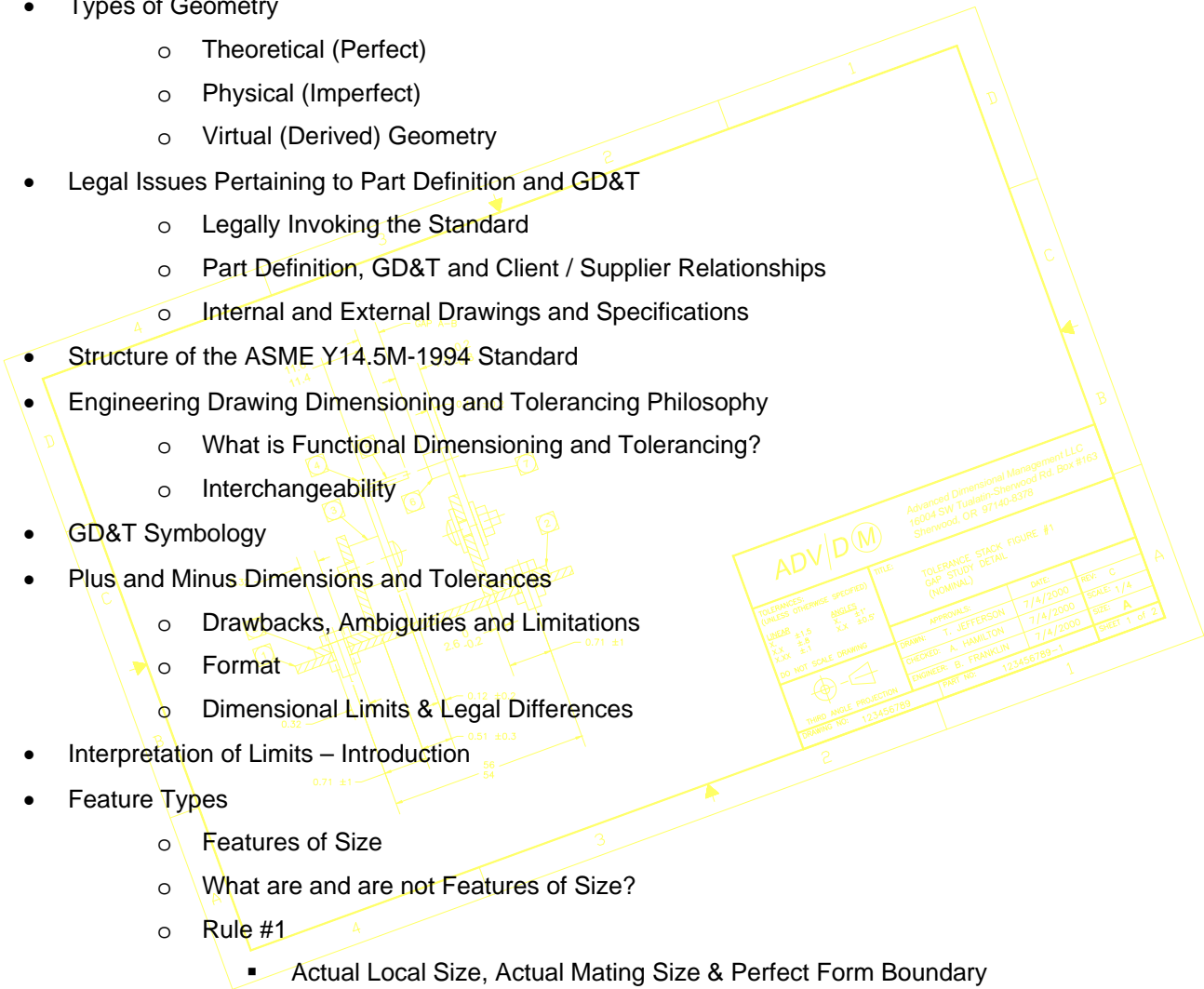


Level 1 GD&T Course Outline

- What is GD&T?
 - Why is it Important?
 - Avoiding Ambiguity in Specifications
- What is ASME Y14.5M-1994?
 - Why is it Important?
- Types of Geometry
 - Theoretical (Perfect)
 - Physical (Imperfect)
 - Virtual (Derived) Geometry
- Legal Issues Pertaining to Part Definition and GD&T
 - Legally Invoking the Standard
 - Part Definition, GD&T and Client / Supplier Relationships
 - Internal and External Drawings and Specifications
- Structure of the ASME Y14.5M-1994 Standard
- Engineering Drawing Dimensioning and Tolerancing Philosophy
 - What is Functional Dimensioning and Tolerancing?
 - Interchangeability
- GD&T Symbology
- Plus and Minus Dimensions and Tolerances
 - Drawbacks, Ambiguities and Limitations
 - Format
 - Dimensional Limits & Legal Differences
- Interpretation of Limits – Introduction
- Feature Types
 - Features of Size
 - What are and are not Features of Size?
 - Rule #1
 - Actual Local Size, Actual Mating Size & Perfect Form Boundary
 - Bounded Features – Introduction
 - Complex Features – Introduction
- Fundamental Rules and Definitions
- Material Condition and Material Condition Modifiers
- Basic Dimensioning Practices
- Defaults for Tolerancing Screw Threads, Splines and Gears – Introduction



- Actual Mating Envelopes
 - Oriented to the Feature
 - Oriented to a Datum Reference Frame
- Datums and Datum Reference Frames
 - Definitions:
 - Datum Reference Frames & Functional Datum Reference Frames
 - Datum Feature Types
 - Introduction to the Degrees of Freedom Concept
 - Datum Feature Precedence
 - Specifying Datum Features
 - Datum Targets
 - Datum Reference Frames: 3-2-1 vs. Degrees of Freedom Concepts
 - Datum Reference Frame Examples
 - Deriving Datums from Datum Features and Datum Feature Simulators
 - Constructing Datum Reference Frames and Coordinate Systems
 - Tolerancing Datum Features
- Form Tolerances
 - Flatness
 - Circularity
 - Cylindricity
 - Straightness
- Virtual Condition and Resultant Condition – Introduction (MMC)
 - Boundaries
 - Calculations and Formulas
- Center Geometry
 - Tolerance Zones for Center Geometry – Introduction
 - Which Axis is Being Controlled?
- Orientation Tolerances
 - Perpendicularity
 - Parallelism
 - Angularity
- Location Tolerances
 - Positional Tolerancing
 - Rules
 - Surface vs. Center Geometry
 - Bonus Tolerance



- Comparison with Plus and Minus Tolerancing
- Positional Tolerancing and Functional Gaging Examples
- Evaluating a Positional Tolerance Specification
- Boundary Concept – Introduction
- Simultaneous and Separate Requirements
- Floating Fastener and Fixed Fastener: Examples and Calculations
- Projected Tolerance Zones
- Concentricity
 - What it means and doesn't mean
 - Recommendations
- Symmetry
 - What it means and doesn't mean
 - Recommendations
- Runout Tolerances
 - Circular Runout
 - Total Runout
 - Features That May be Toleranced Using Runout
 - Using Multiple Datum Features to Establish a Datum Axis
 - Full Indicator Movement
- Profile Tolerances
 - Profile of a Line
 - Profile of a Surface
 - The Power of Profile
 - Features that May be Toleranced Using Profile
 - Specifying Limited Length or Area of Application
 - Profile of a Surface vs. Profile of a Line
 - Defining the Extents of Application
 - Equal-Bilateral Profile Tolerancing
 - Unequal Bilateral Profile Tolerancing
 - Unilateral Profile Tolerancing
 - All-Around Profile Tolerancing
 - Specifying Profile Between Points
- Examples – Presented throughout the course
- Open Discussion
 - Discussion of Client Parts, Assemblies, Drawings and Procedures or as requested
- Conclusion

