between the two technologies.

DIFFERENCES BETWEEN SIX SIGMA AND DFSS

SIX SIGMA	DFSS
DMAIC: Define, Measure, Analyze, Improve, Control	**DMADV:** Define, Measure, Analyze, Design, Validate **DMADOV:** Define, Measure, Analyze, Design, Optimize and Validate
Looks at existing processes and fixes problems.	Focuses on the up-front design of the product and processes.
More Reactive	More proactive.
Dollar benefits obtained from 6σ can be quantified quickly.	Benefits are more difficult to quantify and tend to be more long-term. It can take six to twelve months after launch of the new product before you will obtain proper accounting on the impact.
Product performance accomplished by "build and test."	Product performance modeled and simulated.
Performance and producibility problems fixed after Product in use.	Designed for robust performance and producibility.
Quality tested	Quality "designed into" the product.

Table 1: Differences Between 6σ & DFSS

With DFSS you are designing quality into the component or product from the very start and hopefully eliminating waste by **minimizing manufacturing variation** before it happens. This approach allows for correcting problems up-front and can significantly reduce the costs of redesign and testing. It is also a great way to meet customer demands by establishing measurable goals. DFSS attempts to predict how the designs under consideration will behave and how to correct for manufacturing variation prior to the first production run.

A graphical representation of the difference between DFSS and Six Sigma may be seen as follows, relative to product costs.

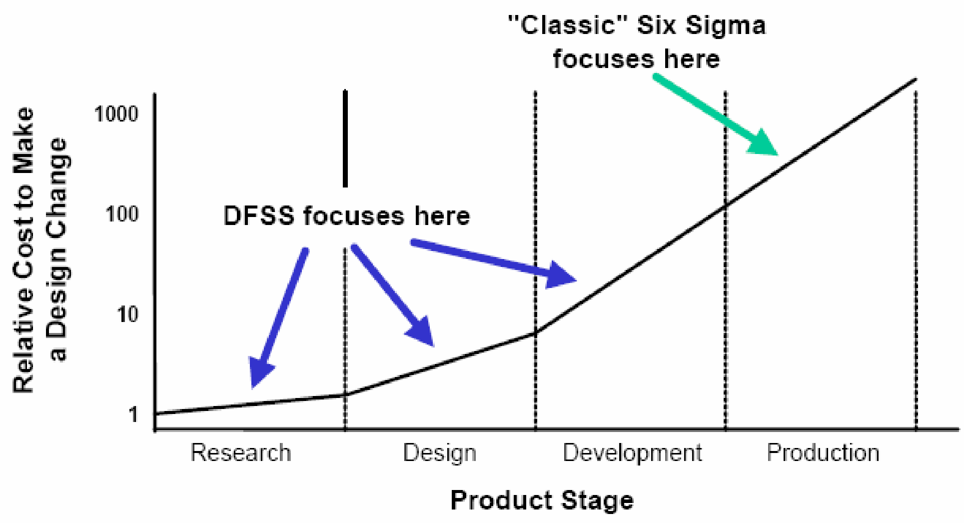

Figure 1—Focus for DFSS vs Six Sigma [3]

Again, you can see that we want to implement DFSS at the research, design and development stages and early enough in the project so that we may introduce the component or subassembly with minimal manufacturing variation. This is the point where costs are at their lowest and changes are "doable" in considerably less time and with considerably less expenditure of capital or human resources.

DFSS TOOLS:

In looking at the tools used with DFSS, the following, Table 3, represents the entire process, broken down by major action items. Please keep in mind that we "pick and choose" the best approach to Identify, Design, Optimize and Validate. All steps along the way may not be needed and would represent considerable duplication of effort. **WE WISH TO PRODUCE A MAUNFACTURED PRODUCT THAT WILL EXHIBIT NO MORE THAN 3.4 DEFECTS PER MILLION OPPORTUNITIES AND WITH THE MAXIMUM "ROLLED THORUGHPUT YIELD".** The table below will demonstrate our objectives.

σ CAPABILITY	DEFECTS PER MILLION OPPORTUNITIES	ROLLED THROUGHPUT YEILD
2σ	**308,537**	**69.1%**
3σ	**66,807**	**93.3 %**
4σ	**6,210**	**99.4 %**
5σ	**233**	**99.97 %**
6σ	**3.4**	**99.99966%**

TABLE 2—SIX SIGMA VS DPMO AND RTY

DFSS Tools

Identify	Design	Optimize	Validate
Project Charter	Assign Specifications	Histogram	Sensitivity Analysis
Strategic Plan	to CTC's	Distributional Analysis	Gap Analysis
Cross-Functional Team	Customer Interviews	Empirical Data Distribution	FMEA
Voice of the Customer	Formulate Design Concepts	Expected Value Analysis (EVA)	Fault Tree Analysis
Benchmarking	Pugh Concept Generation	Adding Noise to EVA	Control Plan
KANO's Model	TRIZ or ASIT	Non-Normal Output Distributions	PF/CE/CNX/SOP
Questionnaires	Pugh Concept Synthesis	Design of Experiments	Run/Control Charts
Focus Groups	Controlled Convergence	Multiple Response Optimization	Mistake Proofing
Interviews	FMEA	Robust Design Development	MSA
Internet Search	Fault Tree Analysis	Using S-hat Model	Reaction Plan
Historical Data	Brainstorming	Using Interaction Plots	
Quality Function Deployment	QFD	Using Contour Plots	
Pairwise Comparison	Scorecard	Parameter Design	
Design of Experiments	Transfer Function	Tolerance Allocation	
Specify CTC's	Design of Experiments	Reducing Standard Deviations	
Performance Scorecard	Deterministic Simulators	of Inputs	
Flow Charts	Confidence Intervals	Design For Manufacturability	
FMEA	Hypothesis Testing	Mistake Proofing	
Visualization	MSA	Product Capability Prediction	
	Computer Aided Design	Part, Process, and SW Scorecard	
	Computer Aided Engineering	Risk Assessment	
	High Throughput Testing	Reliability	

TABLE 3—DFSS TOOLS [3]

Let us quickly look at each action item and define the activity required before advancing on to the next phase of the DFSS process.

IDENTIFY OR DEFINE:

The most critical part of the process is the very first phase, **IDENTIFY**. This is the step that basically asks the customer, "what do you want" then that answer, usually given in generalities, is **quantified** into engineering specifications. I want it this big, this color, this weight, these features, etc becomes an engineering specification **AND** engineering drawing that can be interpreted by a model shop, a tool and die maker and quality control inspectors. The process for doing just this is called **Quality Functional Deployment or QFD**. QFD is a process that **details and ranks** the most desirable features deemed important by the consumers. Well-defined and unambiguous requirements are absolutely necessary and will lessen the probability of "false starts" and "detours" in the development process. We wish to minimize the inconsistencies between articulation of functional requirements and the definition of system requirements and parameter target values. A matrix of features is developed and rated for desirability so engineering can translate those features into "hard" specifications. The following sketch will indicate one possible approach for doing just that.

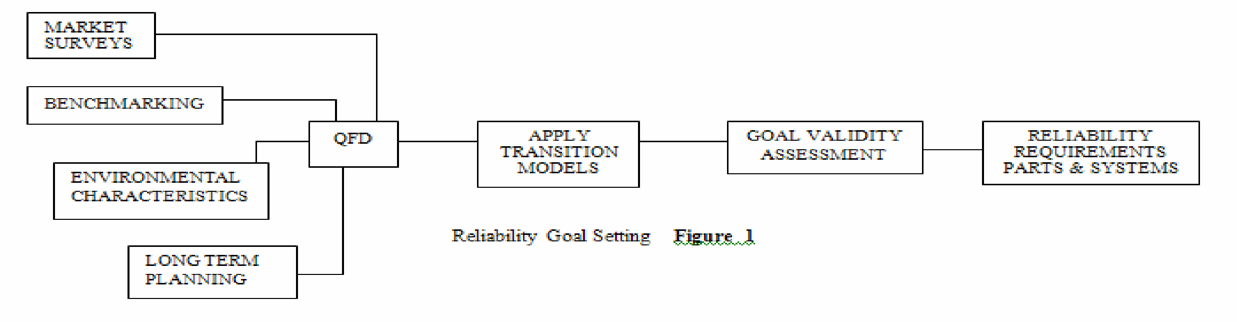

Figure 2 Reliability Goal Setting [20]

This initial phase; i.e. QFD, allows for production of the **Design Guidance** model and ultimately lets us "kick the tires" of the very first prototype. We go back to marketing, show them the drawings and / or prototype for their approval. Revisions are made based upon their desires and needs. We may wish to reconvene the focus group of potential users to get their impressions and verify that their needs and demands will be ultimately met by parts from the first production run. Next in the process is 1.) Design Confirmation, 2.) Pre-Pilot, 3.)Pilot and finally, 4.) The first production of the component or assembly. The Identify phase is also where the project team is selected and charged with the responsibility for "engineering" the component. A team leader is chosen as well as all of the team members, from CAD, model shop, evaluation testing, reliability testing, etc. It is recommended that representatives from quality and manufacturing be included so that all critical functions can have a voice in the design of what will become the final product. If the product is similar to an existing design, **benchmarking** will become necessary to evaluate current field failure rates, manufacturing difficulties, issues with tooling, problems with packaging and shipping so that these mistakes will not occur in the new product. Benchmarking can identify parts that do not survive stated reliability goals. This is a very critical part of **IDENTIFY**. One excellent tool for this exercise is **FAILURE MODE EFFECT ANALYSIS or FMEA**. FMEA is a technique that allows for cataloging of each possible failure mode of a component or an assembly during **normal** use. I highlight the word normal here. The IDENTIFY phase selects the initial **CRITICAL TO QUALITY CHARACTERISTICS (CTQs)** of the components or assemblies. These CTQs define the most critical dimensions, features and specifications of parts AND specifies that those items **MUST** be checked on first piece samples prior to any pilot or production run. Generally, a sample size of thirty parts or assemblies is required to gather enough CTQ statistically-significant data. The CTQs are periodically checked after production is initiated to make sure there is no depreciation of the manufacturing processes. Audits are definitely recommended to insure strict compliance with quality standards for these CTQs.

IMPORTANCE OF SIX SIGMA:

Before we leave IDENTIFY, I would like to restate the importance of meeting Six Sigma goals. Right now, most manufacturing companies are producing to a three-Sigma standard. Three Sigma produces a 93.32 % long-term yield. Reaching these goals gives us the following meaning of 3 Sigma---**"good"**.

- 20,000 lost articles of mail per hour.
- Unsafe drinking water for approximately fifteen minutes per day.
- 5,000 incorrect surgical operations per week.
- Two short or long landings at most major airports each day.
- 200,000 wrong drug prescriptions each year.
- No electricity for almost seven hours each month.

Even if we look at the profile of a 4 σ company we find the following characteristics

1.) Profitable and growing but with a decreasing market share.

2.) Market prices declining for certain products or product lines.

3.) Competitors increasing

4.) Has quality assurance program but deficiencies keep “slipping” through the Q.C. process.

5.) Spending 10-25% of sales dollars on repairing or reworking product before it ships. (This is crucial and a fact that will surface during benchmarking.)

6.) Unaware that **best in class** companies have similar processes that are greater than 110 times more defect free.

7.) Believes that a zero-defects goal is neither realistic nor achievable.

8.) Has 10 times the number of suppliers required to run the business. (Also critical. To “carry” a supplier can cost upwards to $10K just to maintain the database.)

9.) 5-10% of the firm’s customers are dissatisfied with product, sales or service and will not recommend that others purchase products or services. The possible reason for this can be seen in the bell-shaped curve below.

Look at the graphic. This will demonstrate the savings in going from three Sigma to six Sigma.

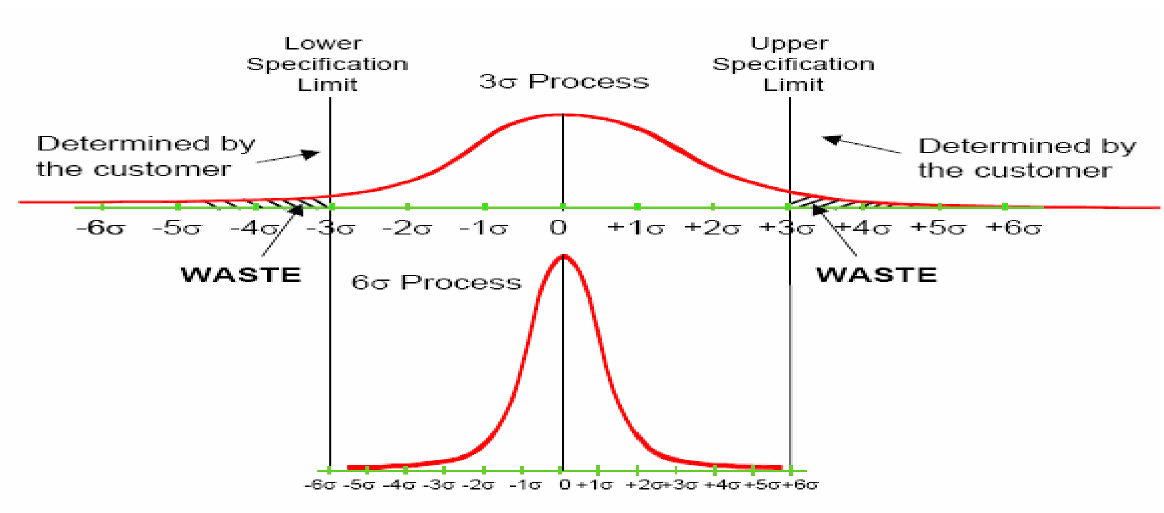

FIGURE 3—PROCESS CAPABILITY BETWEEN 3σ AND 6σ

If we can improve our process capability, we can eliminate the waste that occurs below the lower specification limit (LSL) and above the higher specification limit (HSL). Both limits are determined by the customer in the IDENTIFY (DEFINE) phase of our DMAIC process. Please notice that the lower “bell-shaped” curve represents a six sigma **CENTERED** process that yields a defect rate below 4 parts per million (PPM). This is definitely where we wish to be**. Critical to the effort is the manner in which the components are ultimately defined and detailed.** This is where application of GD&T comes in. It is critical that components be defined completely so there are no questions as to form, fit and function. Our next section will address how this is accomplished and those methods used to bring about a complete definition of the component or product.

MEASURE AND ANALYZE:

As a part of Identify, it is always a very good practice to "benchmark" existing products if they are similar to the product you are modifying and / or launching. Benchmarking is generally defined as follows: "Benchmarking is the process of continually searching for the best methods, practices, and processes and either adopting or adapting their good features and implementing them to become the best of the best". [20] I would add to this definition the need to benchmark the proposed product design for components and subassemblies. This provides a comparison of the proposed design with existing competitive designs and can highlight areas of needed improvement. This process involves the customer and obtaining reliable information on product field failures. Understanding field failures and their cause is critical improving the product performance. The following areas are critical to the DFSS benchmarking process:

- Competitive Benchmarking. (Comparisons between competition's products and your product.)
- Product Design Benchmarking (Determining the sigma value of existing and similar products now being produced by your company.)
- Process Benchmarking (Determining the sigma of the production processes.)
- Best Practices Benchmarking. (Are there better methods to fabricate and assemble your products?)

Customers can be a great aid in the benchmarking process and give us information that otherwise might be very difficult to obtain; i.e. field failure rates. This approach allows for the following:

1.) To find information that would, most likely, not surface during a typical sales call.

2.) To encourage the customer to think "out-of-the-box" and focus on specific behaviors and product designs that would make the product "best-in-class".

3.) To provide data to engineering that will promote needed changes in products. Customers have a vested interest in providing their accounts with the best and longest-lasting products. They generally know their competition.

4.) To find out customer concerns, other than price.

5.) To demonstrate long-term commitment to continuous product improvement.

6.) To encourage customers to provide data to substantiate their perceptions.

7.) To build account credibility by committing to actions which address product design complaints.

8.) To address issues with product performance and gain knowledge of field performance.

9.) To proactively define customer expectations by allowing them to define best-in-class products.

DESIGN:

The design process today is accomplished, generally, by using computer aided design (CAD) and computer modeling and simulation techniques. Computers have replaced the drafting table and manual methods for creating a drawing and designing a part. Definitely progress! With this being the case, there are many excellent software packages that provide simulation for mechanical motion, finite element analysis, impact simulation, fluid dynamics, heat transfer and other disciplines. There are statistical packages which can greatly simplify the application of Six Sigma and DFSS methodologies. Software such as these can **greatly** aid engineering efforts to reach optimum designs quickly and accurately without extended build, test, fix, build, test, fix etc. Many irritations can be omitted saving much time and money yet providing designs which meet customer expectations and manufacturing capabilities. I definitely recommend engineering departments adopt CAD, CAM and simulation software packages to speed the process from "board" to assembly line. I would like to list several excellent examples of what packages are available to accomplish design completion and produce the speed needed to remain competitive in today's engineering / manufacturing environment. Please keep in mind that these represent a fraction of what is available and new products are introduced at a very rapid pace.

APPLICATION SOFTWARE:

CAD:

- AutoDesk ProE
- Smart Draw
- Solid Works
- Turbo Cad

MOTION CONTROL: Miltibody Kinetics and Dynamics

ADAMS

DADS

AutoDesk, Inc

Sim Mechanics

T-Flex PLM

IMPLICIT FINITE ELEMENT ANALYSIS Linear and Nonlinear

- MSC.Nastran, MSC.Marc ADINA ABAQUS:Standard and Explicit
- ANYSS ProMECHANICA

EXPLICIT FINITE ELEMENT ANALYSIS: Impact Simulation

- LS-DYNA PAM=CRASH, PAM STAMP NENSTRAN AUTODESK
- RADIOSS

GENERAL COMPUTATIONAL FLUID DYNAMICS

- STAR-CD PowerFLOW FLUENT, FIDAP
- CFX-4, CFX-5

PREPROCESSING: FEA and Computational Fluid Dynamics Mesh Generation

- ICEMCFD ANSA FEMB GridPro
- Gridgen TrueGrid MSC Partran I-deas
- Altair HyperMesh

POSTPROCESSING: FEA and Computational Fluid Dynamics Results Visualization

- Altair HyperMesh ICEM CFD Visual 3.2.0 (PVS) FIELDVIEW
- I-deas EnSight FEMB
- MSC. Patran

STATISTICAL PACKAGES:

- MINITAB JMP8 BMDP S-PLUS
- X-GOBI SAS SPSS MATLAB
- XLISP-STAT ExplorN MANET Massive Datasets

HEAT TRANSFER PACKAGES:

- BuildingPhysics SAS Thermal Desktop
- RadCAD ESARAD ESTRAN

There are many smaller organizations which do not have the resources to purchase, train employees and use the above software. That should not be a great impediment. Engineering contract services can and will work to provide the needed simulations and analysis. Application of DFSS works regardless as to who is performing the process. Engineering departments are after answers and application of these methods can still save days and weeks of time when time is critical. I would add to the "mix" the need to perform necessary reliability testing to ensure the component or finished product can meet your expectations for longevity in the field under all conditions of use. You will save your company much grief if you design for an acceptable product life and infrequent "down-time" or premature replacement.

DESIGN TO OPTIMIZE:

Optimization of a design occurs as a result of reliability testing **AND** field testing prior to the first production run. ***I CAN NOT OVEREMPHESIZE THE NEED FOR RELIABILITY TESTING SO THAT APPROXIMATION OF DESIGN LIFE MAY BE SEEN.*** This is the process in which "weak" components and assemblies are identified, redesigned and retested so that all quality and reliability goals are met. This testing will ensure maximum customer satisfaction and minimize field failures. It will also identify those components which must be examined on a regular basis in order to provide for preventative maintenance and / or replacement of critical component parts. When performing these tests, it is advisable to use a "statistical software package" to facilitate the time consuming analysis that, by necessity, must accompany a study of this nature. I have used MINITAB© and Excel© for this purpose although I much prefer MINITAB due to the availability of ANOVA, "t"-tail, DOE and many other analysis tools. Also, MINITAB allows you to perform and examine histograms, boxplots, dotplots, stem-and-leaf plots, scatter plots, time-series plots and allows you to consider continuous AND descrete data. Another great feature of MINITAB is regression analysis. Regression analysis provides the formulation of transfer equations which can show cause-effect relationships resulting from DOE (design of experiments) testing. Also, regression analysis can be used to describe the mathematical relationships between the response variable (Y) and the vital "Xs". **PLEASE NOTE: The "Xs" must be continuous data and not discrete data**. The following chart will show the basic process involved with the establishment of reliability goals relative to the DMADV or DMADOV process. The process in this chart equates to the following major characteristics:

Goal Setting = Identify Measure = Develop System Model (Benchmark) Design = Design

Reliability Improvement Testing = Optimize Design = Redesign Where Needed Validate = Production Testing

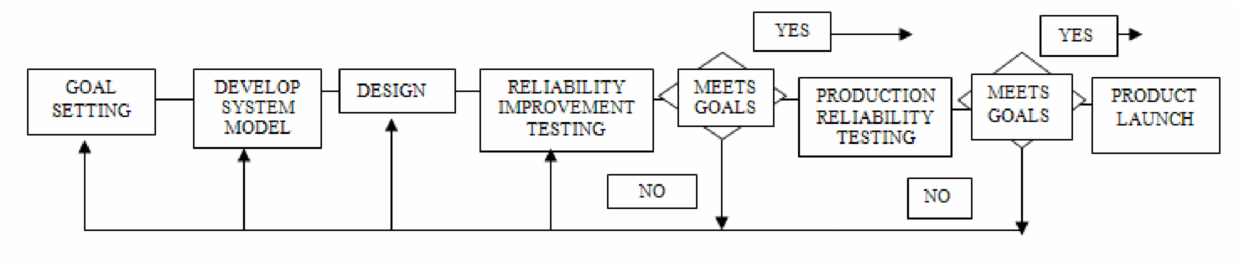

Figure 4 DFSS PROCESS [PDHonline G238]

VALIDATE:

The validation phase of the process begins after the pilot run. Please note that, in most companies, the pilot run is considered to provide "saleable" goods. This is the phase where all of the design has been accomplished, all of the approvals granted; i.e. UL, CSA, etc and tooling is in place. The validation testing occurs at this point and continues throughout the life of the product. At General Electric, we would bring products produced on our assembly lines in one per month for a laboratory audit. During the first production run, we would place a minimum of thirty (30) products into our reliability lab for life-cycle testing. They would be tested to failure to make sure that the Mean Time To Failure (MTTF) and the Mean Time Between Failure (MTBF) exceeded reliability predictions AND met our Six Sigma goals of 3.4 PPM. Of course, when the product was retired and no longer manufactured, the testing stopped. ***PLEASE NOTE: IT IS IMPORTANT TO RETAIN TEST DATA, ESPECIALLY THE AUDIT DATA, FOR THE LIFE OF THE PRODUCT.***

PART II

GEOMETRIC DIMENSIONING AND TOLERANCING (GD & T):

Geometric Dimensioning & Tolerancing (GD&T) is a set of guidelines specifically for dimensioning and detailing so a component can be correctly interpreted. The GD&T on the print provides the guidelines portrayed by that drawing for dimensional inspection. It is a **universal language** used by design engineering to **faithfully capture and transmit** the designer's intent through all activities in the product cycle. It has found the greatest application in mass production, where interchangeability of blindly selected parts is essential. Just-in-time manufacturing increases the demand for parts that absolutely must fit at assembly. A company is much less likely to have spare parts waiting in the warehouse; therefore, **PARTS SIMPLY MUST FIT TOGETHER AT ASSEMBLY.** There is no other way to ensure that the allowable variation of part geometry is adequately defined.

Dimensional variations which occur in each component part of an assembly accumulate statistically and propagate kinematically, causing the overall assembly dimensions to vary according to the number of contributing sources of variation. The resultant critical clearances and fits, which affect performance, are thus subject to variation due to tolerance stack-up of component part variations. There are three major sources of variation in assemblies as follows:

1. Dimensional (lengths and angles)
2. Geometric features (ANSI Y 14.5)
3. Kinematic (small internal adjustments)

We are primarily interested in item number 2; i.e. geometric features, and will concentrate on this issue for Part II of this course.

Understanding the cause and effect of dimensional and geometric variations is a major concern in the design and manufacture of mechanical products. Designers are essentially concerned with the following geometric dimensions and tolerances:

1. Functionality and / or the ability to be assembled

2. Tolerance Analysis: consequences of a proposed GD&T scheme (Please note that the method of manufacturing the part plays a significant role in the resulting tolerances for each component. This is discussed later in the course.)

3. Tolerance Allocation: determining how to distribute the allowable variation on the dimension of interest among all of the independent contributors.

STANDARDS:

GD&T has been adopted by the International Standards Organization (ISO) and the American National Standards Institute (ANSI) [6]. It includes all of the symbols, definitions, mathematical formulas, and application rules necessary to embody a viable engineering language so that people everywhere can read, write, understand and apply the methodology. The standard most commonly used in the Unites States to describe GD&T is ASME Y-14.5M—2009 although the following standards are "in play" on an international basis.

GD&T standards

GD&T standards for technical drawings (2D)

- ASME Y14.5M-1994 Dimensioning and Tolerancing
- ASME Y14.5.1M-1994 Mathematical Definition of Dimensioning and Tolerancing Principles
- ISO 286-1:1988 *ISO system of limits and fits — Part 1: Bases of tolerances, deviations and fits*
- ISO 286-2:1988 *ISO system of limits and fits — Part 2: Tables of standard tolerance grades and limit deviations for holes and shafts*
- ISO 1101:2005 *Geometrical Product Specifications (GPS) — Geometrical tolerancing — Tolerancing of form, orientation, location and run-out*
- ISO 5458:1998 *Geometric Product Specifications (GPS) — Geometrical tolerancing — Positional tolerancing*
- ISO 5459:1981 *Technical drawings — Geometrical tolerancing — Datums and datum-systems for geometrical tolerances*

GD&T standards for CAD systems (3D)

- ASME Y14.41-2003

  Digital Product Definition Data Practices

- ISO 16792:2006 *Technical product documentation -- Digital product definition data practices*

(Note: ISO 16792:2006 was derived from ASME Y14.41-2003 by permission of ASME.)

GD&T standards for data exchange and integration

- *ISO 10303*

  Industrial automation systems and integration — Product data representation and exchange

- *ISO 10303-47:1997* Integrated generic resource: Shape variation tolerances
- *ISO/TS 10303-1130:2006* Application module: Derived shape element
- *ISO/TS 10303-1050:2006* Application module: Dimension tolerance
- *ISO/TS 10303-1051:2006* Application module: Geometric tolerance
- *ISO/TS 10303-1052:2005* Application module: Default tolerance
- *ISO/TS 10303-1666:2006* Application module: Extended geometric tolerance
- *ISO 10303-203:2007/8* Application protocol: Configuration controlled 3D design of mechanical parts and assemblies
- *ISO 10303-210:2001* Application protocol: Electronic assembly, interconnection, and packaging design
- *ISO 10303-214:2003* Application protocol: Core data for automotive mechanical design processes
- *ISO 10303-224:2006* Application protocol: Mechanical product definition for process planning using machining features
- *ISO 10303-238:2007* Application protocol: Application interpreted model for computerized numerical controllers (STEP-NC)

BENEFITS:

If properly used, GD&T has the following significant benefits:

- Clearly defines the intent of the part and provides descriptive geometry, dimensions, orthographic projections and tolerancing. It is a precise communication tool.
- Optimally uses the part's available tolerance and allows for the use of DFSS methodology
- Increases the correlation between customer and supplier
- Provides the basis to correctly determine whether a fabricated part is acceptable or not by providing the basis for produceability.
- The use of material condition modifiers allows "bonus" tolerances which lead to great ease in assembly.
- Explicitly controls **ALL** aspects of part geometry, particularly the shape.
- GD&T is important for calculating tolerance analysis accurately.

Many companies still use "linear dimensioning" to detail their component parts and assemblies. An example of linear dimensioning is shown by Figure 8 . This can be fully acceptable but there are consequences to this process. These are as follows:

- The parts fail inspection but are still functional
- The parts pass inspection but do not work or do not work as intended
- Lack of correlation between customer/ supplier with difficulty in determining why there are issues
- Inability to make pass / fail analysis during inspection

The basic idea behind GD&T is to determine the **datum features** of the part or assembly of parts. This, of course, involves physical positions and relationships. The datums are selected as the origins for dimensioning and the application of tolerances or tolerance zones. **You MUST select functional datums.** A functional datum is simply one that uses the product features which physically locate the part relative to the final product. Using any other datum system; i.e. centerlines, will add variation in the final tolerance stack-up. [18]. Successful application of

GD&T involves concurrent design and engineering teams consisting of representatives from responsible functions; i.e. engineering, quality control, reliability testing, purchasing, etc. are chosen for this process.

RULES:

The following rules detail the fundamentals in a very concise and readable method. These rules are taken from reference [1].

- All dimensions must have a tolerance. Every feature on every manufactured part is subject to variation; therefore, the limits of allowable variation must be specified. Plus and minus tolerances may be applied directly to dimensions or applied from a general tolerance block or general note. For basic dimensions, geometric tolerances are indirectly applied in a related Feature Control Frame. The only exceptions are for dimensions marked as minimum, maximum, stock or reference.

- Dimensioning and tolerancing shall completely define the nominal geometry and allowable variation. Measurement and scaling of the drawing is not allowed except in certain cases.
- Engineering drawings define the requirements of finished (complete) parts. Every dimension and tolerance required to define the finished part shall be shown on the drawing. If additional dimensions would be helpful, but are not required, they may be marked as reference.
- Dimensions should be applied to features and arranged in such a way as to represent the function of the features.
- Descriptions of manufacturing methods should be avoided. The geometry should be described without explicitly defining the method of manufacture.
- If certain sizes are required during manufacturing but are not required in the final geometry (due to shrinkage or other causes) they should be marked as non-mandatory.
- All dimensioning and tolerancing should be arranged for maximum readability and should be applied to visible lines in true profiles.
- When geometry is normally controlled by gage sizes or by code (e.g. stock materials), the dimension(s) shall be included with the gage or code number in parentheses following or below the dimension.
- Angles of 90° are assumed when lines (including center lines) are shown at right angles, but no angular dimension is explicitly shown. (This also applies to other orthogonal angles of 0°, 180°, 270°, etc.)
- All dimensions and tolerances are valid at 20° C unless otherwise stated on the drawing.
- Unless explicitly stated, all dimensions and tolerances are valid when the item is in a free state.
- Dimensions and tolerances apply to the full length, width, and depth of a feature.
- Dimensions and tolerances only apply at the level of the drawing where they are specified. It is not mandatory that they apply at other drawing levels, unless the specifications are repeated on the higher level drawing(s).

SYMBOLS:

It is important to note that ASME Y14.5 gives the following descriptive information relative to the geometry used to define the part. We will discuss examples of how these geometric terms and symbols are used to describe form, fit and function.

SYMBOL DESCRIPTION	GEOMETRY DEFINED
Angularity	Orientation
Concentricity	Location
Cylindricity	Form

Flatness	Form
Parallelism	Orientation
Perpendicularity	Orientation
Position	Location
Profile	Profile
Profile of a Line	Profile
Circularity	Form
Runout	Runout
Straightness	Form
Symmetry	Location
Total Runout	Runout

If we use a graphic method to display the organization of the geometric characteristics, it would look something like the following:

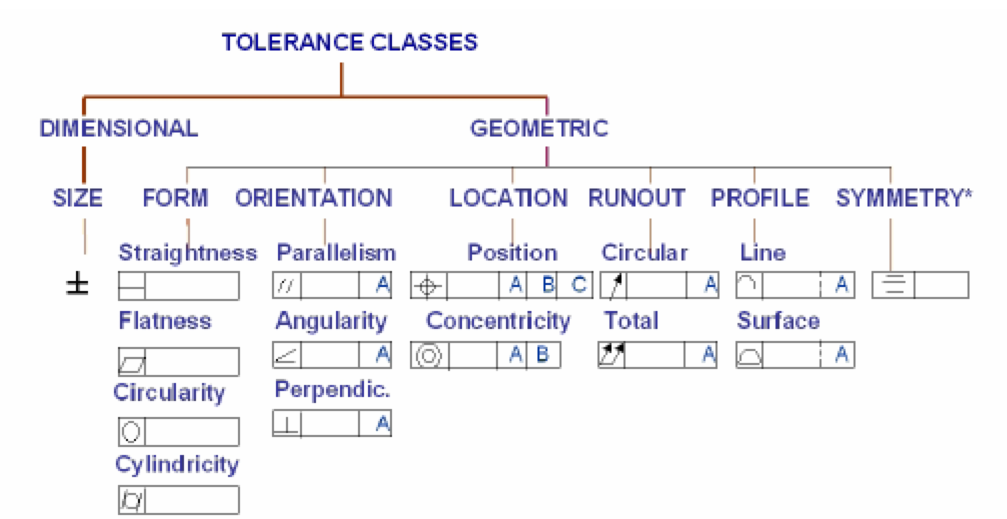

Figure 5 Tolerance Classes [26]

MODEFYING TOLERANCE SYMBOLS:

An integral part of the Feature Control Frame is the modifying tolerance symbols. These are defined by Y14.5 and represented later on in the course. The listing for these modifiers is as follows:

Free State

Least Material Condition (LMC)

Maximum Material Condition (MMC)

Projected Tolerance Zone

Regardless of Feature Size

Tangent Plane

Unilateral

DG&T EXAMPLES:

Let us take a quick look at the type of dimensioning used for drawings that adopt GD&T methodology. The three-dimensional models AND the drawing show the basics of the parts and the application of GD&T. The dimensions are in MM. For the part given in figure 6, please note that the datum "A" is specified as the circumference, or the outer surface, of the part. The eighteen serrations or splines have a runout tolerance of 0.02 MMs relative to datum "A". As you can see from the solid model, the runout is critical because there is a mating part to the total assembly. The outer diameter is 60 MM with a tolerance of 0.01 MM. The internal diameter is 45 MM with a tolerance of 0.01 MM. The internal diameter, at the serrations, is 40 MM with a tolerance of 0.020MM.

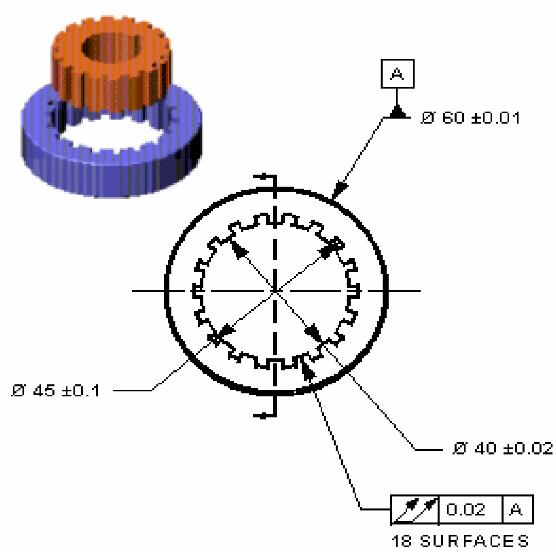

Figure 6 Spline Assembly

The drawing below will show a handle dimensioned using GD&T. The datum surfaces have been selected as "A", "B" and "C". The basic dimensions are included in rectangular blocks. Figures 6 and 7 are very basic but they do represent examples of applied DG&T.

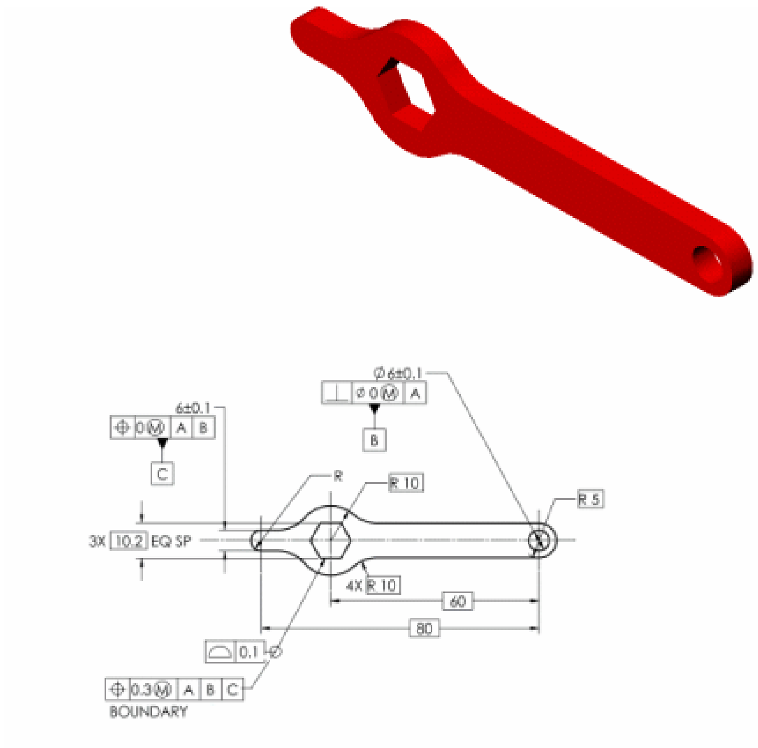

Figure 7 Handle Using GD&T

The part shown below is an actual part I have been asked to fabricate. It is an example of "linear dimensioning". My client is a Fortune 500 company yet does not use GD&T as protocol during the CAD process. I don't want to be overly critical, but there are several glaring omissions relative to this drawing and the notes for the drawing.

- There is no specification for flatness. (I know how the part is used and it must have a flatness of at least 0.060 inches relative to top and bottom parallel surfaces.)
- There is no specification for the perpendicularity of sides. (It could be a parallelogram or a trapezoid and still comply with the drawing BUT, it would not fit the convection fan platform it is designed to mate with!)
- There is no specification for the parallelism of sides. They do need to be parallel and within a specific tolerance zone.
- The material specification is very incomplete. There needs to be an ASME or ASM specification for clarity.
- The note "part to be inspected to sheet metal tolerances" is really bogus. The drawing should include the required tolerance for each dimension.
- Note 4 calls for symmetry about the centerlines. A centerline is very difficult to measure and does not represent a suitable datum, especially since there is no call for parallelism or perpendicularity. A datum is a physical entity—a centerline is NOT. (This drives QC inspectors crazy !!!!!!)

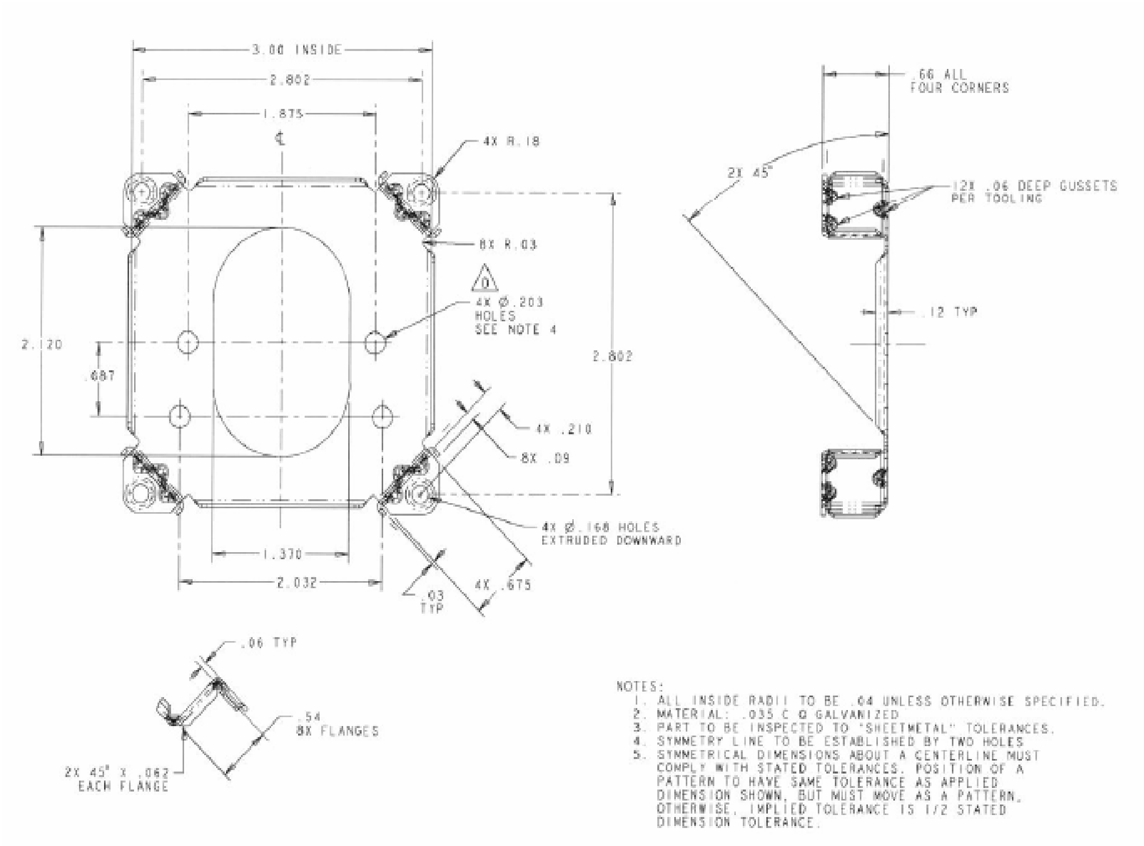

Figure 8 Example of Linear Dimensioning

We have already looked at two fairly simple drawings, and their solid models, but now let us consider a much more complex part that is properly dimensioned using GD&T. All of the dimensions and tolerances are included within the feature control frames and every critical dimension has feature control frames WITH tolerances given. The datums; i.e. A and B, have been selected so as to provide a "baseline" from which all details are given. The datum positions ARE physical features of the part, not centerlines. The feature control frames carry information on diameter, position, maximum material condition, perpendicularity, profile, parallelism and flatness. Also, each dimension is duly noted with a tolerance callout. You may think this is a "busy" drawing but all of the required data is given for fabrication and inspection. There is no ambiguity as far as what shape, dimensions and tolerances are required. I could submit this drawing in the United States, China, New Zealand, Canada, Mexico, etc and a company or person understanding GD&T could make the part with no difficulties. As a matter of fact, I would suspect all parts submitted by individuals for inspection from these countries would produce very similar products. This is exactly what we want. We will discuss the symbols and tolerance modifiers later on in the course.

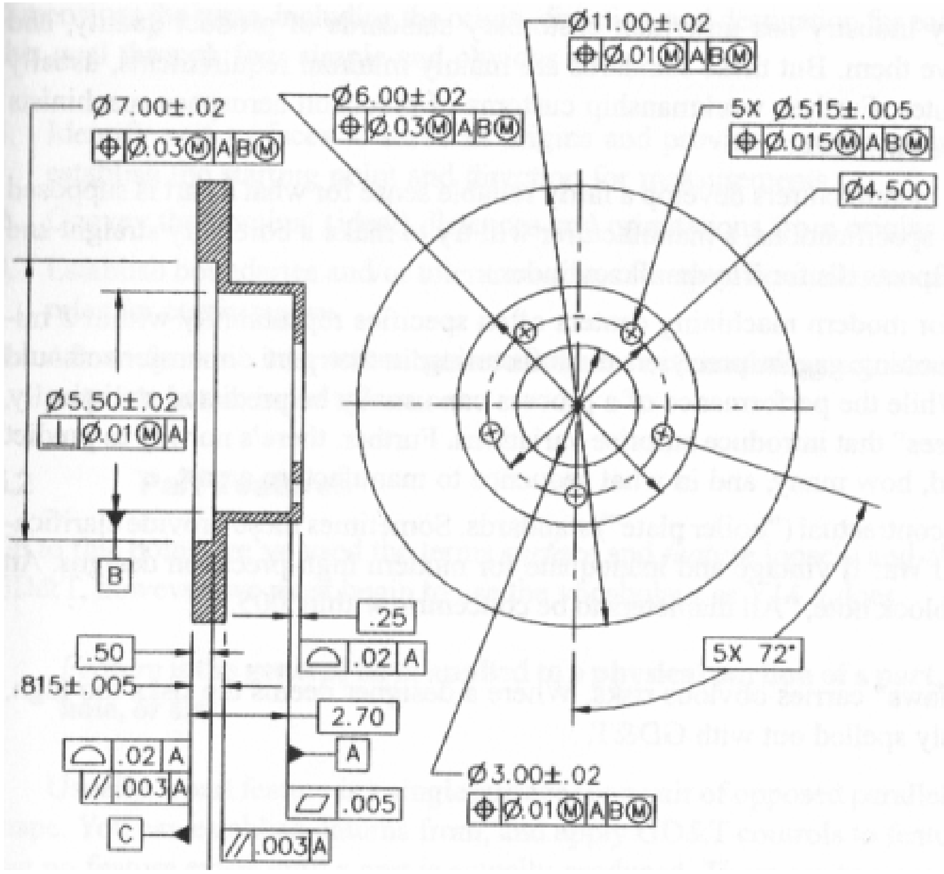

Figure 9 Collar [2]

I don't want to belabor the point but, what if we wanted to use English to detail this part? No dimensions, no tolerances, just good old English. This is an interesting exercise and will demonstrate the hopeless feeling you get when trying to "put it to words". Given below in Figure 10 is what we would have. As you can see, we increase the ambiguity one-thousand fold and make the drawing basically unreadable. Notes might help but the meaning is really clouded with ambiguity.

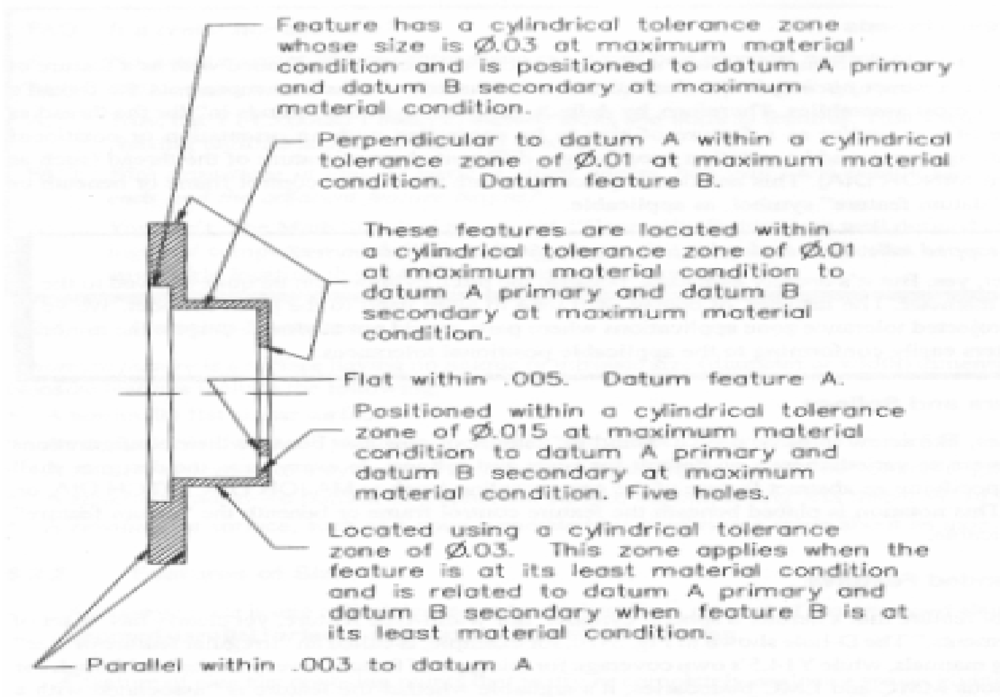

Figure 10 Collar in English Only [2]

FEATURE CONTROL FRAME:

We now wish to consider the Feature Control Frame and the layout of that frame. Next to selecting the datum references, the feature control frame is the most descriptive characteristic of any one drawing using GD&T and carries the bulk of the information needed for detailing the part. The layout is as follows:

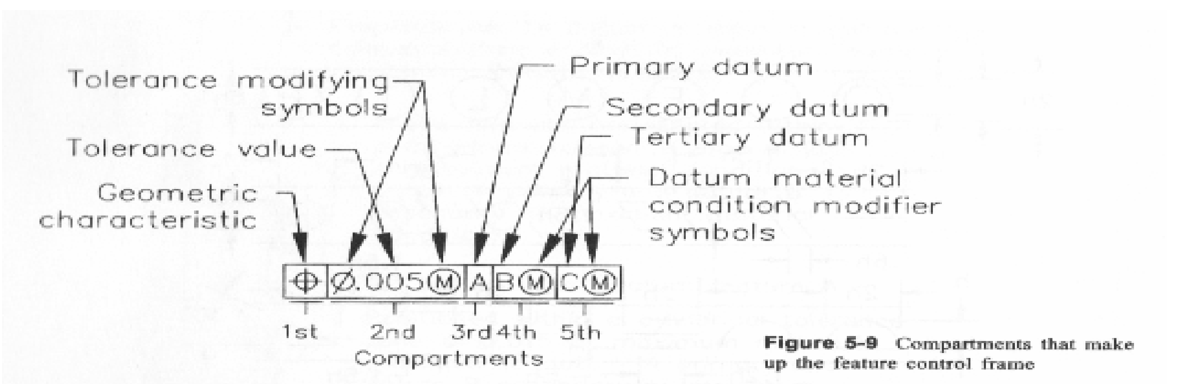

Figure 11 Feature Control Frame [2]

The placement of the geometric characteristics, tolerance values, modifying symbols, datum selection and material condition modifiers is very specific and MUST not be applied in a random fashion. We always apply the features **left to right** within the feature control frame and in the following order:

1. Geometric characteristics
2. Tolerance modifying symbols
3. Tolerance value(s)

4. Primary datum reference

5. Secondary datum reference

6. Tertiary datum reference

7. Datum material conditions

The table below will summarize the application options and rules for each of the fourteen (14) types of **geometric characteristics**. Please note the characteristic, symbol, and the type of feature controlled is given. These characteristics are placed in the first "box" of the feature control frame.

CHARACTERISTIC	SYMBOL	TYPE OF FEATURE CONTROLLED	FEATURE CONTROL FRAME PLACEMENT OPTIONS (SEE LEGEND)	BOUNDARY/TOL ZONE SHAPE MODIFIER	TOLERANCE MODIFIABLE TO MMC OR LMC	NUMBER OF DATUM REFERENCES	MMC/LMC ALLOWED FOR DATUM REFERENCE(S)	BASIC DIMENSIONS REQ'D
STRAIGHTNESS	—	CYL-SURFACE ELEMENTS	b			0		
STRAIGHTNESS	—	CYL-DERIVED MEDIAN LINE	a, d	Ø	✓	0		
STRAIGHTNESS	—	PLANE-LINE ELEMENTS	b, c			0,1		
FLATNESS	▱	PLANE	b, c			0		
FLATNESS	▱	WIDTH-DERIVED MEDIAN PLANE	a, d		✓	0		
CIRCULARITY	○	REVOLUTE, SPHERE	a, b, d			0		
CYLINDRICITY	⌭	CYLINDER	a, b, d			0		
PROFILE OF A LINE	⌒	ALL	b			0–3	✓	✓
PROFILE OF A SURFACE	⌓	REVOLUTE	b			0–3	✓	✓
PROFILE OF A SURFACE	⌓	OTHER (NON-REVOLUTE)	b			0–3	✓	✓
PROFILE OF A SURFACE	⌓	COPLANARITY OF PLANES	b			0		
PERPENDICULARITY PARALLELISM	⊥ //	PLANE (INCL LINE ELEMENTS)	b, c			1–3	✓	
PERPENDICULARITY PARALLELISM	⊥ //	CYLINDER	a, d	Ø	✓	1–3	✓	
PERPENDICULARITY PARALLELISM	⊥ //	WIDTH	a, d		✓	1–3	✓	
PERPENDICULARITY PARALLELISM	⊥ //	REVOLUTE-RADIAL ELEMENT	b, c			1–3	✓	
ANGULARITY	∠	PLANE (INCL LINE ELEMENTS)	b ,c			1–3	✓	✓
ANGULARITY	∠	CYLINDER	a, d	Ø	✓	1–3	✓	✓
ANGULARITY	∠	WIDTH	a, d		✓	1–3	✓	✓
ANGULARITY	∠	REVOLUTE-RADIAL ELEMENT	b, c			1–3	✓	✓
POSITION	⌖	CYLINDER	a, d	Ø	✓	1–3	✓	✓
POSITION	⌖	WIDTH	a, d		✓	1–3	✓	✓
POSITION	⌖	SPHERE	a, d	SØ	✓	1–3	✓	✓
CONCENTRICITY	◎	ALL NON-SPHERICAL	a, b, d	Ø		1–3		
CONCENTRICITY	◎	SPHERE	a, b, d	SØ		1–3		
SYMMETRY	⌯	OPPOSED POINTS	a, d			1–3		
CIRCULAR RUNOUT	↗	REVOLUTE	a, b, d			1–2		
TOTAL RUNOUT	⌰	CYLINDER	a, b, d			1–2		
TOTAL RUNOUT	⌰	PLANE PERP TO AXIS	b, c			1–2		

FEATURE CONTROL FRAME PLACEMENT OPTIONS (LEGEND)

Table 4 GD&T Characteristic [2]

In addition to geometric characteristics, there are modifying tolerance symbols necessary for a complete description of the part. These are as follows:

Characteristic	Symbol
At maximum material condition	Ⓜ
At least material condition	Ⓛ
Projected tolerance zone	Ⓟ
Free state	Ⓕ
Tangent plane	Ⓣ
Diameter	Ø
Spherical diameter	SØ
Radius	R
Spherical radius	SR
Controlled radius	CR
Reference	( )
Arc length	⌒
Statistical tolerance	⟨ST⟩
Between	↔

Table 5 Modifying Tolerance Symbols [2]

These are quite important and somewhat self-explanatory but I would like to explain three that are frequently confused and one very pertitenant to DFSS and six sigma.

- Maximum Material Condition (MMC)—The condition in which a feature contains the most amount of material everywhere within the stated limits of size.
- Least Material Condition (LMC)—The condition in which the feature contains the least amount of material everywhere within the stated limits of size.
- **Statistical Tolerance—The statistical tolerance symbol denotes that dimension or tolerance which was derived or established by using statistical methods and / or six sigma calculations. Generally, these tolerances were established by measuring components during the benchmarking process. We will be considering this modifier later when we "marry" DFSS and GD&T. Another determination of statistical tolerancing is derived from process capability studies; i.e. how good is my process and what tolerances result from investigating those tolerances during the process.**

READING BASIC SYMBOLS:

Now an explanation as to how to read these basic symbols. Each geometric tolerance class is represented by a region or zone; the shape of the zone depends upon the tolerance type and the feature being toleranced. The size depends upon the tolerance value, material condition modifiers and certain rules; the position / orientation of the zone depends upon the tolerance type and datums selected. Figure 12 below shows the zones for perpendicularity, flatness and runout. [26]

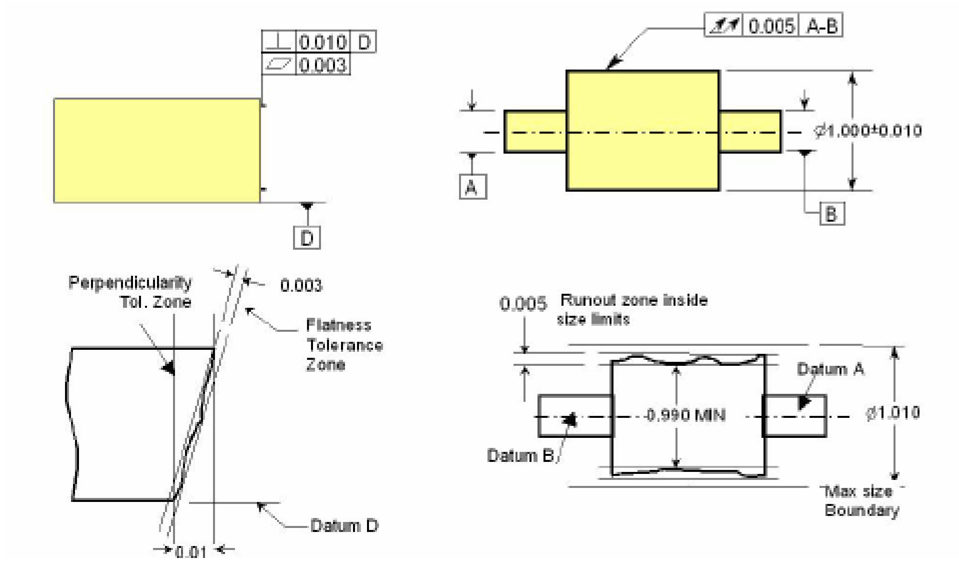

Figure 12 Flatness, Runout and Perpendicularity [26]

As you can see, there is a boundary, zone or region that defines the geometric characteristic. Looking at the feature control frame relative to flatness, there is a tolerance of 0.003 inches which the part must conform to relative to datum "D". This 0.003 represents two parallel surfaces 0.003 inches apart. The flatness must lie within this zone to be within specifications. The part must also be perpendicular to datum "D" within 0.010 inches. Runout is the same. The runout of the part on the right must be within 0.005 inches relative to datums "A" and "B". This is a zone with surfaces 0.005 inches apart.

We discussed, briefly, MMC. In order to allow for trade-offs between feature size and certain types of geometric tolerances, such as position, the standards use material modifiers; i.e. MMC, LMC, etc to indicate what the geometric tolerance is when the size is at its largest or smallest value. When the feature size deviates from that value, a "bonus tolerance" is added to the geometric tolerance; i.e. trading position variation for size variation; when a modifier is applied to a datum feature of size, the geometric tolerance zones shift, which is equivalent to a larger zone. This is shown with the figure below.

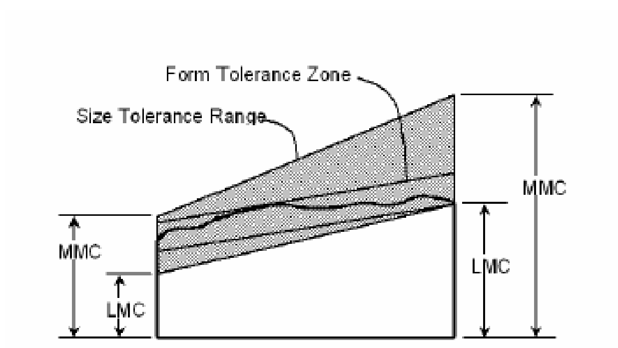

Figure 13 MMC, LMC [26]

I would like to go back now and provide several "bullets" for figure 9. These further explain the use and methodology for GD&T. All dimensions are in inches.

- The largest inside diameter measures 7.00 inches with a tolerance of ± 0.02 inches. The position of the surface represents the maximum material condition.
- The second largest internal diameter is 5.50 inches with a tolerance of ± 0.020 inches. It represents an MMC condition and must be perpendicular to datum "A" within 0.01 inches.

- The thickness of the material at Datum "A" is 0.250 inches.
- The profile of the surface of the collar hub must be 0.020 inches relative to Datum "A" This means it must be in a tolerance zone of 0.020 inches.
- The diameter of the five (5) holes on the part must be located on a bolt circle (BC) and within a diameter of 4.50 inches. They must be 0.515 inches in diameter and with a tolerance of 0.0050 inches. The BC must have a tolerance of 0.015 inches relative to Datums "A" and "B" when the part is in the MMC

Before we forget, the application of a basic dimension to a drawing, using GD&T, looks as follows:

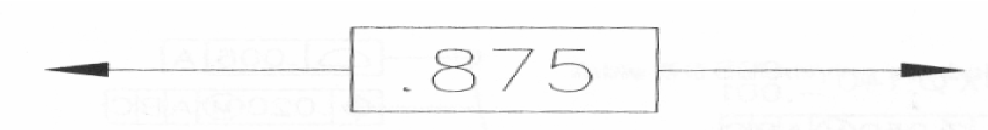

Figure 14 Basic Dimensions [1]

Please note that there are no tolerances applied to the basic dimension. Tolerances are used only in Feature Control Frames.

I would like to emphasize that there are many sources for training and consultation regarding GD&T. Several are given below as follows:

- Your local technical community college
- Dimensional Management Systems, LLC
- Tec-Ease, Inc.
- Geometric Learning Systems, Inc.
- Advanced Dimensional Management, LLC
- International Institute of GD&T
- Engineers Edge
- James D. Meadows & Associates, Inc.
- The QC Group

PART III

DFSS AND GD&T:

In this section we will attempt to show how DFSS AND GD&T are interrelated. We will find that GD&T is a methodology **critical to DFSS** if repeatability and predictive assembly are to be accomplished. DFSS is involved with TOLERANCE ALLOCATION. **Tolerance allocation is a method which will allocate tolerances to various components and an assembly of mating components so as to predict the outcome.** In this fashion, we can use the tenants of six sigma; i.e. bell-shaped curve, USL and LSL, to aid our efforts. We start with defined goals for the assembly, decide how each part is to be manufactured, and allocate what tolerances AND standard deviation should be applied to each dimension so that the part can be economically produced and assembly requirements are met. One method of obtaining information relative to USL, LSL and standard deviation is **Statistical Process Control**. Many companies have adopted SPC in their fabrication processes to determine Cpk for purposes of controlling the outcome of any manufacturing method with the hopes of reducing or eliminating unusable (scrap) components. If production data is not available, you can estimate recommended tolerances and standard deviation from existing data bases relative to individual manufacturing methods. Using these concepts, we assign tolerances; i.e. GD&T to drawings knowing that "good practice" will allow us to be within generally accepted dimensional deviations. The purpose of tolerance analysis is to study the accumulation of variations on the geometric attributes of interest; i.e. dimension, location, orientation, etc. The need for this arises from the fact that the analyzed dimensions are not explicity specified. The most common case is the analysis of clearances in assemblies. All dimensions and tolerances that affect the clearance are contributors. This tolerance chain is called the **stack path** and usually does involve multiple components and possibly multiple assemblies. Worst case analysis is done to determine the maximum or minimum values resulting from the limits specified by the contributors. Statistical analysis is used to determine the frequency distribution of the contributors. Worst case design guarantees 100% interchangeability of parts.

TOLERANCE ALLOCATION:

I would like now to present a table that will indicate the second basic step to tolerance allocation and tolerance analysis. The first step is the structuring of a "LOOP" diagram to define the layout of the components and the assembly of components. Please note that this table is just a partial table of what data is available throughout the literature. The list literally goes on and on, but the most important point is that the data for standard deviation is available **if you have not developed it through Statistical Process Control (SPC) relative to your individual processes.** The standard deviations given below are process specific relative to the individual manufacturing method. These values represent a very good "starting point" for the tolerance analysis that is to follow.

PROCESS	STANDARD DEVIATION (INCHES)	PROCESS	STANDARD DEVIATION (INCHES)
N/C MILLING	0.00026	JB END MILLING	0.000105
N/C SIDE MILLING	0.00069	JB SIDE MILLING	0.000254
N/C SIDE MILLING > 6 IN.	0.00093	JB BORE HOLES <0.13 DIA	0.000048
N/C DRILLING HOLES (LOCATION)	0.00076	JB BORE HOLES (LOCATION)	0.000054
N/C DRILLINGHOLES (DIAMETER)	0.00056	JB DRILLING HOLES (LOCATION)	0.000769
N/C TAPPED HOLES	0.0025	JB COUNTER SINK	0.001821
N/C COUNTERSINK (LOCATION)	0.00211	JB REAMING (DIAMETER)	0.000159
N/C END MILL PARALLEL	0.0002	JB END MILL>16 SQ IN	0.00009
TURNING ID	0.000127	TREYPAN ID	0.000127
TURNING OD	0.000132	GRINDING LAP	0.000027
GRINDING SURFACE	0.000029	GRINDING OD	0.000029

Table 6 Standard Deviation vs Process

Again, the table below is a partial table but will demonstrate what data is available relative to the manufacturing process.

PROCESS	STANDARD DEVIATION (INCHES)	PROCESS	STANDARD DEVIATION (INCHES)
ALUMINUM CASTING (INCHES)		STEEL CASTING (INCHES)	
CAST UPTO 0.250	0.00083	CAST UPTO 0.250	0.00059
CAST UPTO 0.0500	0.001035	CAST UPTO 0.500	0.00106
CAST UPTO 1.000	0.001597	CAST UPTO 1.000	0.001346
CAST UPTO 2.000	0.002102	CAST UPTO 2.000	0.002099
CAST UPTO 3.000	0.00.2662	CAST UPTO 3.250	0.003064
CAST UPTO 11.00	0.008126	CAST OVER 11.00	0.011711
CAST FLAT < 2 SQ IN	0.001543	CAST FLAT < 2 SQ IN	0.00152
CAST FLAT <4 SQ IN	0.002003	CAST FLAT < 4 SQ IN	0.002059
CAST STRAIGHT OVER 10 IN	0.00904	CAST STRAIGHT OVER 10 IN	0.009289

Table 7 Standard Deviation vs Process

After constructing a loop diagram, we want to develop a tolerance allocation spreadsheet using data similar to what has been given in tables 6 and 7. This is very critical and allows us to "book-keep" the data and capture all of the relevant assumptions. Please keep in mind this is just an example and does not pertain to any "real" part or assembly of parts. The tolerances are applied to the geometric characteristics of the individual parts so a realistic "gap" dimension, gap tolerance and gap standard deviation may be assigned.

VARIABLE NAME (DIMENSION)	NOMINAL DIMENSION	SENSITIVITY	TOLERANCE (PLUS OR MINUS)	STANDARD DEVIATION	PROCESS
A	0.3595	-1	0.0155	0.00286	CAST FLAT > 6.00 IN
B	0.032	1	0.002	0.00211	N/C COUNTERSINK
C	0.06	1	VARIABLE	0.000029	GRINDING SURFACE
D	0.4305	1	FIXED	0.000127	TURNING
E	0.1200	1	0.0075	0.00152	CAST < 2.00 SQ IN
F	1.5030	1	0.007	0.00201	CAST UPTO 2.000
G	0.1200	1	0.0100	0.000105	JB END MILLING
H	0.4305	1	0.0015	0.00026	N/C END MILLING

Table 8 Tolerance Allocation Spreadsheet

STATISTICAL ALLOCATION:

This spreadsheet and method is viable and usable so that the tolerances are allocated in a fashion to guarantee assembly. We want to take a look at STATISTICAL ALLOCATION. This is where we assign tolerances, combined with standard deviation, to produce a predictive outcome. The "ST" GD&T modifier we discussed earlier is specified on the drawing and does result from using the statistical approach to tolerance allocation.

Here are the steps necessary:

- Produce a "LOOP" diagram showing the dimensions vs the "gaps"or separation between mating parts. Of course the object here is to determine if there are interference points "designed into" the overall assembly of parts. **(THIS SHOULD BE THE VERY FIRST STEP IN THE PROCESS TO GET A VISUAL IDEA OF THE ASSEMBLY)** Please note that **a "sign" convention** is necessary to obtain the proper results; i.e. right to left is negative (-) and left to right is positive (+). Either way is fine but you must keep the stated convention during the solution to the problem. NO CHANGING !!.
- Construct an Allocation Spreadsheet similar to the sheet above or your version of what is needed.
- "Lay in" the dimensions, tolerances (first pass) and the expected standard deviation relative to the manufacturing process.
- Calculate the expected assembly performance using six sigma; i.e. USL and LSL methodology.
- Reallocate the tolerances depending upon the six sigma analysis.
- Recalculate the assembly performance.
- Apply the tolerances, using GD&T techniques to your component drawing(s) and overall assembly.

A figure showing the steps to this process is as follows:

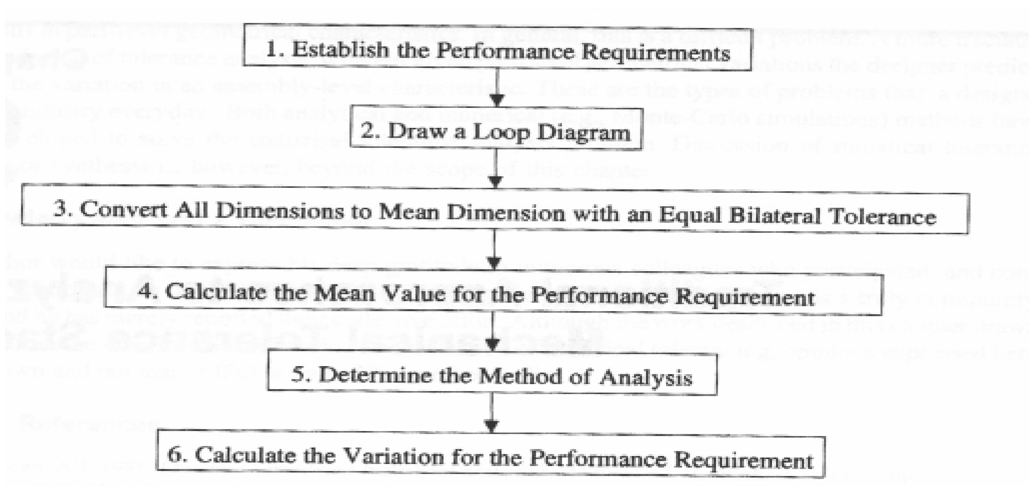

Figure 15 Steps for Evaluating "Gap" Allocation

In calculating the mean value of the gap, we use the following formula:

$$d(g) = \Sigma\ (i=1 \text{ to } i=N) = a(i)d(i)$$

where

d(g) = the mean value of the gap. If d(g) is positive, the mean gap has clearance: if negative, the mean gap has interference.

N = the number of independent variables (dimensions) in the stackup

a(i) = sensitivity factor that defines the direction and magnitude of the ith dimension. In a one-dimensional stackup, this value is essentially +1 or -1. Sometimes, in a one-dimensional stackup, the value maybe +0.50 or –0.50 if a radius is the contributing factor for a diameter callout on a drawing.

d(i) = the mean value of the ith dimension in the loop diagram.

Now let us take an example of getting this accomplished. We are going to look at the statistical tolerancing of a range drawer. On a freestanding or slide-in range, there is generally a lower drawer that slides in and out. The drawer is located below the oven door and provides storage for pots, pans, etc. The proper fit is definitely necessary to maintain appearances and eliminate drag or unnecessary resistance when the drawer is loaded with utensils. The picture will show the location of the lower range drawer relative to the oven cavity.

Figure 16 GE Lower Drawer

PROBLEM EXAMPLES:

We will select as our sign convention---down is positive (+) and up is negative (-). This is really not intuitive, unless you have range experience, but we are choosing, as the dimension to control, the gap between the top of the range drawer and the trim piece between the oven cavity door and the drawer. This we call "gap". It is very important to maintain the same gap across the front of the range. Varying gap dimensions will be picked up by the consumer very quickly. We will have component dimensions labeled "A","B","C","D","E","F" and finally "gap". For a six sigma design, we choose to control the gap. Also, when doing these calculations, I always have a part, or at least a picture of the part available for observation. You can really get lost when doing these things, so any and all visual aids are very helpful. I will show a motor assembly later on that will demonstrate the need for drawings, parts, pictures, etc.

LOCATION	DIMENSION	µ VALUE	Δ VALUE
A	4.600	4.580	0.020
B	4.600	4.580	0.020
C	9.350	9.320	0.030
D	9.350	9.410	0.051
E	5.250	5.230	0.020
F	5.250	5.190	0.060

Table 9 Dimensions and Mean Values

The calculations follow the loop diagram.

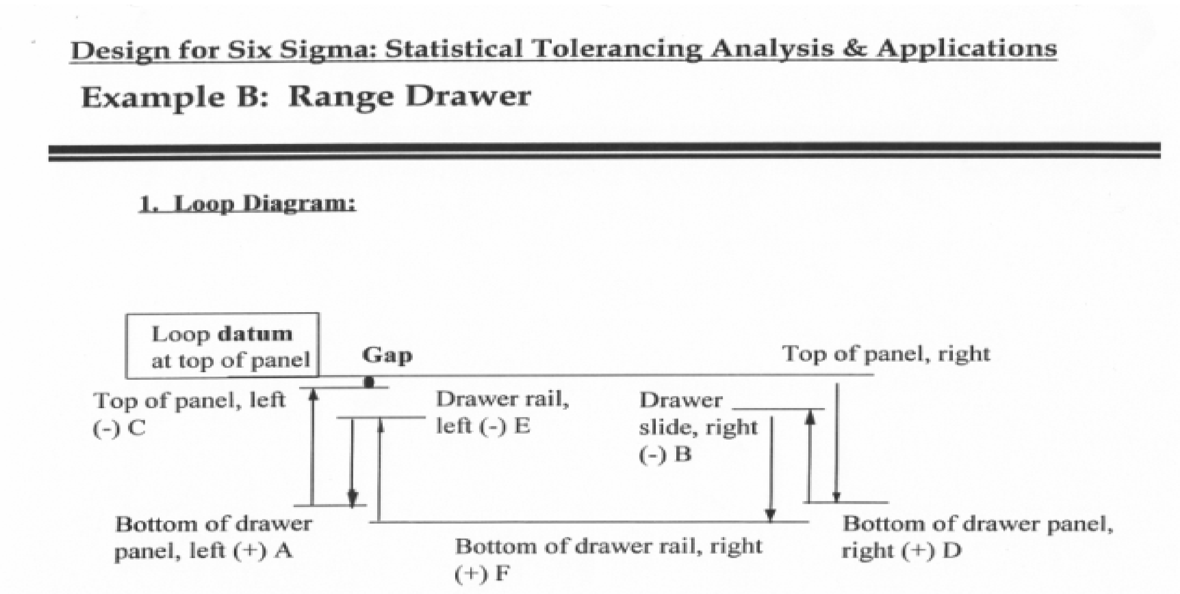

Figure 17 Loop Diagram for Oven Range Drawer [20]

As an explanation, the mean values are determined from process control data. These may be critical to quality dimensions on the drawing. As you can see, the mean gap is 0.050 inches and is within our plus or minus 0.080 inch specification. Now, the standard deviation of the dimension is known also. From this, we can calculate the "sigma-gap" data using the formula given.

Design for Six Sigma: Statistical Tolerancing Analysis & Applications

Example B: Range Drawer

2. Determine if the system is six sigma capable. If not, what gap standard deviation is required to meet the consumers' expectation?

Find the nominal or target and the standard deviation of the gap and compare them to the consumer expectations and six sigma requirements. When you're creating a new design, evaluate the short term capability relative to 6.0σ. When you're improving an existing design, determine or estimate the long term capability relative to 4.5σ.

The technical requirement is a gap of 0 +/- .080 (meaning we don't care which side is higher as long as the overall requirement is not exceeded).

$T_{gap} = T_D - T_B + T_F - T_E + T_A - T_C$
$T_{gap} = 9.35 - 4.60 + 5.25 - 5.25 + 4.60 - 9.35$
$T_{gap} = 0$

$\mu_{gap} = \mu_D - \mu_B + \mu_F - \mu_E + \mu_A - \mu_C$
$\mu_{gap} = 9.41 - 4.58 + 5.19 - 5.23 + 4.58 - 9.32$
$\mu_{gap} = .050$

$\sigma_{gap}{}^{\wedge 2} = \sigma_A{}^{\wedge 2} + \sigma_B{}^{\wedge 2} + \sigma_C{}^{\wedge 2} + \sigma_D{}^{\wedge 2} + \sigma_E{}^{\wedge 2} + \sigma_F{}^{\wedge 2} - 2(\rho)(\sigma_A)(\sigma_B)$
$\sigma_{gap}{}^{\wedge 2} = (.0137)^{\wedge 2} + (.0165)^{\wedge 2} + (.003)^{\wedge 2} + (.002)^{\wedge 2} + (.009)^{\wedge 2} + (.008)^{\wedge 2} - 2(-.755)(.0137)(.0165)$
$\sigma_{gap} = .031$ Note: If we had failed to consider the correlation effect σ_{gap} would have been .025 which is a significant understatement of the actual variation.

11

Design for Six Sigma: Statistical Tolerancing Analysis & Applications

Example B: Range Drawer

To have a six sigma design, we would require gap tolerances a to be 6.15 times the gap standard deviation. The result would be 0+/- (6.15) * (.031) or +/-.191 which significantly exceeds the technical requirements. The existing drawer design has the following Z values:

Short term (new design, assumes tooling is centered):
$Z_{LSL} = [(0 - (-.080)]/.031 = 2.5830 \rightarrow$ PPM = 4,940
$Z_{USL} = (.080 - 0)/.031 = 2.5830 \rightarrow$ PPM = 4,940
$Z_{BENCH} = 2.33$, not 6.0!

What are we currently seeing?
Long term (existing design, factors in existing shifts in target means):
$Z_{USL} = (.080 - .050)/.0496 = .605$
$Z_{LSL} = [.050 - (-.080)]/.0496 = 2.62$
$Z_{BENCH} = .5920$, not 4.5!

To achieve a six sigma design, we would require the following σ_{gap}
$\sigma_{gap} = .080/6.15 = .013$

As you can see, the standard deviation for the gap can be no greater than 0.013 inches to have a six sigma product. Now, let us examine the standard deviation for the drawer slides using the very same methodology.

Design for Six Sigma: Statistical Tolerancing Analysis & Applications

Example B: Range Drawer

3. What standard deviation of the drawer slide would be required to deliver a six sigma design?

Assume $\sigma_A = \sigma_B = x$

$\sigma_{gap}{}^{\wedge}2 = \sigma_A{}^{\wedge}2 + \sigma_B{}^{\wedge}2 + \sigma_C{}^{\wedge}2 + \sigma_D{}^{\wedge}2 + \sigma_E{}^{\wedge}2 + \sigma_F{}^{\wedge}2 - 2(-\rho)(\sigma_A)(\sigma_B)$

$(.013)^{\wedge}2 = x^{\wedge}2 + x^{\wedge}2 + (.003)^{\wedge}2 + (.002)^{\wedge}2 + (.009)^{\wedge}2 + (.008)^{\wedge}2 - 2(-.755)(x^{\wedge}2)$

$\sigma_A = \sigma_B = .0018$

4. What other design options could be considered to deliver a six sigma design?

Each one of the process standard deviations should be weighed for potential improvement if possible and affordable. Additionally, the entire design should be reviewed for a more robust configuration i.e. slotted holes that allow for adjustable fixturing on the assembly line.

As I mentioned earlier, these calculations can get quite cumbersome and detailed. Please take a look at the following motor assembly and you certainly will understand what I mean.

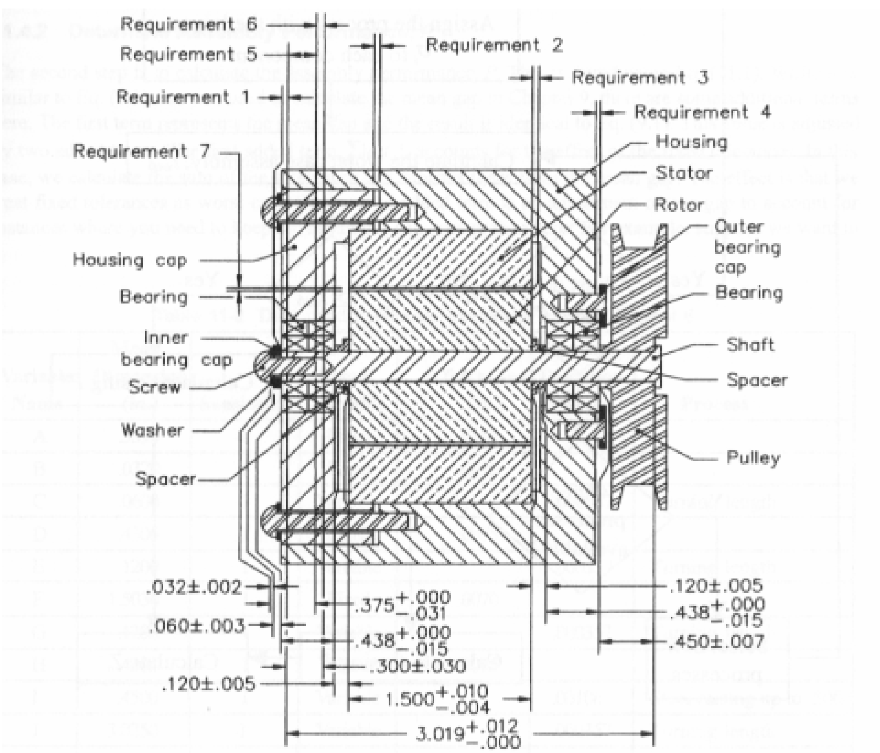

Figure 18 Motor Assembly [2]

We would start by developing a loop diagram. All of the "gaps" and pertinent dimensions would be laid onto the diagram for added clarity. Next comes the allocation spreadsheet with dimensions, tolerances and projected standard deviations. The DFSS calculations would be generated from this assembly drawing and from the diagram.

I would like to give one more example; this one being refrigerator door trim. This time we are choosing as our sign convention ---right to left as (+) and left to right as negative (-). We have dimensions "A","B","C" and "F" with the gap between the trim offset and the trim bump as the dimension needing to be controlled. A sketch of the door trim assembly is given above the loop diagram for some degree of clarity. We will use the following dimensions for our calculations:

T(A) = 30.303 inches

T(B) = 0.700 inches

T(C) = To be found

T(D) = 0.030 inches

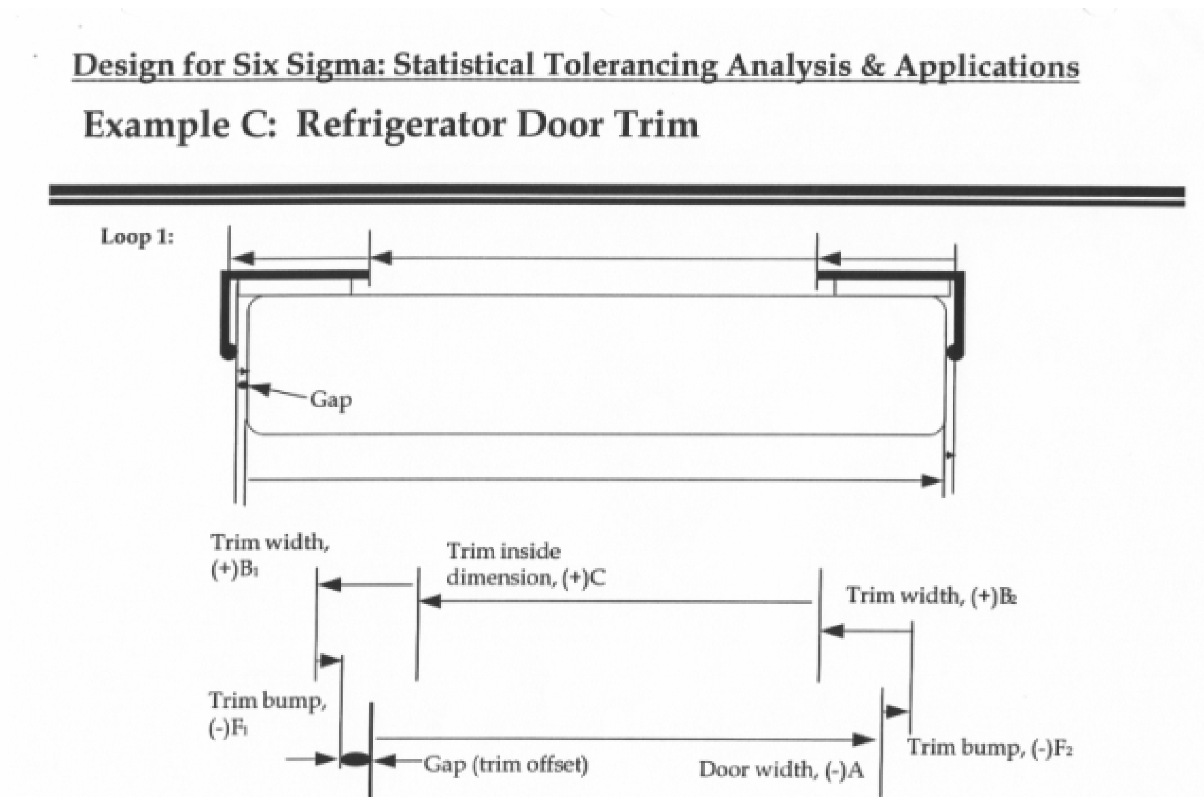

Figure 19 Refrigerator Door Trim [20]

The sketch below will show the details of the exact configuration we wish to examine.

Design for Six Sigma: Statistical Tolerancing Analysis & Applications

Example C: Refrigerator Door Trim

Loop 2:

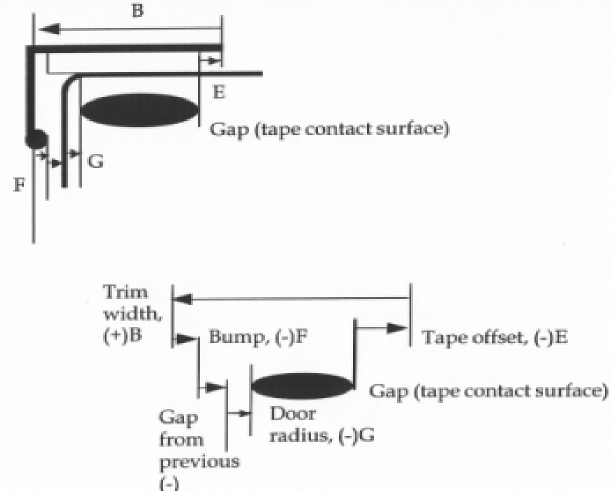

Figure 20 Refrigerator Door Trim Enlargement [20]

Design for Six Sigma: Statistical Tolerancing Analysis & Applications

Example C: Refrigerator Door Trim

Part 1:

$\sigma_{gap}{}^{\wedge}2 = \sigma_A{}^{\wedge}2 + \sigma_{B1}{}^{\wedge}2 + \sigma_{B2}{}^{\wedge}2 + \sigma_C{}^{\wedge}2 + \sigma_{F1}{}^{\wedge}2 + \sigma_{F1}{}^{\wedge}2$
$\sigma_{gap}{}^{\wedge}2 = (.009)^{\wedge}2 + 2(.007)^{\wedge}2 + (.010)^{\wedge}2 + 2(.002)^{\wedge}2$
$\sigma_{gap} = .017$

To prevent an interference fit, we will set the nominal dimension for the gap at 6.15 times the standard deviation of the gap.

$T_{gap} = 6.15 * (.017) = .105$
$T_{gap} = -T_A - T_F + T_B + T_C + T_B - T_F$
$.105 = -30.303 - .030 + .7 + T_C + .7 - .030$
$T_C = 29.068$ +/- $(6.15) * (.010)$
$T_C = 29.068$ +/- $.062$

We see that, for the assembly to be six sigma, we must have a dimension T(c) of 29.068 but with a tolerance of ±0.062 inches.

CONCLUSIONS:

We have dealt with two very important technical methodologies in this course: 1.) Design for Six Sigma and 2.) Geometric Dimensioning and Tolerancing. Both represent practical approaches to the design and manufacturing of mechanical components. Our goal, as engineers, is always to design products that meet customer needs AND provide the safety required for normal and even some abnormal use. DFSS and GD&T are only two of the vehicles that can help us realize these goals but they are very important to each design "type" and, when used, will save hours of "cut and try". I personally feel that DFSS is only achievable if GD&T is used to describe a component or an assembly of components. That is the very reason that I have chosen to discuss both in this course. We have only touched the surface of both technologies; so I do hope you will consider further investigation so that incorporation of these techniques into your company and engineering department will result. It does take time, training and the resolve to get it done. For complete success, every member of your design, reliability, quality control, manufacturing and inspection team must be "on board" and willing to accept the challenge so that process improvement can occur.

GEOMETRIC DIMENSIONING AND TOLERANCING (GD&T) AND DESIGN FOR SIX SIGMA(DFSS)

REFERENCES

(1.) American Society of Mechanical Engineers, Drafting Manual, ASME Y14.5M-2009

(2.) Paul Drake, Jr. ,Dimensioning and Tolerancing Handbook , (McGraw-Hill, 1999).

(3.) Dr. Mark J. Kiemele, "Using the Design for Six Sigma (DFSS) Approach to Design, Test and Evaluate to Reduce Program Risk ", (Air Academy Associates, NDIA Test and Evaluation Summit, Victoria, British Columbia, February 24, 2003)

(4.) Mr. Peter Peterka , "Design for Six Sigma", (Six Sigma Consulting, 2009)

(5.) H. Rassouli , "Geometric Dimensioning and Tolerancing (GD&T,)" ,(Machine Design, May 30, 2009)

(6.) Dr. Vedaraman Sriraman and Dr. John DeLeon ,"Teaching Geometric Dimensioning and Tolerancing in a Manufacturing Program", (Journal of Industrial Technology, 1999.)

(7.) "ASME Releases Revision of Geometric Dimensioning and Tolerancing Standard", (ASME April 9, 2009.)

(8.) "Reaching Beyond Six Sigma" by Breakthrough Management Group International, June 30, 2009

(9.) "Concept to Customer", (C2C Solutions, 1998)

(10.) Kenneth Crow, "Design for Six Sigma" , (DRM Associated, 2006)

(11.) ,ASI Consulting Group, " Design for Six Sigma", (ASI Consulting Group, 2009)

(12.) "Geometric Dimensioning and Tolerancing", (Engineer's Edge, 2009.)

(13.) Tech-Ease, Inc., "GD&T Tips", (Tech-Ease, Inc., 2009)

(14.) Effective Training, Inc., "GD&T Glossary" , (Effective Training, Inc., 2009)

(15.) Efunda Engineering Fundamentals.,"Geometric Dimensioning and Tolerancing—Why?", (Efunda Engineering Fundamentals, 2009)

(16.) "Brown and Sharpe ,"Geometric Dimensioning and Tolerancing", (Brown and Sharpe, 2009.)

(17.) Absolute Astronomy.,"Geometric Dimensioning and Tolerancing", (Absolute Astronomy, 2009).

(18.) Mr. Alex Krullkowski and Dr. James R. Rolle, "Avoiding Design Problems with GD&T", (Machine Design, 2009.)

(19.) UGS, NX Digital Product Development White Paper , "Design for Six Sigma", 2009

(20.) The General Electric Company " Six Sigma—Blackbelt Training Course", (General Electric Company, 1998)

(21.) Alex Krulikowski, Geometric Dimensioning & Tolerancing, (Effective Training, Inc, 1996)

(22.) C2C Solutions, "Integrating DFSS Best Practices Into a Product Development Flowchart", (C2C Solutions, 1999)

(23.) Advanced Dimensional Management, LLC, "Geometric Dimensioning and Tolerancing (GD&T), 2009.

(24.) Mechanical Engineering Department, Brigham Young University, Provo, Utah, DE-Vol 82, Jinsong Gao, Kenneth W. Chase, Spencer P. Magleby; Volume 1, ASME 1995, Proceedings of the ASME Design Engineering Technical Conferences, Boston, Ma, September 17-20, 1995, PP 353-360, " Comparison of Assembly Tolerance Analysys by the Direct Linearization and Modified Monte Carlo Simulation Methods".

(25.) Colin Milberg, Iris D. Tommelein, and Thais Alves, 3rd International Conference on Concurrent Engineering in Construction; University of California, Berkeley, 1-2 July 2002,

(26.) Jami J. Shah, Gaurav Ameta, Zhengshu Shen and Joseph Davidson; Arizona State University; Computer-Aided Design and Applications, Vol 4, No. 5, 2007, PP 705-717, " Navigating the Tolerance Analysis Maze".

(27.) Hamid Karimzadeh, Automotive Engineering Partners, February 2004, " Statistical Tolerance Analyses Increases Economic Efficiency in Series Production"

(28.) Zhihua Zou, Edward Morse; Center for Precision Metrology, Department of Mechanical Engineering & Engineering Science, The University of North Carolina at Charlotte, " Statistical Tolerance Analysis Using GapSpace". 7th CIRP International Seminar on Computer Aided Tolerancing, ENS de Cachan, France.