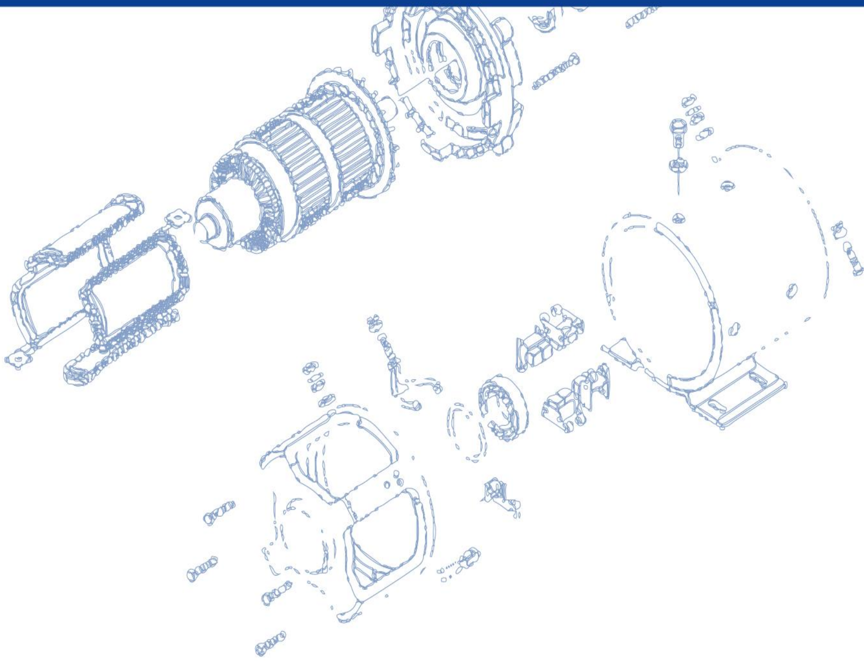
LEDA
Precision·Delivered

Precision Defined, Risk Undermined.



MANUFACTURING GUIDE STANDARDS

What it means for you: Your drawings define the specifications;
our standards define the certainty. We translate your requirements into a controlled manufacturing process, eliminating variables and preventing costly surprises, so you receive parts that fit, function, and assemble—every time.

目录

LEDA Intelligent Design for Manufacturing Guide (DFM Guide)
—— Designing for Manufacturability, Reliability, and Cost-Effectiveness

Preface: Collaborate with LEDA to Gain a Competitive Edge from the Design Source

In the grand blueprint of hardware innovation, the most exciting breakthroughs often begin with an ingenious concept, while the most frustrating setbacks frequently stem from the invisible gap between design and manufacturing. A design that is perfect in simulation may fail in production due to inefficient manufacturability; a prototype that performs excellently in the lab might falter during mass production ramp-up due to minor process variations.

We believe that exceptional products are not only "designed" but also "manufactured." True competitive advantage arises from the deep integration of design and manufacturing at the earliest stages.

This is precisely the purpose behind creating this guide.

LEDA Precision is not only your trusted manufacturing partner but also the guardian and co-creator of "manufacturability" in your product development process. Our mission is to leverage nearly two decades of accumulated engineering expertise and manufacturing data to help you anticipate and avoid future production challenges as early as your first sketch. We call this **"Precision by Design"**—ensuring precision originates from design, not just post-production corrections.

Why is DFM So Critical?

A widely recognized industry principle states: Up to 80% of a product’ s final cost is locked in during the design phase. This means that a minor, forward-thinking design optimization early on can yield cost savings, quality improvements, and cycle time reductions that are tens or even hundreds of times greater than incremental improvements made during production. DFM is not a constraint on your creativity but rather the wings that make it achievable and profitable.

In this guide, you will gain not just a "checklist of design specifications," but the core principles and logic of LEDA’ s engineering team:

Systematic Knowledge: We detail practical design rules for processes ranging from CNC machining and injection molding to 3D printing, organized by chapter.

Cost Insight: Reveal how design decisions profoundly influence mold costs, unit costs, and assembly efficiency.

Industry Perspectives: Extract specialized design considerations for sectors such as aerospace, medical

devices, and consumer electronics.

Common Language: Establish a clear and efficient communication foundation between design and manufacturing teams.

We sincerely invite you to consider LEDA as an extension of your R&D team from the very start of your next exciting project. Let us begin the dialogue at the concept stage, turning challenges into opportunities and transforming ideas into market success with precision, efficiency, and reliability.

May this guide become a valuable tool on your desk, and we look forward to partnering with you to define excellence in the next generation of products, starting from the design source.

LEDA Precision Engineering Team

1.1 Welcome Letter: Your Success Begins with Excellence in Manufacturing

Dear Partner,

Welcome to the collaborative journey with LEDA Precision. As you open this guide, it signifies your commitment to transforming exceptional ideas into tangible market success. We firmly believe that great products stem not only from ingenious design inspiration but also rely on the manufacturing capability to translate that inspiration into physical entities with precision, reliability, and efficiency.

In the extensive chain of product value realization, manufacturing is the central link that carries forward what comes before and initiates what follows. It is the physical expression of design language, the ultimate carrier of performance parameters, and the direct shaper of user experience. A design perfect on paper cannot realize its value if it cannot be manufactured stably and economically; whereas an exceptional manufacturing partner can safeguard your design intent, even infusing additional value through process expertise.

At LEDA, we regard "manufacturing" as both a rigorous engineering science and a creative, empowering art. We possess not only advanced equipment and technology but also a deep understanding of the entire process logic—from material properties and process windows to quality control. Our role is to act as an extension of your R&D team, serving as the most faithful "interpreter" and "realizer" of your design blueprint.

This Design for Manufacturing Guideis the first "collaborative key" we deliver to you. It condenses the practical wisdom and systematic methodologies accumulated by the LEDA team through serving global innovators. Through it, we aim to establish a common language with you regarding "excellence in manufacturing," embedding the genes of "manufacturability," "reliability," and "cost-effectiveness" into every detail of your product from the very earliest stages of the project.

Your success is our ultimate objective. And all of this begins with our shared belief and commitment to "excellence in manufacturing" at this very moment.

LEDA Precision is ready.

1.2 The Core Value of DFM: Why is 80% of the Cost Locked in During Design?

In the field of product development, a widely validated principle states: Up to 80% of a product's total final cost is determined during its design phase. This figure is not an exaggeration; it reveals the most leveraged decision point within the entire value chain from concept to market.

The key to understanding this lies in recognizing that costs are not merely generated on purchase orders or production work orders; they are "pre-set" by a series of early, fundamental design choices.

How Does Design "Lock In" Cost?

Path Dependency of Materials and Processes: The materials you choose (e.g., specialty alloy vs. general-purpose material) and the process paths you decide on (e.g., 5-axis CNC vs. casting) directly define the fundamentals of the supply chain, equipment investment, and processing cycle, forming the "hard framework" of cost.

Multiplicative Effect of Complexity and Precision: Every non-standard surface, excessively tight tolerance requirement, and additional assembly step is amplified in the manufacturing stage, translating into more machining time, more expensive tooling, more complex fixtures, and lower yields. Complexity is an amplifier of cost.

Irreversible Commitment of Molds and Tooling: For processes like injection molding and die casting, once mold manufacturing begins, most of its cost is already incurred. Design changes leading to mold modifications cost far more than adjusting lines on an early-stage drawing. This is the most intuitive manifestation of cost locking.

Setting of Supply Chain and Procureability: The design determines the specificity and standardization level of required components. The procurement cost, lead time, and supply chain risk of a highly customized part are predetermined from the moment the design is finalized.

The cost of late-stage changes is staggering. Research finds that identifying and correcting a problem during the design phase might cost 1 unit; correcting it during the prototyping phase might cost 10 units; if discovered on the production line, the correction cost could soar to 1000 units. This is not only a waste of money but also a loss of crucial time windows and a drain on team morale.

Therefore, the core value of DFM lies in pushing the gate of cost control as far forward as possible. It is a "virtual production" and "stress test" conducted in the digital world at the lowest cost. Through DFM, we collaboratively identify and optimize those risk points during the design stage that could lead to soaring costs, quality fluctuations, and delivery delays later on, transforming "cost locking" into "value locking."

At LEDA, the DFM analysis we provide acts as a "manufacturing lens" for you, making invisible costs visible, controllable, and adjustable before they become reality. This not only reduces manufacturing costs but, more importantly, lays the most deterministic foundation for the commercial success of your product.

1.3 How to Use This Guide

This guide is intended to be a practical desk reference, not a manual left gathering dust on a shelf. To help you maximize the utilization of the knowledge and insights within, we suggest the following approach:

First, understand its structure:

This guide is organized logically from "Philosophy" to "Practice," and then to "Application."

Parts I & III (Philosophy & Special Topics): Recommended for a comprehensive read. Regardless of your focus on specific processes, this section will help you build a systematic DFM mindset and understand LEDA's collaborative philosophy and the underlying logic of quality, cost, and reliability.

Part II (Core Processes): Use it as a technical dictionary for on-demand queries. You can directly jump to the relevant chapter based on the manufacturing process used in your current project to obtain specific, actionable design specifications and recommendations.

Parts IV & V (Application & Resources): Treat these as catalysts for inspiration and quick-reference tools. Industry key points and success cases can spark ideas, while the checklists and tables in the appendices allow for rapid design validation.

Second, clarify your role:

If you are an R&D Engineer or Designer: Please focus on the process chapters directly related to your project and study Chapters 9, 10, 11, and 12 in depth. This will comprehensively enhance the manufacturability and commercial value of your designs.

If you are a Project Manager or Product Owner: It is recommended to prioritize reading Part I and Chapter 11 to grasp the core concepts of managing project costs and risks through DFM. The case studies in Chapter 14 can provide strong support for decision-making and negotiation.

If you are new to LEDA or DFM: Start with the Preface and Chapter 1. It will establish the foundational mindset for efficient collaboration with LEDA.

Finally, treat it as a starting point for dynamic collaboration:

This guide is a crystallization of LEDA's engineering experience, but it is not a substitute for direct communication between us. The most effective way to use it is to discuss the key points within with LEDA's engineers early in your design phase. You can use this guide as the common linguistic and technical framework for our discussions.

We encourage you to take notes and ask questions as you read. When you are ready, Chapter 2 will guide you on how to take the first step in collaborating with us.

Part 1: LEDA DFM Core Philosophy & Collaborative Process

At LEDA, we firmly believe thatExcellence in Manufacturing begins withExceptional Design, andExceptional Design must coexist synergistically with profound manufacturing insight. This section will elaborate on the core philosophies that guide all our engineering practices and clarify how to embark on an efficient and transparent collaborative journey with us.

Our goal is to establish a partnership with you, based on a common language and mutual understanding, right from the project's inception. This allows us to transform potential risks into definitive advantages and jointly navigate the complex journey from concept to mass production.

Chapter 1: Our DFM Philosophy – Precision by Design

At LEDA, we regard DFM as a strategic investment, not a post-design compliance check. The core of our philosophy is **"Precision by Design"** – that is, systematically embedding thegene of "precision" into every stage of the product throughForward-thinking design and collaboration, rather than merely pursuing control during the manufacturing phase. This signifies a paradigm shift in thinking: moving from a linear "design-then-manufacture" process to an integrated co-creation approach of "designing for manufacturability, achieving precision through design."

1.1 From "Manufacturable" to "Easily Manufacturable"

Traditional DFM often stops at answering the question, "Can this design be made?" At LEDA, we set a higher standard: we are committed to answering, "How can it be made in the optimal way?" We not only assess feasibility but also proactively utilize our rich process database and simulation tools to seek the optimal balance across multiple dimensions such as quality, cost, efficiency, and scalability. What we explore with you is the ideal path leading to a design that is "easily manufacturable, easily assemblable, easily inspectable, and with the lowest total cost."

Our Four Pillars of "Easily Manufacturable":

Predictability: Before manufacturing begins, we use tools like mold flow analysis, tolerance stack-up analysis, and cutting simulation to predict and visualize potential issues (such as short shots, deformation, interference), reducing the cost of trial-and-error to the virtual world.

Scalability: We evaluate whether the design can transition smoothly from prototype to mass production. For example, a delicate prototype relying on manual finishing is "manufacturable," but a design incorporating guides and error-proofing features for automated assembly is "easily manufacturable."

Affordability: We conduct "cost-driver analysis" to identify design features that most significantly impact total cost (e.g., a non-standard tiny aperture may require expensive special tooling and multiply machining time) and collaborate with you to optimize, pursuing the best balance between performance and cost.

Sustainability: We factor in manufacturing efficiency and environmental impact. An "easily manufacturable" design also implies less material waste, lower energy consumption, and a more streamlined supply chain.

A Simple Comparison:

"Manufacturable" Answer: "This deep-pocket part can be machined, but it requires custom extended-length tools and the machining time will be very long."

"Easily Manufacturable" Answer: "We recommend reducing the depth of this pocket by 20% or adding a process hole at the bottom. This allows us to use standard tools, reduces machining time by 65%, and completely eliminates the quality risk caused by tool chatter, with the total cost expected to decrease by 40%. Here is our comparative simulation data."

1.2 Quality is Designed In: Shifting Quality Control Upstream

In traditional thinking, quality control is often seen as an end-of-line activity – screening for qualified products through inspection. However, in LEDA's philosophy, true quality cannot be "inspected into" a product; it must be "designed and manufactured in." Our core strategy is to systematically shift the 关口 of quality control upstream to the earliest stages of product development, building a robust line of defense at the design source.

Quality cannot be "detected" at the end of the production line; it must be "designed in." Through DFM, we significantly shift quality control upstream to the design and process planning stages. We analyze the design's sensitivity to process variation, identify potential quality risks (such as warpage, sink marks, stress concentration), and eliminate or reduce these risks through design optimization. Our goal is to make the production process itself highly robust, ensuring product consistency and reliability from the source, rather than relying on downstream screening and rework.

Why Must Control Be Upstream?

The cost of quality issues increases exponentially with the delay in discovery. A problem that requires only a drawing modification during the design phase might lead to tens of thousands in modification costs and weeks of delay if it remains until the mold stage; if it erupts during mass production, it could cause line stoppages, batch scrap, customer complaints, or even catastrophic damage to brand reputation. Therefore, the most economical and effective investment in quality is to eliminate all known risks before the first physical part is even manufactured.

How Does LEDA Achieve "Quality Upstream"?

Design Reviews Based on Failure Prevention: We go beyond checking drawing specifications; we employ a mindset of Potential Failure Mode and Effects Analysis (FMEA), systematically questioning: Is this thin-walled structure prone to short shots during injection molding? Could this sharp corner become the origin of a fatigue crack under stress? Might this assembly gap fail under temperature variation? By simulating and answering these questions during the design phase, we proactively prevent failures rather than react to them.

Designing for Manufacturing Process Robustness: We deeply understand that inherent variations exist in the production environment. Our DFM analysis strives to optimize the design to be more tolerant of unavoidable process fluctuations (such as material batch differences, minor tool wear, ambient temperature changes). For example, by optimizing geometry to reduce sensitivity to injection pressure or machining clamping force, we ensure qualified parts can always be produced within the specified process window.

Design as Data, Data as Control: We translate design intent into executable, data-driven quality control points. Critical dimensions and tolerances are not only 标注 on drawings but are also directly linked to the programming of our Statistical Process Control (SPC) and fully automated inspection equipment. This means that, starting from the first prototype, we collect data and verify process capability, ensuring the "quality" defined by the design can be stably and accurately achieved.

Collaboration with You: Building a Common "Quality Language"

The success of quality upstream relies on our close collaboration with you during the early design stages. We need to understand every requirement you have for the product's function and translate these requirements into manufacturable and measurable design features. What we build together is not just a set of drawings, but a clear, unambiguous "quality language" and control benchmark that runs through the entire product lifecycle.

By choosing to collaborate with LEDA, you gain not just an "inspection service," but a product design with high-quality genes built-in. This 意味着 fewer field failures, lower warranty costs, higher customer satisfaction, and a strong brand reputation built on a foundation ofExceptional quality.

1.3 Cost is Designed In: A Total Lifecycle Cost Perspective

Our focus extends beyond the unit price of a component to its **Total Cost of Ownership (TCO)**. This includes investment in molds/tooling, production yield, assembly efficiency, maintenance costs, and even end-of-life disposal. During the DFM phase, we collaborate with you on "cost modeling," which reveals the impact of different design choices on costs across the entire chain. For instance, adding a feature might reduce machining costs but increase mold complexity; relaxing a tolerance might significantly improve yield. We provide data-driven insights to support your most economical business decisions.

At LEDA, we understand your focus on the "unit price," but our professional insight tells us: true cost control begins with the systematic management of **Total Cost of Ownership**. An apparently attractive low unit price may hide high mold investment, low production yield, complex assembly processes, or frequent field failures. Therefore, a core task of our DFM is to work with you to look beyond simple quoted prices,透视 and optimize the product's total lifecycle cost from concept to decommissioning.

What is "Total Lifecycle Cost"?

It far exceeds the price on a purchase order; it is a comprehensive model comprising both explicit and implicit costs:

One-Time Costs: Development and manufacturing costs for molds, fixtures, and inspection gauges.

Production Costs: Raw materials, direct labor, equipment depreciation, energy consumption, and costs associated with production yield loss.

Application Costs: Assembly complexity, testing time, packaging, and logistics costs.

Usage and Maintenance Costs: Product reliability in the customer's hands, maintenance convenience, and spare part availability.

End-of-Life Costs: Product disposal or recycling costs.

LEDA's "Design-Side Cost Optimization" Methodology

We use the DFM process to identify and optimize these cost drivers upfront during the design stage:

Cost Driver Mapping and Analysis: When reviewing your design, we look not only at the geometry but also analyze the cost drivers behind each feature. For example, a small non-standard internal fillet may require expensive custom tools and multiply machining time; an unnecessarily high surface finish requirement might increase polishing costs by 300%. We will clearly indicate the specific impact of decisions on mold complexity, processing cycle time, material utilization, and scrap rate.

Data-Driven "Cost Modeling" and Scenario Comparison: We go beyond qualitative analysis. Leveraging our internal process database and historical project data, we can perform quantified comparisons of key design choices. For example: "Option A (more complex single part) has higher mold costs but lower assembly costs; Option B (comprising two simpler parts) has lower mold costs but requires additional connection steps and components. Based on your projected volume, Option B has a lower total cost at 50,000 units, while Option A is more economical above 100,000 units." We will present these trade-offs with data.

Designing for "Producibility" to Lock in Mass Production Costs: We focus on the decisive impact of design on mass production yield and efficiency. A design sensitive to process variation will lead to continuous yield loss and adjustment time during mass production. We optimize the design (e.g., by adding reasonable tolerances, avoiding thick walls that are difficult to fill or cool) to enhance process robustness, thereby locking in stable, predictable mass production costs and avoiding subsequent endless "fire-fighting" cost waste.

Collaborating with You: Making Informed Business Decisions

Our role is not to make decisions for you, but to act as your manufacturing 智库, providing critical decision support. In DFM collaboration, we will:

Reveal Cost Trade-offs: Clearly demonstrate the correlation between "X% performance improvement" and "Y% cost increase."

Focus Key Levers: Identify the 1-2 design factors that have the greatest impact on total cost and concentrate optimization efforts there.

Provide Preferred Solutions: Typically, we offer multiple optimized alternative solutions that meet core requirements, accompanied by their respective cost, cycle time, and risk analysis, supporting you in making the most informed business decisions based on project priorities (whether cost-sensitive or time-to-market driven).

Ultimate Goal: Through in-depth DFM collaboration with LEDA, you will obtain a design that achieves the optimal balance between performance, quality, cost, and manufacturability. This not only means a lower purchase price but, more importantly, a product with optimized total cost and minimized commercial risk throughout the project lifecycle, laying a solid financial foundation for your market competitiveness.

1.4 LEDA DFM Service Process: From File Upload to Production Release

Our DFM service is not a one-time report, but a structured, transparent collaborative process:

Preliminary Analysis & Rapid Feedback: You upload your design to the platform; the system, leveraging AI and a rules database, provides initial manufacturability risk alerts and an instant quotation within hours.

In-depth Engineering Review: Your dedicated project manager and a team of process engineers conduct a thorough review, including mold flow/stress analysis, and engage in online or offline workshops with your design team.

Optimization Solutions & Joint Decision-Making: We provide clear optimization options (A/B/C), impact assessments (cost/cycle/performance), and recommendations, supporting collaborative decision-making.

Data Handover & Production Preparation: The finalized design data is seamlessly transferred into our Manufacturing Execution System (MES). Relevant DFM knowledge is translated into specific process control plans, ensuring the optimization intent is faithfully executed during mass production.

At LEDA, we believe efficient collaboration requires a clear pathway. Therefore, we translate our forward-looking DFM philosophy into a standardized, end-to-end collaborative workflow. This is not merely a series of checkpoints, but a systematic project involving deep integration, knowledge transfer, and ultimately, the transformation of design intent into production reality. Our DFM service accompanies your project throughout its entire journey, until successful mass production is achieved.

Phase 1: Instant Analysis & Preliminary Feedback (Within Hours)

Your Action: Upload your 3D models, 2D drawings, and relevant technical requirement documents via the LEDA online collaboration platform.

LEDA's Action: The platform system automatically triggers analysis. Combining AI algorithms with our core process rules database, it generates a preliminary Design for Manufacturability report and provides an instant quotation based on the model geometry within hours. This report highlights potential "critical alerts" (e.g., unmanufacturable features, clear violations of design guidelines) and "advisory notes" (designs that may increase cost or risk).

Deliverables: Preliminary DFM Alert Report, Project Instant Quotation, and contact information for your dedicated project manager. This provides a rapid feasibility assessment and budget reference for your project.

Phase 2: In-depth Engineering Review & Collaborative Workshop (1-3 Business Days)

Your Action: Confirm the initiation of the in-depth review with the LEDA team. Your design team (mechanical, electronic, etc.) prepares to participate in technical discussions.

LEDA's Action: Your dedicated project manager coordinates internal resources to form a cross-functional engineering team (including process, mold, and quality engineers). This team conducts a manual, in-depth review of your design and may initiate specialized engineering simulation analyses. Subsequently, we schedule an online or offline collaborative workshop to discuss design details, functional requirements, potential risks, and optimization directions face-to-face with your team.

Deliverables: Detailed DFM Review Meeting Minutes, Simulation Analysis Report (if applicable), and a mutually agreed-upon list of key issues and optimization directions.

Phase 3: Optimization Solutions, Joint Decision-Making, and Design Freeze (Typically 3-7 Business Days, depending on complexity)

Your Action: Based on the review feedback, proceed with design adjustments. The LEDA team provides synchronous support.

LEDA's Action: Based on the workshop conclusions, we prepare a formal DFM Optimization Proposal containing comparative solutions. This document clearly outlines different optimization paths (e.g., Option A focuses on cost, Option B emphasizes performance) along with their quantified impact assessment on mold investment, unit cost, production cycle, and product performance. We review these options with you to support the decision that best aligns with your project goals.

Deliverables: Formal DFM Optimization Proposal, Final Design Change List, and the mutually signed Design Freeze document. This marks the end of the design phase and the beginning of the manufacturing phase.

Phase 4: Seamless Data Transfer & Production Preparation

Your Action: Confirm the final design data package.

LEDA's Action: The approved design data (3D/2D) is securely and completely imported into our Manufacturing Execution System (MES). More importantly, all knowledge generated throughout the DFM process — including key quality control points, special process parameters, and assembly considerations — is systematically translated into production control plans, work instructions, and inspection specifications. These documents directly guide our mold manufacturing, production processing, and quality inspection, ensuring the design intent we co-optimized is accurately realized in mass production.

Deliverables: Formal confirmation for production start, production scheduling plan, and subsequent standardized deliverables according to plan, including mold manufacturing, first article inspection, pilot production, and mass production ramp-up, until the product is successfully released for full production.

Core Value: Choosing LEDA's DFM process means selecting a manufacturing partner that is transparent, predictable, and accountable for results. We define success together from the very beginning and share the roadmap to achieve it.

Chapter 2: Kick-off Collaboration – Design Data Submission Standards

Clear, accurate, and complete design data serves as the "common language" for our efficient collaboration and is the starting point for a successful project. A standardized data package ensures that our engineering team can precisely understand your design intent, thereby providing the most reliable manufacturability analysis and quotation at the earliest opportunity. This helps avoid project rework and delays caused by missing information or misunderstandings.

This chapter details the design data standards required to initiate collaboration with LEDA. We recommend that you use this chapter as a pre-flight checklist once your design is substantially complete, beforeformally requesting a quotation and DFM analysis, to ensure your data meets all requirements for a rapid process initiation.

Adhering to these standards means we can embark on a professional and smooth collaborative journey within hours, not days.

2.1 Essential File List: 3D Model, 2D Drawing, Technical Requirements

To ensure the LEDA engineering team can fully and accurately understand your design intent and provide the most valuable manufacturing analysis, we advise preparing the following three core documents when submitting a project inquiry. Together, they form a complete technical data package, each being indispensable.

File Type	Core Purpose and Requirements	Submission Standards Summary
1. 3D Digital Model	Defines the geometric shape and spatial relationships of the product. It serves as the sole geometric basis for all manufacturing analysis, CNC programming, and mold design.	Must be the final version solid model, not surface or intermediate assembly. Recommended formats: STEP (.stp/.step) or IGES (.igs). The model should include all necessary features and have removed auxiliary geometry unrelated to analysis.
2. 2D Engineering Drawing	Clarifies product dimensions, tolerances, process requirements, and acceptance criteria. The drawing is the legal document of design intent and the basis for quality inspection.	Must be in PDF format. The drawing should clearly include all necessary views, complete dimensions and geometric tolerances, technical requirement notes, surface treatment symbols, material grade, and title block information.
3. Technical Requirements Document	Describes non-geometric requirements such as product function, performance, testing, and compliance. These requirements determine the	Can be a separate Word/PDF document or detailed in the "Technical Requirements" section of the 2D drawing. Must specify key performance indicators, testing standards, industry

	selection of materials and processes.	certifications, operating environment, special processes, etc.

Detailed Specifications:

A. 3D Digital Model

Format: We prioritize and recommend the STEP AP 214 (.stp/.step) format, as it most completely preserves information on solids, surfaces, and assembly structures, offering superior cross-software compatibility. The IGES format (.igs/.iges) can be used as an alternative. Please avoid sending native CAD files (e.g., .sldprt, .prt) directly unless confirmed by both parties.

Content: The model should represent the final design state of a single part or a complete assembly. For assemblies, please provide independent models for all key components simultaneously. It is essential to "clean up" the model by removing manufacturing-irrelevant sketches, datum planes, surfaces, hidden bodies, and obsolete versions.

Integrity: The model should accurately reflect the design. All features such as chamfers, fillets, and threads must be modeled explicitly, not just defined in the 2D drawing.

B. 2D Engineering Drawing

Completeness: The drawing is the instruction for manufacturing and inspection. Please ensure it includes:

Multiple Views: Sufficient orthogonal views, section views, and detail views to fully represent the part's shape.

Complete Annotation: All critical dimensions and geometric tolerances that determine product function and assembly.

GD&T Annotation: Rational use of Geometric Dimensioning and Tolerancing (e.g., flatness, position, profile) to define functional datums and relationships. This provides better assurance for assembly quality compared to solely using plus/minus tolerances.

Title Block Information: Part number, revision, material, designer, scale, and other information must be filled in completely and accurately.

Clarity: The exported PDF should ensure all lines and text are clear and legible, without blurring or overlap.

C. Technical Requirements Document

The content should be specific and unambiguous, typically including but not limited to:

Material & Performance: Clear material grade, standard (e.g., ASTM, ISO), and mechanical property requirements.

Surface Treatment: Anodizing type and thickness, coating type and thickness, roughness values, etc.

Heat Treatment: Hardness requirements, processing technology, etc.

Inspection & Testing: Full dimensional inspection, functional testing, pressure testing, non-destructive testing (NDT), and other special requirements.

Compliance: Industry-specific standards that need to be met.

Appearance Standards: Acceptable criteria for appearance defects.

Submission Recommendation:

Consolidate the above three types of files into a single folder using a clear naming convention (e.g., ProjectName_PartNumber_Revision_Date), and upload via the LEDA online platform. A standardized, complete data package is the best guarantee for receiving a rapid and accurate response.

2.2 Model & Drawing Standards (Format, Version, Units)

Clear, unified data standards are the foundation for avoiding communication errors and ensuring manufacturing accuracy. This section defines the specific requirements for 3D models and 2D drawings regarding file format, version control, and units of measurement when collaborating with LEDA. Adhering to these standards ensures your design data is accurately interpreted by our systems and can be directly applied to production and inspection.

Category	Core Standard	Explanation & Rationale
1. File Format		
Preferred 3D Format	STEP AP 214 (.stp, .step)	As an internationally recognized neutral format, STEP best preserves solid entities, assembly structure, color, and Product Manufacturing Information (PMI). It is compatible with all major CAD/CAM software, making it the most reliable choice for data exchange.
Acceptable 3D Format	IGES (.igs, .iges)	Can be used as an alternative, but carries a risk of information loss during the conversion of solid and surface data. Use STEP format preferentially when possible.
2D Drawing Format	PDF (.pdf)	Drawings must be submitted in PDF format to ensure consistent display on any device and prevent unintended modifications. Ensure the printed or exported PDF is clear and complete.
2. Version Control		
File Naming Rule	ProjectAbbr_PartNo_Revision_Date.pdf/.stp (e.g.: ProjectA_HSG-001_RevC_20240527.stp)	A clear naming convention is key to avoiding version confusion. We strongly recommend including a revision identifier (e.g., RevA, B, C or V1.0, V1.1) and date in the filename.
Design Change Sync	Must synchronously update all related files.	When a design change occurs, all related files (3D model, 2D drawing, technical requirements document) must be updated and resubmitted simultaneously. Do not merely describe modifications via email, as this 极易 leads to production errors.
Single Source of Truth	Ensure 100% consistency between model and drawing.	The submitted 3D model must be the exact geometric source depicted in the 2D drawing. Both must correspond completely in geometry, features,

Category	Core Standard	Explanation & Rationale
		and revision.
3. Units of Measurement		
Unified Unit System	Mandatory use of the International System of Units (SI).	Within the same project, all modeling units for 3D models and annotation units for 2D drawings must be unified to millimeters (mm). Mixing imperial and metric units is a common cause of serious manufacturing errors.
Internal Model Units	Model properties must be correct.	When creating the model in CAD software, its internal units should be set to millimeters. A model created in an "inch" environment but annotated in "millimeters" will be scaled by a factor of 25.4 when imported into our system.
Drawing Annotation	Clearly annotate units.	Although unified use of millimeters is the premise, the title block of the drawing must still clearly note "UNITS: MM".

Summary of Key Requirements:

Format: 3D: STEP; 2D: PDF.

Version: Include version numbers in filenames; synchronize updates for all files upon design changes.

Units: Use millimeters consistently throughout, and verify the model's internal unit properties.

Common Issues & Consequences:

Error: Submitting a native-format assembly file without including all component parts.

Consequence: Requires extra time for processing/decomposition, potentially losing associations and delaying analysis.

Error: Drawing annotations are in millimeters, but the 3D model was created using an imperial template.

Consequence: The part will be manufactured at 1/25.4 of its intended size, causing complete scrap.

Error: Informing via email to "please refer to the latest model" without updating the drawing revision.

Consequence: Production and inspection will be based on an obsolete drawing, potentially causing batch quality incidents.

Adhering to this standard is the first and most important quality checkpoint to ensure a seamless project journey from design to manufacturing.

2.3 How to Clearly Define Key Characteristics and Tolerances

Clearly and unambiguously defining the key characteristics and tolerances of a part is the most precise "technical language" between design and manufacturing. It directly determines: whether the part can achieve its intended function, whether it can be manufactured efficiently and economically, and whether it can ultimately be accurately inspected.

Vague or excessively tight tolerance requirements are among the most common causes of soaring costs, delivery delays, and quality disputes. This section aims to guide you on how to define them scientifically and standardly.

1. Identifying and Defining "Key Characteristics"

Key characteristics are those dimensional, form, or location elements that have a decisive impact on the product's assembly, function, performance, safety, or appearance.

Functional Dimensions: Such as the fit diameter between a shaft and a bearing, the flatness of a sealing surface, or the pitch of a gear.

Assembly Datums: Features used for locating and mounting other parts, such as the position of locating holes or the flatness of a mounting surface.

Safety and Compliance Dimensions: Those involving electrical clearance, creepage distance, or regulatory requirements.

Appearance Surfaces: Surfaces directly visible to the user, with specific requirements for their roughness, texture, or color difference.

On drawings, key characteristics should be clearly identified, for example, by encircling their dimensions with special symbols (e.g., a box □) or listing them separately in the technical requirements section to draw the high attention of manufacturing and inspection personnel.

2. Adopting a Scientific Tolerance Annotation System: Prioritizing GD&T

We strongly recommend and prioritize the use of **Geometric Dimensioning and Tolerancing (GD&T, per ASME Y14.5 or ISO 1101 standards)** for drawing annotation. Compared to traditional plus/minus tolerances, GD&T offers significant advantages:

Annotation Method	Core Philosophy	Advantages	Example Illustration
Traditional ± Tolerance	Controls the absolute position of a feature "point".	Simple and intuitive, suitable for simple features.	The position of a hole is labeled as: 10±0.1mm from the edge.
GD&T Tolerance	Controls the shape, orientation, and position of the feature "itself" relative to a "Datum Reference Frame".	Function Driven: Defines how the part assembles and functions. Cost Optimization: Often allows for a wider manufacturing tolerance zone than ± tolerances while ensuring assembly. Eliminates Ambiguity: Clearly defines inspection methods and acceptance	The position of the hole is labeled as: Position Ø0.2mm relative to Datum [A, B, C]. This ensures the hole's axis must lie within a cylindrical tolerance zone centered on the true position with a diameter of 0.2mm, simultaneously considering the feature's form and orientation.

Annotation Method	Core Philosophy	Advantages	Example Illustration
criteria.			

Core Elements of GD&T:

Datum: Establishes the theoretical reference frame for the part in inspection and assembly (typically three mutually perpendicular datum features A, B, C).

Feature Control Frame: Contains the geometric characteristic symbol (e.g., position ϕ, flatness ⌓), tolerance value, and possibly material condition symbols (e.g., Maximum Material Requirement Ⓜ).

3. Best Practices for Tolerance Annotation

Avoid "Over-Tolerancing" and "Under-Tolerancing":

Do not assign the tightest tolerance to all dimensions. This makes the part extremely expensive and difficult to manufacture.

Ensure all functional requirements are unambiguously expressed through dimensions and tolerances. Non-critical dimensions can be labeled as "free tolerance" or reference a general tolerance standard (e.g., ISO 2768-m).

Ensure Tolerances are Reasonable and Manufacturable:

Understand the economical machining accuracy of different processes. For example, requiring ± 0.005mm from conventional CNC is feasible, but requiring the same accuracy from a sand casting is unreasonable.

The relationship between tolerance and cost is exponential. Use the loosest possible tolerance that satisfies the function.

Consider Assembly Stack-up:

For assemblies comprising multiple parts, dimensional chain analysis is necessary to properly allocate tolerances among the parts, ensuring the final assembly's function.

Clarify the Relationship between Surface Treatment and Tolerances:

If a part requires surface treatment (e.g., plating, painting, anodizing), the drawing must clearly specify whether the final dimensional requirements are "before plating" or "after plating." Because plating adds thickness, it affects key fit dimensions.

4. Collaboration with LEDA: From Drawing to Perfect Part

When you provide a clearly defined drawing, LEDA's engineers can:

Provide Accurate Quotations: Immediately assess manufacturing difficulty, required processes, and equipment, offering a reasonable price.

Generate Efficient Programs: Create optimal tool paths for CNC machining or mold design and process parameters.

Customize Inspection Plans: Design special gauges or program CMM measurements based on your GD&T requirements, ensuring inspection results fully align with your design intent.

An excellent engineering drawing is half the success of manufacturing. After you complete your design, LEDA's DFM team is very willing to provide a pre-review consultation of your drawings, helping you optimize tolerance annotation and find the best balance between performance and cost.

5. LEDA Online Platform: Gateway for Instant Quotation and Preliminary DFM Analysis

To effectively implement our "Precision by Design" philosophy and collaborative process, LEDA Precision has developed an intelligent online collaboration platform. This is the official digital entry point for initiating cooperation with LEDA, designed to compress the traditional quotation and preliminary technical communication process from days or even weeks down to hours, allowing your project to start with the highest efficiency.

Platform Core Entry

Official Website: By visiting the LEDA Precision website (www.ledaprecision.com), you can prominently find the "Get Quotation" or "Customer Portal" entry in the navigation bar. After registering and logging in, you can use the complete project collaboration features.

Convenient Login: Supports enterprise email registration and verification, ensuring professionalism and security in business communication.

Platform Core Functions and Service Process

Secure, Convenient File Upload

Operation: After logging in, you can directly create a new project within the platform and drag-and-drop to upload your 3D models (STEP/IGES), 2D drawings (PDF), and other technical documents.

Security: All uploaded data is transmitted via encrypted links and stored in a secure enterprise-grade cloud environment. We strictly adhere to confidentiality agreements; your design files are protected with the highest level of security.

AI-Driven Instant Analysis and Quotation

Intelligent Analysis: After upload, our backend system automatically activates a preliminary analysis engine based on a rules database and algorithms. Within minutes, the system performs a basic manufacturability review of the model, for example:

Identifying potentially unmanufacturable features (e.g., undercuts, excessively deep or thin holes).

Checking model integrity (e.g., broken surfaces, invalid solids).

Preliminary matching of recommended processes and materials.

Instant Quotation: Based on the preliminary process judgment from the intelligent analysis, the material cost database, and real-time production capacity data, the system will generate a clear, geometry-based preliminary quotation within 1-2 hours. This quotation will clearly list the unit price, estimated mold cost (if applicable), and price tiers for different quantity ranges.

2.4 Structured Preliminary DFM Report

Along with the quotation, the system automatically generates a visual preliminary DFM report. This report intuitively highlights potential risk areas within the design using clear annotations and color-coding:

Red Alerts: Indicate critical issues that could render the part unmanufacturable or require extremely high costs (e.g., features necessitating wire EDM or special electrodes).

Yellow Suggestions: Point out optimizable areas that may impact yield, increase cost, or prolong the cycle time (e.g., recommending increased draft angles or optimized wall thicknesses). The report includes optimization suggestions, providing initial direction for your design iterations.

2.5 Dedicated Project Space & Collaborative Management

Each project is allocated an independent online workspace. Within this space, you can:

View all historical quotations and file versions.

Communicate online with your dedicated LEDA project manager and engineers, utilizing features for comments and image annotations to conduct design reviews.

Track the status of key project milestones throughout the entire workflow, from quotation and design optimization to production and shipping.

Value of the Platform

Ultimate Efficiency: Access professional feedback instantly with 24/7 self-service, bypassing waits associated with traditional emails and phone calls.

Front-loaded Decision-making: Validate the manufacturability and economic feasibility of your project through low-cost, rapid preliminary analysis before committing significant engineering resources, thereby reducing early-stage decision risks.

Transparency and Fairness: Quotations are generated based on standardized algorithms, minimizing human fluctuation factors and making cost structures more transparent.

Knowledge Retention: All project communications and files are stored online, forming a complete project knowledge base for easy team management and audit trails.

Subsequent Collaborative Path

After you optimize your design based on the platform's preliminary feedback and decide to move the project forward, simply click to confirm within the platform. Your dedicated project manager and engineering team will immediately engage, initiating the in-depth engineering review and collaborative workshop described in Chapter 2. All information from the online platform is seamlessly transferred, ensuring continuity in collaboration.

By using the LEDA online platform, you are not just taking a simple step, but stepping into an efficient, transparent, and professional digital manufacturing collaboration ecosystem. Upload your design now to experience the "LEDA Speed".

Part 2: In-Depth Explanation of DFM Design Specifications for Core Processes

In Part I, we explored the core philosophy and collaborative workflow of DFM. However, the 落地 of these principles ultimately relies on a profound understanding and practical application of specific manufacturing processes.

"Manufacturing" is not an abstract concept; it is defined by a series of physical and chemical processes (such as cutting, melting, plastic deformation, and curing). Each core process — CNC machining, injection molding, die casting, sheet metal fabrication, additive manufacturing, and others —possesses its own unique technical principles, capability boundaries, economic drivers, and design constraints. True DFM translates "manufacturability" from a slogan into quantifiable, executable design rules tailored to a specific process.

This section serves as a practical toolkit for your in-depth technical dialogues with the LEDA engineering team. We have systematically condensed the engineering knowledge accumulated by LEDA across ten core manufacturing domains into clear design guides. Each chapter is dedicated to a core process, following a logical structure of **"Material Selection → Feature Design → Process Considerations → Cost Trade-offs."** It aims to help you:

Understand the Process Essence: Learn how each process "thinks" and "works," enabling you to design in harmony with its inherent logic .

Anticipate Manufacturing Challenges: Identify and avoid common design pitfalls during the design phase that could lead to high costs, low yields, or long cycle times .

Unlock Optimization Potential: Master strategies for proactively optimizing designs to unlock the full potential of the process, achieving the best balance of performance, quality, and cost .

Enable Efficient Collaborative Communication: Establish the fundamental vocabulary and common understanding necessary for discussing technical solutions with manufacturing experts .

Whether your project involves a single process or a complex combination of technologies, the content in this part provides the specific and critical technical insights required from the conceptual design stage through to the design freeze. We recommend you consult the relevant chapters based on the processes involved in your project and use this section as a key basis for your design self-check and review processes.

Chapter 3: CNC Machining Design Guide

In the realm of precision manufacturing, CNC machining serves as both a "universal tool" and a "benchmark for accuracy." This process utilizes high-speed rotating cutting tools, under the precise control of a computer program, to progressively remove material from a solid blank, ultimately shaping complex three-dimensional geometries. This subtractive manufacturing technology is the preferred choice for applications ranging from rapid prototype validation to the production of high-complexity, low-to-medium volume precision components, owing to its unparalleled flexibility, exceptional accuracy, and excellent material versatility .

However, this very flexibility also means that its cost, efficiency, and quality are highly dependent on upfront design decisions. A design that is merely "CNC machinable" can differ significantly from one that is "optimized for CNC" in terms of manufacturing lead time, cost, and final part performance .

This chapter aims to provide you with a set of manufacturing-oriented design principles for CNC machining. We will move beyond simply "what can be done" to focus on "how to do it better, faster, and more economically." You will learn how adhering to core design principles allows you to:

Significantly reduce machining difficulty and cost: For instance, by optimizing internal fillets to avoid the need for expensive non-standard tooling .

Enhance part rigidity and quality: For example, through rational wall thickness and rib design to prevent machining vibrations and deformation .

Simplify assembly and inspection: Such as by establishing clear, machinable datum features .

We begin with the fundamental decision of material selection, as the material directly dictates tooling choices, cutting parameters, cost, and the final performance of the part. Subsequently, we will delve into feature design specifications, the core of this chapter, covering everything from basic geometric constraints to the handling of complex structures. Finally, we will examine key elements for ensuring part functionality and inspectability, such as tolerances and datum setup .

Whether you are designing a high-load aerospace bracket or a precision component for a minimally invasive surgical instrument, the principles outlined in this chapter are universally applicable. Mastering them will enable you to communicate effectively with LEDA's process engineers on a common wavelength, working together to translate your precision requirements into executable, predictable, and profitable manufacturing solutions.

3.1 Material Selection Considerations: Machinability, Cost, Post-Processing

The material forms the physical basis of the part, and its selection is the primary decision determining the product's performance, reliability, and manufacturing cost. In CNC machining, material selection goes far beyond merely meeting a performance datasheet; it profoundly impacts machining efficiency, tool life, surface quality, subsequent processing, and the total manufacturing cost. This section guides you in systematically evaluating and selecting materials based on manufacturability, economics, and project objectives.

3.1.1 Core Considerations

Material selection is a multi-objective optimization process that requires balancing the following key factors:

Functional Requirements: Strength, hardness, toughness, wear resistance, corrosion resistance, thermal properties, electrical conductivity/insulation, weight, etc.

Machinability: The ease with which a material can be cut, directly affecting machining time, tooling costs, surface quality, and dimensional accuracy .

Cost: Includes raw material procurement cost, machining consumption cost (tool wear, labor time), and potential post-processing costs .

Supply Chain & Procureability: Standard material availability, lead times, supplier stability, and certification requirements for special materials .

Post-Processing Compatibility: How readily the material accepts required surface treatments, heat treatments, welding, or bonding .

CNC Material Categories & Characteristics Comparison

Material Category	Typical Grades (Examples)	Core Properties & Advantages	Machinability Assessment	Key Considerations & Typical Applications
Aluminum Alloys	6061, 7075, 5052	Lightweight, good strength-to-weight ratio, excellent thermal/electrical conductivity, easy to machine, relatively low cost.	Excellent	Fast machining speeds, low tool wear, good surface quality. The preferred choice for prototypes, enclosures, brackets, heat sinks. 7075 offers higher strength but slightly lower corrosion resistance.
Stainless Steel	304, 316, 17-4PH	Excellent corrosion resistance, good strength and hardness.	Fair to Difficult	"Gummy," can cause work hardening, demands higher tooling and parameter control, costlier than aluminum. Used for medical devices, food machinery, chemical components, high-strength fasteners.
Alloy Steel	4140, 4340, A2, D2	High strength, high toughness; properties can be significantly enhanced via heat treatment.	Fair	Requires appropriate tool selection and cutting parameters. Commonly used for molds, shafts, gears, high-strength structural parts.
Tool Steel	P20, H13, S7	Very high hardness, wear resistance, and hot strength.	Difficult	Difficult to machine, requires carbide or CBN tools, high cost. Specifically for injection molding/die casting molds, punches, wear-resistant parts.
Titanium Alloys	Ti-6Al-4V (Gr5)	Excellent strength-to-weight ratio, superior biocompatibility,	Difficult	Poor thermal conductivity leads to heat buildup; low elastic modulus can

Material Category	Typical Grades (Examples)	Core Properties & Advantages	Machinability Assessment	Key Considerations & Typical Applications
		corrosion resistance.		cause spring-back. Requires specialized tools, low speeds, high coolant. For aerospace, medical implants, high-performance racing components.
Copper & Copper Alloys	C110 (Cu), C360 (Brass), C17200 (BeCu)	Excellent electrical/thermal conductivity, corrosion resistance, generally good machinability.	Excellent to Fair	Pure copper is soft and sticky; brass is easy to machine; beryllium copper offers high strength. Used for electrodes, heat sinks, electrical components, bearings.
Engineering Plastics	POM (Acetal), PA66 (Nylon), PEEK, PC (Polycarbonate)	Lightweight, insulating, chemical resistance, self-lubricating, capable of complex shapes.	Excellent to Fair	Low cutting forces, but attention needed to heat dissipation (prevent melting), rigidity (prevent deformation), and clamping force control. For insulating parts, gears, bearings, low-friction parts, chemically resistant components.

3.1.2 Key Decision-Making Guidelines

Avoid "Over-Engineering": Do not default to the highest-performance material. First, clearly define the part's actual operational loads. A bracket used indoors might be perfectly satisfied with 6061 aluminum or standard steel instead of expensive aerospace-grade titanium, potentially reducing costs by over 60% .

Understand the Cost Implications of "Machinability": Materials with poor machinability (e.g., titanium alloys, hardened steels) lead to :

Significantly increased machining time (due to lower cutting parameters).

Sharply rising tooling costs (rapid wear, need for specialized coated tools).

Potential quality risks (deformation, residual stresses).

Decision: Prioritize easily machinable materials where functional requirements are met.

Evaluate Total Lifecycle Cost: Calculate the "Total Cost of Ownership." For example :

Material A has a lower unit price but takes twice as long to machine as Material B → total cost might be higher.

Material B requires expensive surface treatment to meet corrosion resistance requirements.

Material C, while a standard item, has a long lead time (e.g., 8 weeks), potentially delaying the entire project.

Design for Post-Processing: Plan the final state in advance. If a part requires anodizing, prioritize aluminum from the 6xxx or 7xxx series. If induction hardening is needed, select steel with appropriate hardenability. For precision machining of PEEK, consider dimensional changes due to moisture absorption.

3.1.3 Collaboration with LEDA for Optimization

During the conceptual design phase, discuss your performance, environmental, budget, and appearance requirements with the LEDA engineering team. We can provide:

Material Database Support: Recommend proven material-process combinations based on thousands of successful cases.

Machinability Pre-Assessment: Conduct rapid machining simulations for your initially selected materials to predict potential challenges.

A/B Solution Comparison: Provide detailed cost-performance-cycle analysis reports for different material options (e.g., Aluminum vs. Stainless Steel, POM vs. PEEK) to support your optimal business decision.

The correct material choice is the first and most critical milestone on the path to a successful, economical, and manufacturable part.

3.2 Feature Design Specifications

Selecting the appropriate material only defines "what to make." The specific geometric features of the part determine "how to make it" and "the quality of the final result." Feature design is the bridge connecting functional requirements to physical realization, and it is the aspect that most directly impacts manufacturability, cost, quality, and performance .

Unreasonable feature design can lead to manufacturing difficulties, soaring costs, or part failure, even when using the most machinable material. This section details the most critical and common features in CNC machining by categorizing them into three core sets of specifications:

Wall Thickness, Ribs, and Aspect Ratio: Pertain to the structural integrity of the part and machining stability. Excessively thin walls or high ribs are prone to vibration and deformation during machining; overly deep holes or cavities pose extreme challenges for tools and processes.

Internal Corner Radii and Stress Concentration: Pertain to the mechanical performance of the part and tool accessibility. Sharp internal corners are multipliers of stress and origins of cracks, and also necessitate smaller, weaker tools, increasing machining time and cost.

Slots, Deep Cavities, and Tool Interference: Pertain to the feasibility and economy of the machining strategy. These features directly determine the type of tools required, whether machining can be completed in a single setup, and if expensive special processes are needed.

Understanding and applying these specifications is not about restricting your design freedom, but about embedding the "manufacturability" gene into the design from the very beginning. This avoids major design rework later and achieves the design intent via the optimal path. Each will be broken down in detail below.

3.2.1 Wall Thickness, Ribs, Aspect Ratio

Wall thickness, ribs, and aspect ratio are fundamental geometric parameters that determine a part's structural rigidity, machining stability, and final quality. Unreasonable designs directly lead to chatter, deformation, and even damage to the tool or part during machining.

1. Wall Thickness Design Specifications

Wall thickness is the primary factor enabling a part to resist cutting forces and its own weight. Insufficient wall thickness is one of the most common causes of part deformation, vibration, and dimensional inaccuracies in CNC machining.

Material Category	Suggested Min. Thickness	Recommended Design Thickness	Key Considerations & Risks
Aluminum Alloy	0.8 mm	≥ 1.5 mm	Below 1mm, "chatter" vibration easily occurs during machining, leading to poor surface quality. Thin areas may deform permanently due to residual stress or clamping forces. For large parts (>200mm), wall thickness should increase accordingly.
Steel / Stainless Steel	1.0 mm	≥ 2.0 mm	Higher material strength, but cutting forces are greater. Thin walls are more prone to elastic deformation ("tool spring"), making dimensions difficult to control.
Titanium Alloy	1.5 mm	≥ 2.5 mm	Low elastic modulus results in poor rigidity, making it highly prone to bending under cutting forces. Sufficient wall thickness is essential for stable cutting.
Engineering Plastics	1.0 mm	≥ 2.0 mm	Poorest rigidity and sensitive to heat. Thin walls deform easily during machining and ejection, and can melt due to overheating.

Best Practices:

Aim for Uniformity: Adjacent wall thicknesses should be as consistent as possible to avoid warping caused by uneven cooling. Transition areas between thick and thin sections should use a gradual taper.

Increase Local Stiffness: For areas where thin walls are necessary, increase local rigidity by designing reinforcing ribs or rolled edges, rather than simply increasing the entire wall thickness.

2. Rib Design Specifications

Ribs are used to increase stiffness and strength while minimizing weight and material usage. Unreasonable rib design can become a machining challenge. The design principles for rib thickness are similar to those for walls, but special attention must be paid to the height-to-thickness ratio to prevent vibration during machining. Furthermore, the connection between the rib and the main body should be reinforced with a sufficient fillet radius to reduce stress concentration.

3. Aspect Ratio Design Specifications

Aspect ratio is the core metric measuring the relationship between the "depth" and "diameter/width" of a hole, cavity, or boss. It is the direct basis for assessing machining difficulty and selecting tools.

Feature Type	Suggested Safe Aspect Ratio	High Difficulty Range	Risks & Solutions
Milled Pockets/Slots	Depth ≤ 4 × Cutter Diameter	> 4 × Cutter Diameter	Excessive overhang causes a sharp drop in tool rigidity, leading to tool deflection, chatter marks, and inaccurate dimensions. Solution: Use trochoidal milling, extended carbide tool holders, or plunge milling.
Drilled Holes	Through Hole: ≤ 10 × Hole Dia. Blind Hole: ≤ 5 × Hole Dia.	> 10 × Hole Dia.	Difficulty in chip ejection, coolant cannot reach the bottom, leading to chip clogging, rapid tool wear, hole deviation, or tool breakage. Solution: Use specialized deep-hole drills, high-pressure through-tool coolant, peck drilling + reaming.
Turned slender shafts	Length/Diameter Ratio ≤ 8 × Diameter	> 8 × Diameter	The workpiece tends to bend and vibrate during turning, causing "barreling" errors. Solution: Use a follow rest, steady rest, or employ "fly cutting" milling instead of turning.

General Principles:

Larger is Better: Where function allows, maximize the hole diameter/slot width or minimize the depth to reduce the aspect ratio.

Bottom Fillet: The bottom of blind cavities and holes must have a fillet radius ≥ 0.5mm to ensure a tool can enter and finish machining, avoiding stress concentration.

Collaborate with LEDA: For designs exceeding conventional aspect ratios, please inform us in advance. Our process engineers can assess feasibility using custom tools, multi-axis strategies, or changing part orientation, and provide corresponding cost and timeline estimates.

Summary: Robust wall thickness, reasonable ribs, and a controlled aspect ratio are the cornerstones for ensuring your design can be efficiently, stably, and economically transformed into a precision part. Applying these specifications at the conceptual design stage will significantly lower subsequent manufacturing risks and costs.

3.2.2 Internal Corner Radii and Stress Concentration

Stress concentration is not a material defect but a phenomenon of localized stress amplification caused by geometry. When a load passes through a sharp corner, its path changes abruptly, causing high density of stress lines.

Quantitative Relationship: For a tensile plate, the theoretical stress concentration factor K_t can be approximated as $K_t \approx 1 + 2\sqrt{(t/r)}$, where t is a characteristic dimension (e.g., plate thickness) and r is the internal corner radius. This means K_t is inversely proportional to the square root of the radius.

Fatigue Life Impact: Under cyclic loading, a part's fatigue life is highly sensitive to Kt. A sharp corner with Kt=3 can reduce fatigue life to a fraction of that without stress concentration. This impact is catastrophic in dynamic or vibrating applications.

Design Insight: Increasing the internal corner radius is the most effective and lowest-cost method to reduce Ktand enhance part durability. Typically, increasing the radius from 0.1mm to 1mm can reduce stress concentration by about 70%.

1. Manufacturability Challenges and Cost Trade-offs

From a manufacturing perspective, the internal corner radius is directly linked to machining strategy, tool selection, and cost.

Internal Radius Requirement	Manufacturing Strategy & Cost Impact	Typical Tool Diameter (Example)
R < 0.5 mm	Requires small diameter tools, leading to conservative cutting parameters, multiplied machining time, rapid tool wear, and easy breakage. May require EDM for corner cleaning, adding steps and cost.	≤ Ø1 mm End Mill
R 0.5 - 1.0 mm	Can use smaller tools from the standard library. Machining efficiency and tool life are in an economical range. Recommended for most precision parts.	Ø1 - Ø2 mm End Mill
R 1.0 - 3.0 mm	Allows for more robust tools, enabling high-speed and efficient cutting. This is the most cost-effective range.	≥ Ø2 mm End Mill
Sharp Corner (R=0)	Theoretically unachievable by milling. In practice, the toolpath leaves a remnant radius equal to the tool radius. If specified on the drawing, it leads to acceptance disputes.	Not Applicable

LEDA Insight: Specifying an excessively small radius means you are paying for an unnecessary, high-difficulty, high-cost manufacturing challenge. Our DFM analysis will automatically identify such features and suggest optimizations.

2. Detailed Design Specifications and Recommended Values

Based on material properties, load type, and process capability, we recommend the following design specifications:

Application Scenario / Material Type	Recommended Min. Radius	Key Considerations & Explanation
General Structural Parts (Static Load)	≥ 0.5 mm	A safe and economical starting point, suitable for most non-load-bearing or low-stress internal corners.
High-Cycle Fatigue Areas	≥ 1.0 mm or ≥ 0.2t (t=adjacent section thickness)	A larger radius is necessary to significantly smooth stress flow, key to improving fatigue life.
Die Cast/Injection Molded Parts	≥ 0.3 mm	Balances demolding and mold life. Too small a radius weakens the mold and becomes a starting point for heat cracks.
Specific Material Guidelines		
Aluminum Alloy, Engineering Plastics	≥ 0.5 mm	Good machinability, but still avoid extremely small tools to ensure efficiency and quality.
Steel, Stainless Steel	≥ 1.0 mm	Higher strength materials are more sensitive to stress concentration;

Application Scenario / Material Type	Recommended Min. Radius	Key Considerations & Explanation
		higher cutting forces require robust tools.
Titanium Alloy, High-Temp Alloys	≥ 1.5 mm	Extremely sensitive to stress concentration and work hardening; ample design margin must be provided.

Core Design Rules:

Unify and Maximize: Where function allows, unify non-critical internal corner radii and maximize them. For example, unifying multiple R0.5 corners to R1 can reduce tool changes and improve programming and machining efficiency.

"Undercut" Design: If assembly truly requires a square corner, use an "undercut" design. The main feature is machined with a larger radius (e.g., R1), and a small, deeper relief groove is designed at the corner for cleaning with a specialized small tool. This is superior to using a very small radius everywhere.

Clear Drawing Annotation: Never specify "sharp corner." If no special requirement exists, annotate "Unspecified internal fillet R0.5 MAX" to provide a clear, reasonable basis for manufacture and acceptance.

3. Collaboration with LEDA: Complete Assurance from Design to Verification

At LEDA, we help you optimize internal corner design and balance performance with manufacturing cost through systematic engineering methods:

Automated DFM Alerts: After uploading the 3D model, our system automatically scans and flags all internal corners smaller than the recommended value, providing a risk assessment (High/Medium/Low) and a specific cost/cycle time impact analysis.

Engineering Simulation Support: For critical load-bearing or dynamic components, our engineering team can provide FEA services to visually demonstrate stress distribution and fatigue safety factors under different fillet designs, supporting your decisions with data.

Collaborative Process Optimization: When a small radius is unavoidable, we review it with you to provide the most economical process combination (e.g., "Use a Ø3mm tool for roughing/finishing most areas, and only use a Ø1mm tool for local corner cleaning"), clearly stating the impact on cost and delivery time.

Conclusion: A well-considered internal corner radius is a prime example of the fusion of "design for performance" and "design for manufacture." It achieves higher product reliability, lower manufacturing cost, and more stable quality output with minimal design changes. At LEDA, we help you integrate this philosophy into every design detail.

3.2.3 Slots, Deep Cavities, and Tool Interference

Slots, deep cavities, and other enclosed or semi-enclosed pocket features are key to achieving part lightweighting, housing electronic components, arranging flow channels, or enabling specific motion functions. However, they pose significant manufacturability challenges, centered on tool accessibility, rigidity, and the control of vibration and heat during machining. Unreasonable slot/cavity design can lead to unmanufacturable features, extremely low efficiency, soaring costs, or scrapped parts due to deformation. This section systematically provides a "safe guide" for designing such features and explains how LEDA tackles these challenges with advanced processes.

1. Core Challenges and Design Principles

Machining slots and deep cavities involves removing material by plunging a slender, rotating tool deep into the workpiece. This creates three core conflicts:

Accessibility and Interference: The tool and its holder must be able to enter and reach the machining area without collision.

Rigidity and Deformation: Tools with excessively high Length-to-Diameter ratios act like a "guitar string," bending under cutting forces and vibrating, leading to dimensional errors and poor surface finish.

Chip Evacuation and Heat Dissipation: In confined spaces, chips are hard to remove, and coolant cannot reach the cutting edge, causing heat buildup, rapid tool wear, and part thermal deformation.

Core Design Principle: "Design for the Tool." When designing slots and cavities, visualize how a virtual tool would perform the cutting. The goal is to maximize the open space of the cavity and minimize the tool overhang.

2. Detailed Design Specifications and Recommended Values

Feature Type	Key Geometric Parameters	Suggested Safe Design Value	High Risk / Unmanufacturable Design	Reasons, Risks & Solutions
Open-Ended Slot	Slot Width	≥ Standard Tool Diameter (e.g., Ø3, Ø4, Ø6mm)	Slot width is an uncommon non-standard value (e.g., 1.7mm)	Non-standard width requires custom tools, high cost, long lead time. Solution: Design slot width to a standard tool diameter, or allow a slightly larger width and machine with a standard tool in two passes.
	Slot Depth	≤ 4 × Slot Width (using conventional tools)	> 6 × Slot Width	Excessive tool L:D ratio, sharp drop in rigidity, prone to tool deflection and chatter. Solution: Use extended carbide tool holders, or machine from both sides of the part.
Closed Pocket	Minimum Wall Radius	≥ Tool Radius (typically ≥ R0.5mm)	Sharp corner or very small radius	The tool cannot clean the corner, leaving remnant material. Solution: Define the corner as an "undercut" feature, or allow a radius matching the tool radius.
	Pocket Depth	≤ 3 × Smallest Tool Diameter	> 5 × Smallest Tool Diameter	Difficult deep pocket machining, very poor chip evacuation and heat dissipation. Solution: Use high-efficiency, low radial force roughing strategies like plunge milling or trochoidal milling; use specialized long-flute tools for finishing.

Feature Type	Key Geometric Parameters	Suggested Safe Design Value	High Risk / Unmanufacturable Design	Reasons, Risks & Solutions
Deep Cavity/Hole	Opening Size	As large as possible, allowing large tools to enter	Small opening, large internal cavity ("T-shaped" or "bottle-shaped" internal cavity)	Large diameter tools cannot enter, forcing the use of tools with extremely high L:D ratios for layer-by-layer milling, resulting in very low efficiency and high risk. Solution: Avoid such designs. If essential, deep collaboration with process engineers is needed to assess feasibility.
Undercut (Corner Cleanup)	Undercut Width & Depth	Width ≥ 1.5mm, Depth ≤ 3 × Width	Narrow and deep undercut groove	Special undercut tools have very poor rigidity. Solution: Where function allows, use a larger overall fillet; or specify an acceptable process radius in the corner.

Key Concept: Tool Length-to-Diameter (L:D) Ratio

Definition: The ratio of the tool's extended length to its diameter. This is the golden indicator for measuring tool rigidity.

Safe Range: Typically, L:D ≤ 4:1 indicates good tool rigidity; 4:1 < L:D ≤ 6:1 is acceptable but challenging; L:D > 6:1 enters a high-risk zone requiring special processes.

Design Application: When designing slot depth, perform a quick self-check using: "Slot Depth ≤ 4 × [Planned Standard Tool Diameter]".

3. Predicting and Avoiding Tool Interference

Tool interference often occurs in multi-axis machining of complex parts. Designers can predict it using the following principles:

Global Interference: Check if the tool holder and collet will collide with the part or fixture. Pay special attention when machining the bottom of deep cavities or inclined surfaces.

Local Interference: During side milling, ensure the tool's side flutes have enough space for axial movement to avoid collision with adjacent bosses or ribs.

4. -Axis Accessibility: For extremely complex surfaces or internal cavities, determine if 3-axis machining is feasible. If not, 5-axis machining is required, necessitating consideration of potential collision between the 5-axis head and the workpiece.

LEDA Tool: We strongly recommend using simple motion simulation in 3D CAD software after the initial design or consulting our engineers. We can provide 3D toolpath simulation videos to visually demonstrate the machining process, identifying and resolving potential interference issues in advance.

5. Collaboration with LEDA: Engineering Solutions Beyond Conventional Limits

When your design must push conventional limits due to functional requirements, LEDA's engineering team is your strong backup. We provide solutions through:

Customized Tooling Solutions: For special slot widths or deep cavities, we can design and procure non-standard custom tools, such as anti-vibration tool holders, long-neck ball nose cutters, and step mills.

Multi-Axis & Special Processes: For challenges like "T-shaped" cavities, we employ advanced strategies like 5-axis simultaneous side milling, trochoidal milling, or plunge milling for efficient and controlled machining.

Process Sequence Optimization: By cleverly planning the machining sequence (e.g., adding "process windows," manufacturing in parts and joining), we can transform unmanufacturable features into manufacturable ones.

Simulation and Verification: Before production, we use professional CAM software for full virtual manufacturing simulation, ensuring every step from roughing to finishing is safe, efficient, and interference-free.

Summary: Designing slots and deep cavities is a dialogue with the laws of physics. Following the specifications above will enable you to design efficient, economical, and reliably manufacturable features. When design requirements touch the boundaries of manufacturability, start a conversation with the LEDA engineering team early. Our value lies not only in "can it be machined" but in "how to realize your design intent optimally," working together to turn challenges into high-value product advantages.

3.3 Practical Advice on Tolerances and Surface Roughness

Tolerances and surface roughness are the language of engineering drawings, precisely conveying your expectations for the part's function. However, inappropriate tolerance and surface requirements are among the primary factors leading to non-linear increases in manufacturing costs. This section aims to provide a set of pragmatic, economical, and actionable strategies for defining tolerances and surface roughness, helping you find the optimal balance between meeting functional needs and optimizing manufacturing costs, and establishing efficient collaboration standards with LEDA .

1. The Philosophy of Tolerances: From "Function-Driven" to "Cost-Conscious"

Tolerance setting should strictly follow the **"function-driven"** principle, not "as good as possible." An unnecessarily tight tolerance (e.g., ± 0.01 mm) can cost several times more than a reasonable tolerance (e.g., ±0.05 mm) .

Tolerance Definition Process:

Identify Functional Interfaces: Identify which surfaces on the part contact, fit, move against, or seal with other parts. These are critical features requiring defined tolerances.

Define Fit Requirements: Determine if it's a clearance, transition, or interference fit. Consult mechanical design handbooks to determine the tolerance zone code (e.g., H7/g6) based on the fit type and basic size. This is more professional and universal than directly specifying numerical values.

Assign Non-Functional Tolerances: For free dimensions that do not affect assembly or function, do not specify individual tolerances. Instead, reference a general tolerance standard (e.g., ISO 2768-m). This grants reasonable flexibility in manufacturing.

LEDA Standard Economic Accuracy Reference:

The table below shows the economic machining accuracy stably achievable under LEDA's standard process conditions without special measures, serving as a benchmark for your preliminary design .

Process Method	Typical Economic Tolerance (mm)	Achievable Precision Tolerance (mm)	Application Scenario & Cost Tip
Conventional CNC Milling	±0.05	±0.025 ~ ±0.01	Most mechanical structural parts. ±0.01 requires precision machines, constant temperature environment, and strict process control.
Turning	±0.02 (Diameter)	±0.01 ~ ±0.005	Rotating bodies like shafts and holes. High precision requires precision lathes.
Grinding	±0.005	±0.002 ~ ±0.001	Fitting surfaces with extremely high requirements for hardness, roundness, flatness. Cost significantly higher than milling/turning.
Wire EDM	±0.005	±0.002	Precision contours for hard materials, mold inserts. High accuracy, but slower speed.
Drilling (Location)	±0.05 (Position)	±0.02 (Requires jig/CNC)	Hole position accuracy needs separate consideration. Depth tolerance can usually be relaxed.

Key Recommendations:

Avoid Chain Dimensioning: Use a datum reference frame. Use one or several key features as datums (A, B, C), and dimension the position and profile of other features relative to these datums. This aligns with inspection logic, better reflects assembly function, and is often looser and more economical than chained ± tolerances.

Consider Measurement Datums: Any tolerance you specify must be measurable in reality. If it cannot be easily measured with standard gauges (calipers, micrometers) or a CMM, the tolerance may be impractical or extremely costly .

2. The Significance of Surface Roughness: Beyond "Looking Smooth"

Surface roughness not only affects appearance but also directly influences friction coefficient, wear resistance, fatigue strength, sealing, and coating adhesion .

Commonly Achievable Roughness Ranges for Machining Methods (Ra, µm):

Machining Method	Typical Roughness Ra (µm)	Surface Characteristics & Explanation
Standard Milling/Turning	3.2 ~ 1.6	Visible tool marks. Suitable for non-fitting surfaces, internal structural surfaces.
Fine Milling/Fine Turning	1.6 ~ 0.8	Slight tool marks visible. Most common requirement for functional surfaces, e.g., bearing seats, seal grooves.
Precision Grinding	0.8 ~ 0.2	Machining traces barely discernible. For high-load bearing surfaces, hydraulic rods, precision fitting surfaces.

Machining Method	Typical Roughness Ra (μm)	Surface Characteristics & Explanation
Lapping/Polishing	0.2 ~ 0.025	Glossy surface, mirror effect. For optical surfaces, high-seal surfaces, appearance decoration.
EDM	Varies with parameters, typically > 0.8	Surface has micro-pits (spark erosion pattern), beneficial for oil retention. Can be used directly or polished.

Practical Advice:

Match Functional Needs: Sealing surfaces typically require Ra 0.8 or better; sliding bearing surfaces may need Ra 0.4-0.2; static mounting surfaces often suffice with Ra 1.6-3.2 .

Specify Maximum Value: Usually specify the upper limit of roughness (e.g., Ra 0.8), meaning "cannot be worse than this value." A finer finish at lower cost is acceptable.

Define Lay Direction: If the surface has a lay requirement (e.g., for reciprocating seals), specify the lay direction symbol.

3. Collaborating with LEDA: "Translating" and Optimizing from Drawing to Perfect Part

When you provide initial drawings, LEDA's engineering team will collaborate with you to optimize tolerance and surface requirements through the following process:

Tolerance Analysis: For critical assembly dimensional chains, we can perform tolerance stack-up analysis to determine if the assembly's worst-case scenario still meets functional requirements under your defined tolerances. This identifies overly tight or loose tolerances and suggests optimizations.

Manufacturability Review: Our engineers review each tolerance and roughness requirement, assess the required processes, equipment, and inspection methods, and provide cost comparisons for alternative solutions. For example: "Relaxing the tolerance here from ±0.01 to ±0.02 can save 20% machining cost without affecting assembly."

Inspection Plan Collaboration: We will clearly inform you how each critical tolerance will be inspected (e.g., using CMM, special go/no-go gauges, or optical comparators), ensuring our measurement methods align with your design intent and preventing future disputes.

Golden Rule: Before the design is frozen, conduct a formal drawing review with your LEDA project manager or application engineer. This is the most effective step to avoid the biggest risks (cost overruns, delays, functional non-compliance) at the smallest cost (communication time).

Summary: Intelligent design of tolerances and surface roughness is the hallmark of an excellent engineer. It signifies a deep understanding of the intrinsic links between design, manufacturing, and cost. By applying the above practical advice and collaborating closely with LEDA, you will be able to deliver designs that are not only functionally superior but also economical to manufacture and stable in quality, thereby gaining a decisive competitive advantage in the market.

3.4 Considerations for Datum and Fixture Setup

In precision manufacturing, a **datum** is the "absolute coordinate system" for a part in both the virtual design space and the physical manufacturing world. A **fixture** is the "anchor" that secures the part stably and repeatably within this coordinate system. Unreasonable datum selection or fixture design is a root cause of accumulated machining errors, poor consistency in batch parts, or even total scrap. This section systematically explains how to scientifically set datums and how to create conditions for efficient and reliable fixturing during the design phase, thereby ensuring manufacturing accuracy at the source.

1. The Philosophy of Datums: From "Drawing Origin" to "Foundation of Manufacturing and Inspection"

A datum is not just a coordinate system in a CAD model. It is a real, physical geometric feature that can be probed by machine tool probes, CMMs, or inspection gauges, used to establish the unique reference for the part's orientation, location, and measurement during machining and inspection .

Core Principles of Datum Setup:

Functional Relevance: Prioritize functional features that serve as locators in the final assembly. Examples include a face that mates with a housing or a hole that fits with a shaft.

Stability and Repeatability: Datum features must have sufficient size, rigidity, and stability to be accurately and repeatably located across multiple setups or measurements.

Manufacturability: The datum feature itself must be machinable with precision early in the process. Typically, the first machined feature becomes the datum for subsequent operations.

Datum Hierarchy and Establishment Sequence:

A sound datum system typically follows the **"3-2-1" Locating Principle**, establishing primary, secondary, and tertiary datums.

Primary Datum (e.g., A): Usually a plane that restricts three degrees of freedom (two rotations, one translation). Select the largest, most stable, and functionally critical face on the part.

Secondary Datum (e.g., B): Usually a plane or line perpendicular to the primary datum, restricting two degrees of freedom (one rotation, one translation).

Tertiary Datum (e.g., C): A point, line, or surface perpendicular to both A and B, restricting the final translational degree of freedom.

Clear Expression on Drawings:

Always clearly identify datum features using Geometric Dimensioning and Tolerancing (GD&T) datum symbols on the 2D drawing, and establish the datum sequence within feature control frames. This is the only way to communicate unambiguously with manufacturing and inspection departments .

2. Design for Manufacture: Considerations for Fixture Interfaces

An excellent designer anticipates how a part will be clamped. Considering fixturing during the design phase avoids the subsequent need for expensive custom fixtures or inefficient, unstable clamping methods.

Design Consideration	Recommended Design	Design to Avoid	Rationale & Solution
Clamping Area	Reserve dedicated, flat clamping surfaces or process tabs away from critical features.	Entire part consists of complex curves with no clamping area.	Dedicated tabs can be removed in the final operation. If space is limited, design threaded holes for modular fixture connections.
Clamping Force & Deformation	Ensure sufficient part rigidity under clamping force, or design reinforcing ribs. Avoid direct pressure near thin-walled areas.	Designing vise jaws directly on very thin, flexible walls.	Clamping force causes elastic deformation; after unclamping, the part springs back, causing dimensional errors. Consider vacuum chucking or low-melting-point alloy support.
Multi-Side Machining & Re-Location	Design a unified datum system allowing many features to be machined in a single setup. If multiple setups are needed, ensure convertible common datums.	Defining different, unrelated datums for each machining side.	Each re-location introduces new error. A unified datum is key to ensuring positional accuracy between features on different sides.
Interference	Allow sufficient clearance for fixtures, clamps, screws, and tools. Consider potential collisions between toolpaths and fixtures.	Designing critical features in corners where fixtures or tools cannot access.	Perform simple interference checks in the 3D model. Discuss clamping strategy with process engineers early on.

Common Clamping Methods and Design Adaptation:

Machine Vise: Requires two parallel clamping faces. Can reserve process tabs during design.

Vacuum Chuck: Requires a large, flat, airtight bottom surface. The part base can be designed as a continuous plane or with sealing grooves.

Modular Fixtures / Zero-Point Clamping Systems: Design standardized fixture interface holes (e.g., grid of threaded holes) for rapid, high-repeatability positioning. Ideal for flexible manufacturing.

3. The "Static" and "Dynamic" of Fixture Design: Differences Between Roughing and Finishing

Roughing Fixtures: Core requirements are high strength, rigidity, and clamping force to withstand high cutting loads and vibration. Larger contact areas and forces are acceptable, focusing on efficiency and stability.

Finishing Fixtures: Core requirements are high precision, minimized clamping deformation, and excellent repeatability. Often use three-point locating, elastic fixtures, or precision chucks to apply minimal, controlled force, avoiding stress-induced distortion.

LEDA Insight: During design reviews, we evaluate the most suitable clamping strategy for your part. For complex parts, we may recommend combined fixtures or custom flexible vacuum fixtures to maximize machining efficiency while ensuring accuracy.

4. Collaboration with LEDA: Building a Datum Closed Loop from Design to Inspection

At LEDA, we view datums as the lifeline of quality. Our collaborative process ensures your design intent is accurately transmitted to the final part:

Design Review: Our process engineers review your datum setup to confirm its manufacturability, inspectability, and stability from a manufacturing perspective, suggesting optimizations.

Fixture Scheme Pre-Communication: During the quotation phase, we discuss preliminary clamping and locating strategies, confirm potential interferences or risks, and incorporate requirements for process tabs/interfaces into the design considerations.

Virtual Manufacturing Simulation: During programming, we use CAM software for full fixture, toolpath, and machine motion simulation to identify and resolve potential collisions and interferences in advance.

First Article Inspection Report: The first article part is inspected using a CMM, strictly following the datum system defined in your drawing. A detailed inspection report is provided, visually showing the part's actual deviation from the design datums.

Summary: Well-considered datum setup and fixturing considerations are the bridge connecting excellent design to perfect manufacturing. It transcends single-part design, rising to the level of manufacturing system design. Through early and in-depth collaboration with LEDA, you define not just a part, but an efficient, reliable, and repeatable manufacturing process, ensuring comprehensive quality, cost, and delivery time benefits.

3.5 Top 10 Design Techniques for Cost Reduction

Optimizing cost while ensuring performance and reliability is the ultimate hallmark of excellent design. Over 80% of cost-saving opportunities are embedded within the product design phase. The following ten design techniques, distilled from LEDA's experience across thousands of successful projects, aim to help you instill "cost awareness" at the design source. Through clever design decisions, you can achieve significant manufacturing economies without compromising quality .

Technique 1: Promote Standardization and Modularization

Core: Maximize the use of standardized hole sizes, thread types, tool radii, material grades, and component specifications.

Cost-Reduction Logic: Reduces the procurement of non-standard tools/gauges, lowers programming complexity, and improves the versatility of machines and production lines. Modular design allows for a diverse range of products to be created from a small set of universal parts, significantly cutting costs for molds, tooling, and inventory .

LEDA Collaboration: We can provide our company's preferred standard parts library and materials list, and assist in developing a modular architecture for your product families.

Technique 2: Make Informed Material and Condition Choices

Core: Prioritize materials with good machinability, ample supply, and lower cost, provided functional requirements are met.

Cost-Reduction Logic: Avoids "over-engineering." For example, use 6061 aluminum instead of 7075 for general structures; use easy-machining stainless steel 303 instead of 304 in non-demanding corrosive environments; procure pre-hardened steel (e.g., P20) to avoid subsequent heat treatment .

LEDA Collaboration: We provide comprehensive cost analyses for different material options, covering raw material, machining time, tool wear, and post-processing costs.

Technique 3: Optimize Feature Geometry to Reduce Machining Difficulty

Core: Adhere to the "design for the tool" principle. Standardize internal fillet radii, relax non-critical aspect ratios, use milling for non-round holes instead of drilling, and avoid deep, narrow slots.

Cost-Reduction Logic: Enables the use of larger, more rigid tools for high-speed cutting; reduces the need for special tools and EDM processes; improves machining stability and yield .

LEDA Collaboration: Our online DFM tool automatically flags high-cost-risk features and provides specific geometric optimization suggestions.

Technique 4: Design Geometry that is Easy to Fixture

Core: Reserve large, flat clamping surfaces; design unified process datums; consider the potential for vacuum or modular fixturing.

Cost-Reduction Logic: Reduces the design and manufacturing time for complex, expensive dedicated fixtures; shortens machine setup time; improves fixturing repeatability and reduces scrap .

LEDA Collaboration: During the design review phase, we provide fixture strategy previews and suggest adding or optimizing process locating features.

Technique 5: Relax Tolerances and Surface Finishes to "Good Enough"

Core: Conduct a functional review of every tolerance and surface roughness requirement. Relax tolerances for non-critical features to economic machining accuracy and reference general tolerance standards.

Cost-Reduction Logic: Avoids costly secondary processes like precision grinding or hand finishing to achieve unnecessary "aerospace-grade" accuracy. A tolerance of ±0.05mm can cost over 60% less to manufacture than ±0.01mm .

LEDA Collaboration: We can perform tolerance stack-up analysis to verify assembly functionality with relaxed tolerances and provide clear data on cost impact.

Technique 6: Reduce Part Count and Assembly Operations

Core: Utilize integrated design (e.g., 3D printing complex structures, CNC machining monolithic components) to replace the assembly of multiple parts.

Cost-Reduction Logic: Reduces part management, assembly labor, and connector (screws, clips) costs, while enhancing the product's overall rigidity and reliability .

LEDA Collaboration: We excel at evaluating the technical and economic balance between "monolithic manufacturing" and "multi-part assembly," especially in multi-axis machining and metal 3D printing.

Technique 7: Design for Efficient Chip Evacuation and Cooling

Core: For cavity parts, design appropriate draft angles and smooth internal transitions, avoiding enclosed dead ends.

Cost-Reduction Logic: Improves chip evacuation and coolant flow during cutting, preventing tool wear caused by chip entanglement or overheating, thereby allowing higher cutting parameters, longer tool life, and better surface quality .

LEDA Insight: A slight draft angle (e.g., 1°) typically does not affect function but can significantly improve the feasibility and economy of deep cavity machining.

Technique 8: Leverage Standard Raw Material Forms

Core: Design considering the standard sizes of plates, bars, and tubes, so the blank size is closest to the finished part, minimizing material waste and rough machining time.

Cost-Reduction Logic: Minimizes material procurement cost and machining allowance. For example, design part thickness as a multiple of the standard plate thickness .

LEDA Collaboration: Based on your initial design, we can provide optimized dimension suggestions based on locally available standard stock materials.

Technique 9: Simplify Inspection and Measurement

Core: Design features that are easy to inspect. Define clear, rational datums. Avoid using difficult-to-measure virtual or complex surfaces as key tolerance references.

Cost-Reduction Logic: Reduces the complexity and cost of quality control. A design that is easy to inspect means faster measurement times, lower gauge costs, and fewer quality disputes .

LEDA Collaboration: We can review drawings from an inspection perspective during the design phase, ensuring all critical characteristics can be measured efficiently and accurately.

Technique 10: Collaborate Early and Continuously with the Manufacturer

Core: This is the most important technique. Involve the manufacturing partner's (LEDA's) engineering experience from the conceptual design stage.

Cost-Reduction Logic: Front-loads manufacturing constraints and optimization opportunities, avoiding costly design changes later. Collaborative design uncovers optimal solutions not visible to either party alone .

LEDA Commitment: Through our online platform, design review meetings, and engineering support, we are not just your supplier, but your strategic partner in reducing total cost and accelerating time-to-market.

Summary: Cost is not cut after the fact; it is designed in from the start. By systematically applying these techniques and collaborating deeply with a professional manufacturing partner like LEDA, you will be able to create products that are highly competitive in performance, quality, and cost. Let's start with your next design and build the cost advantage into the blueprint together.

3.6 Industry Focus: Aerospace Structural Components vs. Medical Device Implants

At the pinnacle of precision manufacturing, the aerospace and medical device industries represent the ultimate demands on materials science, engineering design, and manufacturing processes. Although both pursue impeccable reliability and performance, their underlying logic, evaluation criteria, and technical paths are distinctly different. Understanding these differences is key to successfully designing products for these specific sectors. LEDA Precision possesses deep project experience and certifications in both fields. This section reveals the core design philosophies and engineering practices for each .

1. Comparison of Core Design Philosophies

Dimension	Aerospace Structural Components	Medical Device Implants
Primary Driver	Performance and Weight. Pursuit of extreme lightweighting while meeting extreme mechanical and thermal performance. Every gram reduced translates to significant operational cost savings.	Safety and Biocompatibility. Materials and design must coexist long-term and harmoniously with the human body, causing no rejection, toxicity, or adverse reactions. Function serves healing and quality of life.
Failure Mode	Fatigue, creep, fracture under complex loads (aerodynamic, inertial, thermal). Failure consequences are catastrophic.	Bio-corrosion, wear debris, aseptic loosening, infection. Failure directly harms patient health and life.
Verification Core	Physics-based modeling and testing. Relies on complex CAE simulation and vast amounts of ground and flight test data to validate performance and lifespan under extreme conditions.	Biology-based verification and clinical trials. Requires rigorous in vitro biocompatibility testing, animal studies, and clinical trials to demonstrate long-term safety and efficacy.

2. Fundamental Differences in Material Selection

Aspect	Aerospace Structural Components	Medical Device Implants

Aspect	Aerospace Structural Components	Medical Device Implants
Core Considerations	Specific strength (strength-to-weight ratio), specific stiffness, high-temperature resistance, fatigue resistance.	Biocompatibility, resistance to bodily fluid corrosion, wear resistance, elastic modulus matching bone tissue.
Typical Materials	• Composites: Carbon Fiber Reinforced Polymers (CFRP). • Light Alloys: Ti-6Al-4V, 7075 Aluminum, Magnesium alloys. • Superalloys: Inconel 718.	• Pure Titanium & Alloys: CP-Ti, Ti-6Al-4V ELI (Extra Low Interstitial). • Cobalt-Chromium Alloys: CoCrMo (high wear resistance, for joints). • Medical Stainless Steel: 316L (low carbon grade). • PEEK and its composites.
Material Condition	Stringent requirements for batch consistency, metallurgical defects, residual stress. Materials require full melt reports and certificates of conformity.	Must use medically certified materials complying with ASTM/ISO medical standards, providing traceable biocompatibility certificates. Strict upper limits on impurity elements (e.g., Ni, V).

3. Industry-Specific Characteristics in Tolerances, Surfaces, and Structural Design

Aspect	Aerospace Structural Components	Medical Device Implants
Tolerance Strategy	Balances functionality and weight reduction. Uses topology optimization to generate the lightest load paths. Tolerances ensure the assembly of optimized complex surfaces. Widespread use of GD&T to control stack-up in large assemblies.	Biological response is priority. Bearing surfaces require extremely high form and fit accuracy to minimize wear. Porosity and pore size tolerances on bone-contact surfaces (e.g., porous structures) are critical for bone ingrowth.
Surface Treatment	• Functional: Micro-arc oxidation, shot peening to improve fatigue strength; low-friction coatings. • Protective: Anodizing, anti-corrosion coating systems.	• Bio-functional: Sandblasting, acid etching to create micro-roughness promoting osseointegration; Hydroxyapatite coatings to promote osteoconduction. • High Finish: Bearing surfaces require mirror polishing to Ra < 0.05 μm to minimize wear.
Structural Features	Extensive use of monolithic structures, lattice designs, composite layups, and conformal cooling channels to maximize performance/weight ratio.	Designed trabecular structures, porous surfaces to promote tissue ingrowth; avoid dead ends for easy cleaning and sterilization; consider intraoperative visibility (e.g., X-ray markers).

4. The Iron Rule of Certification, Documentation, and Traceability

Aspect	Aerospace Structural Components	Medical Device Implants
Quality System	AS9100 is the international aerospace quality system standard. Emphasizes risk management, supply chain control, and First Article Inspection (FAI).	ISO 13485 is the core quality management system for medical devices. Emphasizes design controls, risk management, and traceability.
Production Control	Strict First Article Inspection. Each part has a complete manufacturing record, including raw material batch number, parameters for each operation, operator, and inspection records.	Must establish a Device Master Record (DMR) and Device History Record (DHR). Achieves full unique traceability from raw material to finished product.

Aspect	Aerospace Structural Components	Medical Device Implants
Regulatory Oversight	Must comply with requirements from airworthiness authorities like the FAA (US) and EASA (EU). Involves certification of materials, processes, and design.	Requires registration and approval from bodies like the FDA (US), CE MDR (EU), and NMPA (China). Clinical data is core evidence.

5. Collaborating with LEDA: Your Industry-Specific Bridge

LEDA Precision deeply understands the unique language and rules of these two industries. We are not just a manufacturer but your guide to compliance and success.

For Aerospace: We provide material and process certification support, perform non-destructive testing compliant with aerospace standards, and offer tooling and assembly solutions for complex components. Our system meets AS9100 quality system and NADCAP special process certification requirements.

For Medical Devices: We produce implants in Class 10,000 cleanrooms, ensuring contamination control. We offer full support for biocompatibility testing, sterilization validation, and packaging services. Our quality management system is fully compliant with ISO 13485 and supports FDA 21 CFR Part 820 compliance.

Decision Guide: Which Category Does Your Project Belong To?

If your component needs to withstand extreme physical environments with the lightest weight and highest load, and its failure could lead to a systemic disaster, follow the **Aerospace** design and manufacturing logic.

If your component needs to exist long-term or permanently within the human body, interacting with biological tissues, and its failure directly endangers an individual's life and health, follow the **Medical Device** design and manufacturing logic.

Summary: Designing for aerospace or medical devices is a deep dialogue with the laws of physics or life sciences. Choosing the correct design philosophy, material system, and manufacturing partner is the cornerstone of project success. With engineering capabilities and quality systems spanning these two high-tech fields, LEDA Precision is your trusted partner for transforming innovative ideas into safe, reliable, and high-performance products.

Chapter 4: Injection Molding Design Guide

Injection molding stands as one of the most efficient and widely used forming technologies for manufacturing plastic products. It involves injecting molten plastic at high speed into a precision mold cavity, enabling the mass production of complex, dimensionally accurate plastic parts, directly transforming raw material into finished goods. This "additive" forming method is particularly suited for high-volume production, offering unparalleled advantages in efficiency and economy .

However, behind a successful injection-molded part lies the precise coordination of product design, mold engineering, and molding process. Design decisions directly determine the complexity of the mold structure, production stability, and the quality of the final product. Excellent injection molding design is not merely about "whether it can be formed" but about whether high-quality products can be produced efficiently, economically, and consistently .

This chapter will serve as your practical guide, providing an in-depth analysis of how to design plastic parts that are easy to manufacture. You will learn that by adhering to core design principles, you can:

Avoid Common Defects: For example, effective prevention of sink marks and warpage through uniform wall thickness design .

Simplify Mold Structure and Reduce Costs: For instance, reducing the need for complex sliders and lifters through appropriate draft angles and by avoiding internal undercuts .

Enhance Part Quality and Strength: For example, increasing structural rigidity while controlling wall thickness through the rational application of ribs and fillets .

Ensure Production Stability: Proper design of the gating and venting systems is crucial for ensuring smooth filling and reducing defects .

We will start by understanding basic concepts like the parting line and demolding direction, which are the bridge connecting product design and mold manufacturing. Subsequently, we will delve into the golden rules for designing features like wall thickness, ribs, boss pins, and snap-fits, which form the core of this chapter. We will also examine how material selection affects shrinkage and design details, and how mold feasibility (such as venting and cooling) interacts with product design .

Whether you are designing a consumer electronics housing requiring a premium appearance or a medical connector demanding high reliability, the principles in this chapter are universally applicable. Mastering them will enable you to communicate efficiently and precisely with mold engineers and manufacturers (such as LEDA), working together to transform your ideas into competitive plastic products .

4.1 Plastic Material Selection: Properties, Shrinkage, and Process Window

Material selection is the starting point of injection molded product design. It fundamentally determines the product's mechanical properties, heat resistance, appearance, service life, and ultimate cost. Correct selection is the cornerstone for achieving efficient and stable production .

4.1.1 Overview of Thermoplastics and Thermosets

Understanding the fundamental differences between these two major categories of plastics is the first step in selection .

Thermoplastics: These plastics have linear or branched molecular chains without chemical cross-links. Their characteristic is that they soften and melt when heated and solidify upon cooling, and this process is reversible, theoretically allowing them to be recycled and reused repeatedly. Most common injection molding materials fall into this category, such as PP, ABS, PC, PA (Nylon), and POM. They are convenient to process and mold, possess good mechanical properties, and have the widest range of applications .

Thermosets: These plastics can also soften and flow upon initial heating but undergo an irreversible chemical reaction (cross-linking or curing) under specific temperature and pressure, forming a three-dimensional network molecular structure. Once cured, the product becomes rigid and will not soften again upon reheating, only decomposing. Common examples include Epoxy (EP) and Phenolic (PF) resins. They typically offer higher heat resistance, dimensional stability, and structural strength but are difficult to recycle .

Core Selection Logic: If your product requires good toughness, recyclability, and a relatively simple processing cycle, thermoplastics should be the primary choice. If your product needs to maintain excellent stability and structural integrity under extreme temperatures, high frequency/pressure, or harsh chemical environments (e.g., electrical housings, brake pads, composites), thermosets should be considered .

4.1.2 Impact of Key Material Properties on Design

Flowability: Often indicated by Melt Flow Rate (MFR) or Melt Flow Index (MFI). It describes the ability of molten plastic to fill the mold cavity under pressure .

Design Impact: Materials with poor flowability (e.g., PC) require higher injection pressure and temperature to fill thin walls or complex structures, prone to short shots and weld lines. For such materials, designs should avoid excessively thin walls, and consider larger gates and runners. Conversely, materials with good flowability (e.g., PP) are more suitable for complex geometries .

Shrinkage: Refers to the percentage of volume contraction as the plastic cools and solidifies from the molten state. It is a core parameter that must be precisely compensated for in mold design .

Design Impact: The mold cavity size must be "part dimension + shrinkage allowance." Different materials have different shrinkage rates (e.g., POM ~1.8-2.5%, ABS ~0.5-0.7%). Non-uniform wall thickness causes differential shrinkage, a primary cause of warpage. Therefore, designing for uniform wall thickness is the golden rule to control the negative effects of shrinkage .

Mechanical Properties: Include tensile strength, flexural modulus, impact strength, and hardness .

Design Impact: Directly determine the structural strength and durability of the product. For example, parts enduring frequent snap-fit actions should choose materials with high impact strength and fatigue resistance like POM or certain modified PA; for structural supports, materials with high stiffness are needed, such as PC/ABS blends or reinforced PP .

Other Key Properties:

Heat Resistance: Measured by Heat Deflection Temperature (HDT). For components operating in high-temperature environments (e.g., automotive engine bay parts, lamp housings), materials with an HDT above the operating temperature must be selected, such as PC, PA, PPS .

Chemical Resistance: If the product contacts oils, solvents, or chemicals, the material's resistance to chemical corrosion must be considered. PP and PE, for instance, offer good resistance to many chemicals .

4.1.3 Selection Guide for Common Engineering Plastics

The table below summarizes several common engineering plastics used in LEDA Precision's applications and their key selection points, providing a quick preliminary reference .

Material Name	Key Characteristics	Typical Shrinkage (%)	Primary Applications	Design Considerations
ABS	Good overall mechanical properties, ease of plating/painting, cost-effective.	0.4 - 0.7	Electronic appliance housings, consumer goods, toys, automotive interiors.	Poor weatherability, prone to degradation outdoors; hygroscopic, requires thorough drying before processing.
Polycarbonate (PC)	High impact strength, high transparency, good heat resistance (HDT ~130°C).	0.6 - 0.8	Mobile phone housings, riot shields, safety glasses, lenses, automotive light covers.	Prone to internal stress, notch sensitive; relatively poor flowability, requires higher injection pressure.
Polyoxymethylene (POM)	High stiffness, low friction coefficient, excellent wear/fatigue resistance ("Acetal").	1.8 - 2.5	Gears, bearings, pulleys, snaps, springs, precision mechanical parts.	High shrinkage requires precise mold compensation; prone to thermal degradation at high temperatures, requiring strict control.
Polyamide (PA, Nylon)	High strength, wear resistance, oil resistance, heat resistance. Hygroscopic.	0.8 - 1.5 (varies with grade/humidity)	Power tool housings, automotive fans, gears, zip ties.	Significant dimensional/performance change with moisture absorption; must be conditioned before use/processing; good flowability.
Polypropylene (PP)	Low density, chemical resistance, good fatigue strength, low cost.	1.6 - 2.2	Food containers, automotive bumpers, daily-use items, living hinges.	Brittle at low temperatures; anisotropic shrinkage can lead to warpage; often requires glass fiber for enhanced rigidity.
Polybutylene Terephthalate (PBT)	Good mechanical properties, heat aging resistance, dimensional stability, low water absorption.	1.5 - 2.0	Electrical/electronic components (e.g., connectors, distributors).	Attacked by strong acids/bases; strength greatly enhanced with glass fiber.

Core Selection Principles Summary:

Match the Environment: Prioritize the product's end-use environment (temperature, chemicals, UV, etc.).

Meet Functional Requirements: Clearly define the mechanical stresses the product must endure (impact, wear, continuous load).

Balance Aesthetics: For parts with appearance requirements, choose materials suitable for painting, plating, or direct texturing.

Consider Cost & Processability: Where requirements are met, select materials with a wider processing window and lower cost. For example, PP is common for food containers due to its chemical stability and low density .

LEDA's Recommendation: Material selection is not an isolated step. We strongly advise communicating with LEDA's engineering team early in your project. We can provide professional material selection support based on your product concept, application scenario, and budget, and utilize Mold Flow Analysis (CAE) to predict potential molding issues, optimizing the design from the outset to ensure manufacturability .

4.2 Design Starting Point: Demolding Direction, Parting Line, and Basic Geometric Constraints

In injection molding design, every detail begins with a fundamental decision: the **demolding direction**. This direction acts like gravity, determining how the mold opens, how the part is ejected, and subsequently defining the parting line and a series of basic geometric constraints. Defining the demolding direction during the conceptual design stage is crucial to avoid costly design changes later .

4.2.1 Establishing the Demolding Direction and Its Global Impact

What is the Demolding Direction?

The demolding direction is the direction in which the moving half (B-side) and the fixed half (A-side) of the mold separate, which is also the direction in which the plastic part is ejected. Theoretically, any direction can be chosen, but there is only one optimal direction .

Core Principles for Establishing the Optimal Demolding Direction

Maximize Projected Area: Prioritize the direction where the part has the largest projected area. This contributes to better clamp force utilization and typically simplifies the mold structure.

Minimize Lateral Undercuts: The optimal demolding direction should result in the fewest and smallest undercut features on the part. Each undercut requires additional side-actions (like sliders or lifters), significantly increasing mold complexity, manufacturing cost, and maintenance risk .

Ensure Core Quality Characteristics: The demolding direction should guarantee that critical dimensions (e.g., holes or shafts with coaxiality requirements) are formed on the same side of the mold (cavity or core) to ensure accuracy. For appearance parts, the primary visible surfaces should be on the cavity side for better surface quality.

Global Impact of Demolding Direction

The choice of demolding direction is a global decision that affects the entire project like a domino effect:

Mold Structure and Cost: Determines whether the mold is a simple two-plate mold or requires complex sliders and lifters.

Parting Line Location and Appearance: Directly determines the path of the parting line on the part, affecting aesthetics and post-processing requirements.

Ejection System Design: Influences the placement of ejector pins and the ejection method, ensuring the part is demolded without deformation or damage.

Production Stability and Efficiency: A simpler mold structure implies shorter cycle times and higher production stability.

LEDA's Recommendation: At LEDA, we strongly advise discussing the demolding direction with our engineering team during the initial conceptual design phase of your product. We can utilize professional expertise and software tools to perform demolding direction analysis based on your 3D model, helping you lock in the optimal solution from the source and avoid risks .

4.2.2 Types of Parting Lines and Selection Strategies

The parting line is the line formed where the parting surface intersects the surface of the plastic part. It is an inevitable outcome of mold manufacturing, and its selection and design are critical .

Parting Line Type	Schematic	Characteristics & Application Scenarios	Design Considerations
Straight Parting Line	A straight line	Most common and simplest. The parting surface is flat, making the mold easy to machine and maintain. Suitable for parts with regular shapes and simple contours.	The edge at the parting line can be sharp. If exposed, features like lip seals or cosmetic grooves are often designed to conceal it.
Curved/Stepped Parting Line	Follows a curved edge or is stepped.	Used for parts with complex shapes, allowing the parting line to follow the product contour, avoiding noticeable lines on primary appearance surfaces.	Mold machining is more complex and costly. Attention must be paid to injection clamping force deflection caused by stepped parting surfaces; balanced mold layout is often needed.
Cosmetic Line	Designed as a specific decorative groove.	Turns passivity into initiative. Instead of trying to hide the parting line, it is incorporated into the styling as an obvious groove or trim strip, making it a design feature.	Requires close collaboration between the designer and mold engineer early on to actively integrate the parting line location into the aesthetic design.

Core Strategy for Parting Line Selection

Hide at Edges: Wherever possible, set the parting line on the edges or bend lines of the product, making it coincide with the visual boundary and less noticeable.

Avoid Primary Appearance Surfaces: Absolutely avoid having the parting line traverse the center of primary appearance surfaces.

Consider Secondary Processes: If the part requires secondary processing like painting, minor parting line marks can be covered. However, for high-gloss or transparent parts, the parting line must be handled with extreme precision.

4.2.3 Design Principles and Standard Values for Draft Angles

The draft angle is the essential taper applied to the internal and external walls of a plastic part along the demolding direction, crucial for ensuring smooth ejection and preventing part scuffing .

Why are Draft Angles Needed?

Without a draft angle, significant friction and vacuum adhesion occur between the part and the mold cavity walls, leading to ejection difficulties, surface scuffing (drag marks), or even whitening and breakage of the part during ejection .

Design Principles for Draft Angles

"Larger is Better" Principle: Where space and function allow, use as large a draft angle as possible. This facilitates demolding and improves production yield.

Material Difference Principle: Materials with high shrinkage, softer materials (e.g., PP, PE), or reinforced materials (e.g., glass-fiber reinforced plastics) require larger draft angles.

Surface Texture Principle: When the surface is to be textured (e.g., etched), the draft angle must be significantly increased. The deeper the texture, the larger the required angle, often following the empirical relationship `Draft Angle ≥ arctan(Texture Depth / Characteristic Value)` or directly referencing standards like VDI 3400.

Dimensional Accuracy Principle: For holes or pillars with precise dimensional requirements, the direction of the draft angle (adding or removing material) must be clearly specified to ensure critical dimensions are controlled within tolerance.

Standard Reference Values for Draft Angles

The table below provides empirical draft angle values for common scenarios (Unit: degrees / °) :

Feature Type	Smooth Surface	Fine Texture (e.g., ~VDI 3400 A1)	Medium Texture (e.g., ~VDI 3400 A2)	Coarse Texture (e.g., ~VDI 3400 A3)
External Surface	0.5° - 1.0°	1.0° - 1.5°	1.5° - 2.0°	2.0° - 3.0°
Internal Surface (Core)	0.5° - 1.0°	1.5° - 2.0°	2.0° - 2.5°	2.5° - 3.0°
Ribs	1.0° - 1.5°	1.5° - 2.5°	2.0° - 3.0°	≥ 3.0° recommended
Deep Cavity Structures	Draft on external surface > Draft on internal surface, to prevent core deflection and ensure uniform wall thickness.			

LEDA's Practice: At LEDA, checking draft angles is a core part of our DFM analysis. Based on your 3D model, and considering material, texture, and structural depth, we perform a detailed draft angle analysis and provide specific optimization suggestions to ensure your design has excellent manufacturability .

4.3 Golden Rules for Part Feature Design

Part feature design is the key step in transforming a concept into a manufacturable entity. The rules outlined in this section are crystallized from successful collaboration experiences between LEDA Precision and numerous clients, aimed at helping you avoid common defects at the source and enhance product quality.

4.3.1 Uniform Wall Thickness Design and Sink Mark Prevention Techniques

Uniform wall thickness is the primary principle of injection molding design. Non-uniform wall thickness causes different cooling rates, leading to internal stresses, warping, and sink marks on the surface in thicker sections, severely affecting the part's appearance and strength .

Core Design Rules:

Aim for Uniform Wall Thickness: Where functional and requirements permit, the wall thickness of the entire part should be as consistent as possible. A common range is typically 2-4mm, depending on part size and material .

Transition Principle: Gradual is Better than Abrupt: When wall thickness variation is unavoidable, transitions must be gradual. The slope of the transition zone should be controlled within a 3:1 ratio, meaning for every 3mm increase in thickness, the transition length should be at least 1mm. This effectively avoids defects caused by flow hesitation and stress concentration.

Sink Mark Prevention Techniques:

Coring Out: For thick sections that must exist, such as supporting pillars or screw bosses, coring out should be used. By hollowing out the interior to create a uniform thin wall, sink marks are prevented at the root cause .

Application of Ribs: Instead of increasing the overall wall thickness for strength, it is more effective to design ribs in appropriate locations. This is an efficient method to achieve lightweighting while ensuring structural strength .

LEDA Case Study: A consumer electronics housing initially had a boss root thickness of 4mm while the main wall was 2.5mm, causing visible sink marks. Following LEDA engineers' suggestion, the boss root was cored out in a "volcano" shape, reducing the effective wall thickness to 2.8mm and transitioning smoothly to the main wall at a 3:1 slope. This ultimately eliminated the sink marks while maintaining structural strength .

4.3.2 Ribs, Mounting Pads, and Anti-Warping Structural Design

These features are used to significantly increase the part's stiffness and stability without substantially increasing weight and thickness.

Golden Rules for Rib Design :

Design Parameter	Recommended Value	Rationale
Rib Thickness	40% - 60% of the main wall thickness	If too thick, it causes sink marks on the opposite surface; if too thin, filling is difficult and strength is insufficient.
Rib Height	≤ 3 times the main wall thickness	Excessively high ribs are difficult to fill and prone to drag marks during demolding.
Draft Angle	0.5° - 1.5°	Ensures smooth demolding; specific value depends on rib depth and material.
Root Fillet Radius	R ≥ 0.25 * main wall thickness	Avoids stress concentration at sharp corners and improves melt flow.

Mounting Pad Design: Mounting pads are used to support screws, bolts, or guide pins.

Internal Hole Design: The inner hole diameter should match the screw diameter, typically with a clearance of 0.2-0.3mm. The bottom should have a relief groove to prevent the screw tip from bottoming out and to aid demolding.

Support Structure: Tall pads must have gussets (triangular supports) added around them to prevent breaking or twisting. Similarly, the base of the pad should be cored out to prevent sink marks.

Anti-Warping Structure: For large flat parts, simply increasing the wall thickness is not the best solution and can easily lead to warping. The correct approach is:

Design Slight Crowns or Stiffening Lips: Designing a slight arch or reinforced edge on a flat surface can provide significant rigidity with minimal material, similar to an I-beam.

Rational Layout of Rib Networks: Creating a grid pattern with 纵横交错 (crisscrossing) ribs effectively distributes stress and prevents deformation.

4.3.3 BOSS Design and Coordination

BOSSes (screw bosses) are key load-bearing points in assemblies, and their design directly affects the connection strength and reliability of the product.

BOSS Design Rules :

Inner/Outer Diameter Ratio: The BOSS outer diameter (OD) is typically 2.0 - 2.4 times the screw diameter. For example, a boss for a self-tapping M3 screw should have an OD of about 6.0mm.

Blind Hole Design: The top of the BOSS should be a blind hole, with the bottom wall thickness ≥ 0.8mm to prevent breakthrough by the screw.

Avoid Isolation: Tall BOSSes must be connected to surrounding walls or the base plate via ribs to form a stable support. The rib thickness should also follow the 50% of main wall thickness rule.

4.3.4 Snap-fits, Living Hinges, and Other Elastic Connection Designs

Elastic connection designs enable fast assembly between parts without additional fasteners, exemplifying Design for Assembly (DFA).

Snap-fit Design:

Snap Arm: This is the flexible part of the snap-fit. The key to its design is allowing space for deflection. The thickness at the root of the arm is critical for calculating its flexibility and strength.

Cantilever Beam Formula: Deflection $\delta = (P * L^3) / (3 * E * I)$. This formula can estimate assembly force and lifespan.

Lead-in Angle and Return Angle: A lead-in angle (typically 30°-45°) provides a smooth assembly feel; a return angle (typically 45°-90°) prevents accidental disengagement. The return angle is usually larger than the lead-in to ensure one-directional movement.

Key Parameter: The snap engagement (undercut) is typically 0.4-0.6mm. Too small risks disengagement; too large makes assembly difficult and may cause excessive permanent deformation.

Living Hinge Design:

Material Selection: Polypropylene (PP) and Polyethylene (PE) are ideal materials due to their excellent fatigue resistance.

Thickness and Clearance: The hinge area should be significantly thinner (typically 0.25-0.5mm), and sufficient clearance must be provided at the bend to allow it to flex thousands of times without failure.

Fiber Orientation: For living hinges, the gate location is crucial. The melt flow direction should be perpendicular to the hinge axis, aligning the polymer molecules along the hinge direction for optimal flex life.

4.3.5 Key Points for Fillet and Hole Design

Fillet (Rounded Corner) Design:

Eliminate Stress Concentration: All internal and external sharp corners must be transitioned with fillets. Sharp internal corners are stress concentration points and origins of cracks under impact. Recommended internal fillet radius is R ≥ 0.5mm, and external radius R1 ≈ R + wall thickness.

Improve Filling: Fillets significantly improve the flow of molten plastic, enabling smoother filling and reducing flow lines and air traps.

Hole Design:

Spacing and Edge Distance: The distance from the hole edge to the part edge, and the distance between holes, should be at least 1.5 times the hole diameter. Insufficient edge distance weakens the structure and leads to weak weld lines during injection.

Avoid Sharp Corners: The start and end points of holes should also have small fillets or chamfers.

Blind Hole Depth: Blind hole depth should not be excessive; typically, depth ≤ 4 times the hole diameter is recommended, otherwise the core pin is prone to bending or breaking.

Through Hole Design: Prefer through holes. When designing, model from both sides of the part so the core pins form an interlocking structure when the mold closes. This is stronger and gives longer mold life compared to a "butt" seal formed from one side.

LEDA Summary: Part feature design is a combination of rationality and sensibility. It must follow the basic rules based on materials science and fluid mechanics, while also requiring trade-offs based on appearance, cost, and priorities in specific projects. At LEDA, we strongly recommend submitting your 3D model for a Design for Manufacturability analysis upon completion. Our engineers will use professional software and experience to conduct a comprehensive check of wall thickness, draft, fillets, potential sink marks, etc., and provide a detailed optimization report, working with you to create the perfect product .

4.4 Gating System Design: From Mold Flow Analysis to Gate Selection

The gating system is the channel through which molten plastic is injected from the injection molding machine nozzle into the mold cavity. Its design directly determines filling efficiency, part quality, and production costs. A well-designed gating system prevents defects such as short shots, sink marks, and weld lines, forming the cornerstone of successful precision injection molding .

4.4.1 Runner, Gate Type, and Location Selection Strategy

1. Runner Design: Balancing Efficiency and Material Loss

The runner is the channel connecting the sprue to the gates. Its core task is to deliver plastic uniformly to each cavity with minimal pressure loss and heat dissipation .

Cross-sectional Shape: A **circular cross-section** offers the highest efficiency due to its minimal surface area-to-volume ratio, reducing heat loss, but requires machining on both mold halves, increasing cost. **Trapezoidal and U-shaped** sections are common compromises, easier to machine on one side .

Dimensional Balancing: The runner diameter needs preliminary calculation based on plastic flow rate (part weight) and flow length. The layout must be balanced; for multi-cavity molds, a **naturally balanced layout** (equal flow length to each cavity) is essential to ensure simultaneous filling of all cavities .

Size Consideration: Runner size is often determined as a percentage of the part's thickness, typically ranging from 1.5 to 1.8 times the nominal wall thickness, to prevent premature solidification .

2. Gate Types and Selection: Precise Matching to Product Requirements

The gate is the narrow channel between the runner and the cavity and is critical for controlling filling quality. Common gate types and selection strategies are summarized below :

Gate Type	Core Characteristics	Application Scenarios	Design Notes
Pin Point Gate	Small size (typically Ø0.5-1.5mm), automatic degating, minimal mark visibility.	Cosmetic parts like mobile phone housings, multi-cavity molds.	Requires a three-plate mold structure; higher injection pressure loss.
Submarine (Tunnel) Gate	"Hidden" entry from underneath or side, automatic degating.	Parts with internal space allowing gate ejection, suitable for automation.	Difficult degating for tough plastics (e.g., PC); sufficient ejection force must be ensured.
Edge Gate	Simple machining, easy size adjustment, wide material compatibility.	Suitable for most plastics except polycarbonate (PC), often for edge-gated flat parts.	Leaves a visible mark on the side wall, may require post-processing.
Fan Gate	Wide entry, reduces orientation stress, promotes stable flow.	Large flat parts, prevents flow lines.	Difficult gate removal, leaves a more noticeable mark.
Direct (Sprue) Gate	Minimal pressure loss, effective packing.	Large, deep cavity parts.	Leaves a large mark on the part, difficult removal, high stress around the gate.

3. Golden Rules for Gate Location Selection

Gate location directly impacts part appearance, strength, and warpage. Core principles must be followed :

Place in Thick Sections: Ensures plastic flows from thick to thin areas, facilitating pressure transmission and reducing sinks.

Avoid Jetting and Flow Marks: The gate should be aimed at a core or wall to prevent material jetting, which causes snake-like patterns. Fan gates or tab gates can disperse the flow.

Control Weld Line Position and Quality: Weld lines, formed where two flow fronts meet, are weak areas. Gate location should position weld lines in non-critical stress or non-visible areas.

Facilitate Venting: Ensure the flow front can push air smoothly towards vents, avoiding air traps causing burns or short shots.

Avoid Impingement on Cores/Inserts: Prevents core shift or insert deformation from direct flow impact.

LEDA Practice: At LEDA, we use mold flow analysis software to simulate filling patterns, weld line locations, and air trap distributions during the design phase, providing visual comparisons to mitigate risks at the source .

4.4.2 Using CAE Mold Flow Analysis (e.g., Moldflow) to Optimize the Gating System

Modern injection molding design relies on CAE mold flow analysis software for digital twin simulation, predicting and optimizing the injection process before mold manufacturing .

Filling Pattern Verification: Software visually shows how melt fills the cavity, identifying over-packing, short shots, and air traps. Engineers can adjust gate location/size to ensure balanced filling .

Weld Line and Air Trap Prediction: Software accurately predicts the location and strength of weld lines and air pockets, allowing pre-emptive optimization through gate adjustment or vent addition .

Shrinkage and Warpage Prediction: Analysis simulates differential shrinkage from molecular orientation and uneven cooling, predicting warpage trends. This allows optimization of packing curves and cooling systems .

Runner System Auto-Balancing: For complex multi-cavity molds, CAE software can automatically calculate optimal sizes for unbalanced runners, achieving dynamic pressure balance .

4.4.3 Gate Influence on Weld Lines, Air Traps, and Defect Control

The gate's design directly dictates defect formation .

Weld Line Control:

Cause: Formed when flow fronts split and rejoin. If the temperature is too low upon reunion, the weld line is weak and visible.

Optimization: Increasing the number of gates or using fan/film gates can change flow paths. Optimizing process parameters (mold temp, melt temp, injection speed) improves weld line strength .

Air Trap Control:

Cause: Trapped air compressed at the flow front causes burns.

Optimization: Optimal gate location creates a "wave-like" flow to push air to vents. CAE analysis identifies air trap locations for precise vent placement .

LEDA Summary: Gating system design combines science and art. At LEDA, we integrate CAE mold flow analysis into our standard service, collaborating with clients to review results and find the optimal balance between function, aesthetics, and cost, ensuring a solid foundation for successful mold trials and stable production .

4.5 Cooling System Design: Pursuing Efficiency and Uniformity

In the injection molding cycle, cooling time can account for over two-thirds of the total cycle time. An optimized cooling system is the cornerstone of efficient production and high-quality parts .

4.5.1 Cooling Channel Layout Principles

The core goal of the cooling system is to achieve rapid and uniform heat removal. An improper layout causes uneven mold temperatures, leading to differential shrinkage, which is a primary cause of warping, sink marks, and high internal stresses .

Follow the "Equal Distance" Principle: Cooling channels should surround the cavity as evenly as possible. The distance from the channel centerline to the cavity surface should be consistent, typically 1.5–2 times the channel diameter.

Ensure Turbulent Flow: Coolant must be in a turbulent state (Reynolds number Re > 4000) for efficient heat transfer, achieved through adequate flow rate and channel diameter.

Control Inlet/Outlet Temperature Difference: The length of a single cooling circuit should not be excessive, ensuring the temperature difference between inlet and outlet is less than 5°C (ideally <3°C for precision parts). Large molds should use manifolds for parallel circuits.

Strengthen Cooling in Hot Areas: The gate area, where the hottest melt enters, requires intensified cooling to prevent local defects.

Avoid Interference and Dead Ends: Channel design must avoid collisions with ejector pins, sliders, etc., and form complete circuits without dead spots (blind holes) where coolant stagnates .

4.5.2 Application of Different Cooling Technologies

While standard drilled straight channels suffice for conventional molds, advanced techniques are needed for complex shapes or deep cores .

Cooling Technology	Description / Schematic	Application Scenarios & Characteristics
Baffle	A metal plate inserted into a deep channel to direct flow to the bottom, creating a up/down circuit.	Used for cooling deep cavities or cores where through-holes are impossible. Simpler machining but lower efficiency due to potential dead zones.
Bubbler	A tube inserted into a core; coolant sprays from the tube and flows back along the outer wall.	Also for deep cores. Efficient direct impingement cooling, but flow distribution may be uneven, prone to clogging.
Beryllium Copper (BeCu) Inserts	High thermal conductivity inserts (3-4x that of mold steel) embedded in hard-to-cool	A cost-effective solution for localized hot spots, rapidly conducting heat to the cooling system.

Cooling Technology	Description / Schematic	Application Scenarios & Characteristics
	areas (e.g., small cores).	
Conformal Cooling	3D-printed channels designed to follow the part's contour closely in three dimensions.	The ultimate cooling technology, enablingUltimate uniform and efficient cooling. Can significantly improve efficiency but presents challenges and higher costs in traditional steel molds.

4.5.3 Impact of the Cooling System on Production Cycle and Part Warpage

Decisive Impact on Production Cycle

The injection molding cycle consists mainly of injection, packing, and cooling times, with cooling often taking 70-80%. Cooling time is proportional to the square of the maximum wall thickness. Therefore, an efficient cooling system directly increases capacity by significantly reducing cycle time .

Fundamental Impact on Part Warpage

Warpage is caused by internal stresses from uneven shrinkage, primarily due to non-uniform cooling .

Mechanism: If one mold side (Side A) cools faster than the other (Side B), Side A solidifies first. Side B, remaining molten longer, contracts upon cooling but is constrained by the solidified Side A, causing the part to bend towards the hotter side (Side B).

LEDA's Solution: At LEDA, we use CAE software to predict warpage trends from cooling imbalances during design. The analysis visually shows mold temperature distribution, guiding channel layout optimization to minimize temperature differences from the source .

LEDA Summary: The cooling system is far from simple drilling; it integrates thermodynamics, fluid mechanics, and materials science. At LEDA, we employ cooling system CAE analysis for high-end projects. Through scientific simulation and optimization, we deliver shorter lead times and ensure superior dimensional stability and mechanical performance, adding value from the manufacturing end .

4.6 Ejection and Venting System Design

The ejection and venting systems are the "last mile" of an injection mold, directly determining whether the product can be released from the mold completely, undamaged, and efficiently. Excellent ejection design ensures the part is not deformed or marked by "blushing"; a rational venting system avoids defects like burns and short shots caused by trapped air. Together, they guarantee production stability and product yield .

4.6.1 Ejection Method Selection: Precise Force Application and Balanced Demolding

The core of the ejection system lies in precisely matching the ejection method to the product's structural characteristics, ensuring the ejection force is evenly applied to areas with the highest holding force while avoiding damage or visible marks on the part.

The table below compares four core ejection methods, serving as a quick reference for preliminary selection.

Ejection Method	Core Characteristics & Advantages	Typical Application Scenarios	Design Key Points
Ejector Pins	Most versatile, economical. Round pins are standardized and easy to machine. Blade (flat) pins suit deep ribs (depth ≥ 15mm).	Vast majority of plastic parts, especially shells and boxes requiring multi-point ejection. Blade pins are dedicated to deep, narrow ribs or long, thin sections.	1. Balanced Layout: Pins should be distributed in a triangular pattern for balanced force, preventing part twisting during ejection. 2. Location & Size: Prioritize locations with high structural strength (e.g., ribs, bosses). Diameter should be ≥ 1.5 times the wall thickness to prevent bending. 3. Anti-Rotation & Fit: Use D-shaped pins on curved surfaces; effective engagement length should be 20-25mm, with relief (0.5mm gap) elsewhere to reduce friction.
Stripper Plate	Ejection force is uniform, stable, with large contact area, leaving no individual pin marks.	Deep, thin-walled cylindrical parts, transparent parts (e.g., lenses, cosmetic containers), and appearance parts where ejector marks are unacceptable.	1. Fit Structure: A tapered fit (3°-10° angle) between the stripper plate and core is recommended to reduce wear and guide movement. 2. Clearance Control: Typical single-sided clearance between the plate and core is 0.20-0.30mm. 3. Connection & Guidance: The plate must be connected via screws to return pins and guided by linear bushings for smooth movement.
Ejector Sleeve	Suitable for annular or cylindrical deep bosses with a center hole. Ejection is peripheral, ensuring part concentricity.	Gears, parts with bushings, deep bosses (height ≥20mm).	1. Strength Requirement: Sleeve wall thickness must be ≥1.0mm to prevent damage. 2. Fixing Method: The ejector pin (core pin) inside the sleeve must be securely fixed to the backing plate using headless screws or spacer blocks.
Air Ejection	Non-contact ejection. Uses compressed air to overcome vacuum adhesion, especially for deep cavities (L/D >5) or soft materials (TPE/TPU).	Deep-cavity thin-walled parts, large shells, automotive soft-touch pads – parts prone to vacuum deformation.	1. System Composition: Requires designing air channels within the mold, connected to the injection machine's air supply and controlled by a timing valve. 2. Timing Precision: Air blast timing after mold opening requires an error of ≤0.1 seconds to avoid premature pressure loss failure.

LEDA Practice: At LEDA, we use motion simulation software to simulate the movement of lifters and ejector pins during the design phase, identifying and resolving potential interference issues upfront to ensure the reliability and safety of the ejection system .

4.6.2 Venting System Design: Depth Standards and Location Strategies

The purpose of the venting system is to allow air in the cavity and gases generated from heated plastic to escape smoothly during injection. The essence of its design lies in three elements: **"Depth, Location, Form."**

1. Vent Depth Standards

Vent depth is critical for controlling whether flash occurs. It primarily depends on the melt viscosity (flowability) of the plastic used.

Reference Vent Depth for Common Plastics:

Good Flowability (e.g., PA, PE, PP): 0.015 - 0.025 mm

Medium Flowability (e.g., ABS, PS, AS, PMMA): 0.03 - 0.04 mm

Poor Flowability (e.g., PC, PVC, PPO): 0.05 - 0.06 mm

Vent Structure: Typically a two-stage design.

Primary Vent (adjacent to cavity): Depth as per the above standards, length generally 3-4mm.

Secondary Vent: After the primary vent, depth increased to 0.5-0.8mm or even 1.0mm, width expanded, to quickly channel gases to the atmosphere.

2. Vent Location Strategy

Follow the "30% Rule": Approximately 30% of the fill length from the gate, injection pressure can still push gas out naturally at the parting line. The remaining 70%-100% end-of-fill zone is a critical area for trapped air and must be heavily vented.

Weld Line Areas: Areas where melt fronts meet are prone to air entrapment; vents should be added to improve weld line strength.

Insert, Core, and Ejector Pin Clearances: Cleverly using the fit clearance of ejector pins and inserts (single-sided 0.04mm) for auxiliary venting is a cost-effective method. For areas difficult to vent, like deep cavity bottoms, porous metal blocks (e.g., venting steel) can be embedded .

4.6.3 Defects from Trapped Air and Systemic Solutions

Trapped air not only causes product defects but can severely damage the mold in serious cases. Below are typical defects and systemic solutions.

Trapped Air Defect	Manifestation & Cause	Systemic Solutions
Surface Burning (Charring)	Brown or black spots on the part. Caused by gas being rapidly compressed, generating instant temperatures of 1000-2000°C, locally degrading the plastic.	1. Optimize Venting: Add vents meeting depth standards at the end of fill. 2. Adjust Process: Appropriately reduce injection speed to allow more time for gas escape.
Short Shot (Incomplete Fill)	The end of the part or thin sections are not filled. Gas cannot escape, creating counter-pressure preventing complete cavity fill.	1. Add Vents: Add vents or venting inserts at the end of fill. 2. Mold Flow Analysis (CAE): Use software to predict

Trapped Air Defect	Manifestation & Cause	Systemic Solutions
		trapped air locations upfront, avoiding issues at the design source.
Vacuum Adhesion & Demolding Deformation	During demolding of deep parts, a vacuum forms between the part and the core, creating huge demolding force, causing parts to "white" or distort.	1. Air Ejection Assist: For deep cavities, use air ejection to break the vacuum instantly upon ejection. 2. Innovative Venting Ejection: Use venting ejection systems made from materials like porous steel, which vent during injection and allow gas introduction during ejection, combining both functions.

LEDA Summary: Ejection and venting are inseparable "twin" systems in precision mold design. At LEDA, we use CAE mold flow analysis as a standard procedure to accurately predict end-of-fill, potential air traps, and ejection stress distribution during the product design phase. Combined with rich field debugging experience, we tailor balanced, efficient, and reliable ejection and venting systems for you, ensuring stable mold production and high product quality .

4.7 Shrinkage and Warpage Control: Comprehensive Strategies from Design to Process

Shrinkage and warpage are the most core and challenging quality issues in injection molding. Shrinkage is the linear reduction in part size, while warpage is the shape distortion caused by uneven shrinkage. The essence of controlling warpage lies in managing the uniformity of shrinkage. This section provides a systematic solution approach from design to process .

4.7.1 Accurate Prediction and Compensation of Shrinkage

The primary step in mold design is accurately predicting the material shrinkage and compensating the cavity dimensions. Inaccurate prediction makes all subsequent optimization less effective.

1. Understanding the Nature of Shrinkage

Shrinkage is not a fixed value but a variable determined by multiple factors:

Material Type: Amorphous plastics (e.g., ABS, PC, PS) have lower and more uniform shrinkage; semi-crystalline plastics (e.g., PA, POM, PP) have significantly higher shrinkage, greatly influenced by process conditions.

Molecular/Fiber Orientation: Shrinkage differs between the flow direction (MD) and transverse direction (TD) - anisotropy - especially pronounced in fiber-reinforced materials.

Process Parameters: Packing pressure/time, melt and mold temperatures decisively influence the actual shrinkage.

2. Advanced Prediction Methods: Beyond Empirical Values

Relying solely on the single shrinkage range provided by material suppliers can no longer meet precision manufacturing requirements. LEDA Precision adopts the following methods for accurate prediction:

CAE Mold Flow Analysis: Software like Moldflow performs "Fill + Pack + Warp" analysis, simulating a cloud map of shrinkage distribution across different part areas, not just a single value. This allows anticipating issues before mold manufacturing.

Digital Compensation Technology: Based on CAE results, LEDA uses reverse deformation compensation technology. By inputting deformation results into global modeling software (e.g., Siemens NX), a pre-deformed cavity model is generated. After the part cools and shrinks, its dimensions and shape meet the design requirements .

LEDA Practice: For a precision gear made of PA6+30%GF, the material supplier's shrinkage range was 0.4%-0.8%. CAE analysis showed the actual shrinkage was about 0.45% at the tip diameter but 0.65% at the hub due to greater thickness. LEDA applied differential compensation to the mold cavity, resulting in final gear tip diameter accuracy consistently controlled within ±0.01mm .

4.7.2 Root Causes of Warpage and Design Avoidance

Warpage is the final manifestation of defects; its root causes must be addressed at the source in product and mold design.

The table below summarizes the main causes of warpage and key design avoidance strategies:

Warpage Root Cause	Generation Mechanism	Golden Rules for Design Avoidance
Non-Uniform Wall Thickness	Thick sections cool slower and shrink more; thin sections cool faster and shrink less. Differential shrinkage causes bending towards the thicker side.	Pursue uniform wall thickness. Core principle: The maximum wall thickness anywhere should not exceed 1.5 times the adjacent minimum wall thickness. For necessary thick sections (e.g., bosses), use coring.
Uneven Cooling	Excessive temperature difference between the core and cavity sides of the mold, or across different mold areas, causes different cooling rates on the part's sides.	Optimize the cooling system. Ensure cooling channels are evenly laid out (follow "equal distance" principle), intensifying cooling at gates and thick sections. For deep cores, use advanced techniques like baffles, bubblers, or beryllium copper inserts.
Molecular/Fiber Orientation	Differences in shrinkage between flow and transverse directions cause the part to warp, shrinking more in the flow direction.	Optimize gate location and number. Use balanced runners and multiple gates to shorten flow paths and reduce orientation differences. For large flat parts, avoid a single edge gate; use a film gate or multiple point gates.
Asymmetric Geometry	The part geometry itself is asymmetric (e.g., ribs on one side only), causing unbalanced forces during cooling and shrinkage.	Reinforce symmetric design. Design the geometry to be symmetrical where possible. For example, changing a "T" shape to a "cross" shape can significantly improve shrinkage uniformity and reduce warpage.

4.7.3 Minimizing Deformation through Process Parameter Optimization

Once the design and mold are finalized, process parameters are the last and most flexible line of defense for controlling deformation.

1. Packing Profile Optimization: The Core of Warpage Control

The packing phase is key to compensating for shrinkage. The strategy is far more important than a constant pressure.

Packing Pressure: Excessively high pressure creates high internal stress, released after ejection causing warpage; excessively low pressure fails to compensate for shrinkage, leading to sinks and undersize. Find the optimum pressure that compacts the material without creating excessive stress.

Packing Time: Packing time must continue until the gate freezes sealed. If too short, melt flows back into the runner, causing severe shrinkage.

Multi-Stage Packing: Use a graduated pressure profile (high to low). Initial high pressure quickly compensates for shrinkage, followed by gradually reduced pressure to relieve some internal stress. This balances sink marks and warpage better than constant pressure.

2. Temperature Management

Mold Temperature: Uniform mold temperature is more important than its absolute value. LEDA recommends multi-zone mold temperature controllers to ensure balanced temperatures between core, cavity, and different areas, ideally within a 5°C difference. Appropriately higher mold temperature aids molecular chain relaxation, reducing orientation stress, but extends cycle time.

Melt Temperature: Within the material's recommended range, a lower melt temperature can reduce overall shrinkage. But too low a temperature increases orientation stress. Adjust in coordination with injection speed.

3. Injection Speed

Use a slow-fast-slow segmented injection strategy:

Start Slow: Prevents jetting, fills the gate area smoothly.

Middle Fast: Fills most of the cavity quickly, reducing flow lines.

End Slow: Reduces the impact of the melt front on the cavity end, aiding venting and reducing residual stress.

LEDA Summary: Shrinkage and warpage control is a systematic project that cannot be solved by a single measure. At LEDA, we provide full-process stability assurance from design to mass production through a three-step method: **"DFM + CAE Mold Flow Analysis + DoE (Design of Experiments) Process Window Optimization"**. For example, CAE predicts 97% of potential deformation risks upfront, and then DoE quickly locks in the optimal process parameters during mold trials, achieving high-quality mass production with maximum efficiency .

4.8 Surface Treatment: Texture, Etching, and High-Gloss Surfaces

Surface treatment is the critical step that elevates an injection molded product from merely "functional" to "tactile and premium." It not only bestows unique aesthetic value but also directly impacts user experience, functionality, and production costs. Selecting the right surface treatment process requires a deep understanding of its design logic and manufacturing constraints .

4.8.1 Design Considerations for Different Grain (Etching) Grades

Grain texture, also known as mold etching or texturing, is a process that uses chemical solutions to corrode the mold steel surface, creating various micro-level concave-convex patterns. Its core purposes are to enhance aesthetic texture, conceal processing defects, provide a comfortable tactile feel, and reduce light reflection .

1. Texture Grades and Standards

The industry typically uses standards to define the coarseness of textures. The **VDI 3400** standard is widely adopted in Europe, grading textures from fine to coarse (e.g., VDI 18, VDI 24, VDI 40) with corresponding surface roughness (Ra) values. Coarser textures (higher VDI values) result in a more matte finish on the product surface; conversely, finer textures yield a higher gloss . Another common standard is the **Mold-Tech** standard, which offers a richer library of texture styles, such as leather, geometric, and linen patterns .

2. Selection Logic for Texture Grade

The choice of texture grade requires a comprehensive trade-off based on the product's end-use scenario.

Fine Texture (e.g., VDI 18-24): Offers exquisite texture, smooth feel, and easy cleaning. Often used in consumer electronics, small appliance panels, and medical devices where high cleanliness and frequent contact are required .

Medium Texture (e.g., VDI 27-40): Provides good texture, effectively hides defects like parting lines and sink marks, and offers moderate scratch resistance. A common choice for automotive interior parts, tool housings, and large home appliances .

Coarse Texture (e.g., VDI 45 and above): Features strong three-dimensionality, noticeable tactile feel, and excellent anti-slip properties, but can trap dirt. Primarily used for non-slip surfaces (e.g., tool handles), internal structural parts needing strong concealment, or designs pursuing a specific style .

3. Core Design Considerations and Mold Preparation

Once the texture grade and type are determined, targeted adjustments must be made during product design and mold manufacturing; otherwise, production failure is highly likely.

Draft Angles: This is the most critical, yet most easily overlooked, parameter in grain texture design. The texture significantly increases friction between the product and the mold cavity. To ensure smooth demolding and avoid "whitening" or even "tearing" of the part, draft angles must be substantially increased based on the texture depth. A fundamental rule of thumb is: for every 10μm increase in texture depth, the draft angle should be increased by at least 1°. For example, a grain texture with a depth of 35μm requires a recommended draft angle of ≥ 4° on the corresponding sidewall .

Mold Surface Preparation: The mold must be meticulously prepared before etching. The cavity surface needs to be uniformly polished to a specified finish (e.g., fine textures require sandpaper #1000 and above, coarse textures can use #600) and must be free of machining marks, weld lines, or EDM marks. Any surface defect will be amplified after texturing .

Parting Line and Structural Boundaries: A smooth border (light edge) of at least 0.3-1.0mm should be reserved from the textured area to the parting line or insert edge to prevent the formation of thin, sharp flash. For textures on moving components like sliders and lifters, seamless alignment with the texture on the fixed cavity must be ensured to avoid mismatches .

LEDA Practice: At LEDA, we strongly advise clients to obtain physical samples from grain texture suppliers before finalizing the appearance. Our engineers perform precise analysis of draft angles in textured areas using 3D software during the DFM stage and provide differentiated draft angle recommendations for varying texture depths, ensuring production feasibility from the design source. For complex curves, we evaluate the possibility of using new processes like laser-chemical composite etching to solve potential issues like texture distortion or visible seams on arcs and deep cavities common with traditional etching .

4.8.2 Design and Mold Conditions for Achieving High-Gloss (Mirror) Surfaces

High-gloss surfaces pursue ultimate smoothness and mirror effects, placing extremely stringent demands on design and moldmaking, as the slightest defect will be glaringly obvious .

1. Design Requirements for High-Gloss Surfaces

High-gloss finishes amplify all details, so product design must adhere to the principle of "ultimate simplicity."

Absolutely Uniform Wall Thickness: Any variation will cause visible sink marks or light distortion on a high-gloss surface. The thickness of ribs and boss pads must be controlled within 0.3-0.4 times the main wall thickness, with sufficient root fillets for transition .

Avoid Weld Lines: Weld lines are the "natural enemy" of high-gloss surfaces. Mold design (e.g., using hot runner valve gate sequencing, optimizing gate location) and process adjustments are necessary to eliminate weld lines or relocate them to non-appearance surfaces .

Simplify Structure: Avoid complex undercuts and side holes on primary appearance surfaces to minimize parting line marks from sliders and lifters .

2. Stringent Conditions for High-Gloss Molds

The high-gloss effect is predominantly determined by the superior quality of the mold.

Mold Steel: Must use high-purity, fine-grained mirror polish steels, such as Sweden's S136 or Japan's CEANA1. These steels have low impurities, are easy to polish to a perfect mirror finish, and offer long life .

Mirror Polishing: The cavity surface needs to be polished to a #A1 grade or above (equivalent to Ra ≤ 0.01 μ m). This requires mold technicians with exceptional polishing skills and is very time-consuming .

High-Temperature Mold Technology (RHCM): To achieve the high-gloss effect and eliminate flow marks and weld lines, Rapid Heat Cycle Molding (RHCM) is used. This involves rapidly heating the mold to 80-130°C (depending on the material) before injection to optimize melt flow, then rapidly cooling it after packing. This "rapid heat-cool" process demands highly sophisticated heating/cooling channel design within the mold .

Excellent Venting System: Trapped air causes surface burns. High-gloss molds must have precisely designed and adequate venting slots (typically 0.015-0.03mm deep, depending on the plastic) to ensure gas can escape smoothly .

LEDA Summary: Surface treatment is the intersection of aesthetics and engineering. At LEDA, we integrate surface treatment requirements into the earliest stages of product design. Whether opting for a concealing grain texture or challenging the perfection of a high-gloss finish, we provide end-to-end solutions from material recommendation, structural optimization, mold steel selection, to final process implementation, ensuring your design vision is efficiently and economically translated into reality .

4.9 Industry Application Focus: Consumer Electronics Housings vs. Medical Connectors

Consumer electronics housings and medical connectors represent two extremes in the spectrum of injection molding applications. The former prioritizes ultimate aesthetic appeal, thin-and-light design, and fast iteration cycles; the latter places biocompatibility, functional reliability, and regulatory compliance above all else. Understanding this difference in core requirements is the foundation for making correct design decisions .

The table below clearly contrasts the core design and manufacturing differences between these two fields.

Feature Dimension	Consumer Electronics Housings (e.g., Smartphones, TWS Earbud Cases)	Medical Connectors (e.g., for Patient Monitors, Surgical Robots)
Core Driver	Visual appeal, cost-effectiveness, assembly efficiency.	Biocompatibility, functional reliability, regulatory compliance.
Appearance Standard	Class A surface, high-gloss/matte finish, zero defects. Parting lines, gate vestige must be perfectly hidden.	Appearance often serves function; surface finish must facilitate cleaning/sterilization, avoid dead zones.
Structural Strength	Achieves "light yet strong" via dense ribs (50%-70% of main wall thickness) and snap-fits, ensuring drop-test compliance.	Ensures structural integrity under repeated mating/uncoupling and chemical sterilization; focuses on long-term aging performance.
Material Selection	Primarily ABS, PC/ABS blends, balancing strength, heat resistance, and cost; high-end parts use PC or metal+plastic composites.	Medical-grade PC, PP; must pass USP Class VI or ISO 10993 biocompatibility certification.
Precision Focus	Appearance gap/step must be controlled within ± 0.05mm, ensuring seamless assembly.	Extremely high terminal/hole precision (±0.02mm), ensuring absolute stability of electrical connection.
Mold Technology	Multi-cavity hot runner (valve-gated) for efficiency; high-gloss mirror molds (VDI 3400 A1 grade) for	Often few-cavity or single-cavity molds, emphasizing stability; mold steel must be corrosion-resistant (e.g., S136); cooling

Feature Dimension	Consumer Electronics Housings (e.g., Smartphones, TWS Earbud Cases)	Medical Connectors (e.g., for Patient Monitors, Surgical Robots)
	appearance.	system is critical.
Cleanliness	Control dust and particles to meet basic appearance and function.	Extremely high sterility requirements; production environment needs Class 10,000/1,000 cleanrooms; strict prevention of chemical/microbial contaminants.

4.9.1 Consumer Electronics: The Trade-off Between Appearance, Structural Strength, and Snap-fit Assembly

The design of consumer electronics involves precise trade-offs between appearance, structural strength, and assembly efficiency .

Appearance-First Design Philosophy

Parting Lines and Gates: Parting lines must be cleverly hidden along product edges or decorative seams. Submarine or pin-point gates are preferred, located internally or in concealed areas for "vestige-free" gating. For high-gloss surfaces, hot runner valve gate sequencing technology effectively controls weld line position, preventing them from appearing on primary appearance surfaces .

Wall Thickness and Stiffness: While pursuing thinness and lightness (wall thickness often 1.2-2.0mm), overall rigidity is ensured by scientifically designed ribs (50%-70% of main wall thickness, with large root fillets) to avoid flexing noise or impact failure. Uniform wall thickness is the iron rule for preventing sinks and ensuring a flat appearance .

Snap-fit Assembly and Poka-Yoke: Snap-fits are widely used for rapid assembly. Snap-fit design must include sufficient lead-in angles (30°-45°) and retention angles (45°-90°), providing clear auditory and tactile feedback. Poka-Yoke design is essential, using asymmetric posts/holes to ensure parts can only be assembled one correct way .

Driven by Cost and Efficiency

Mold Strategy: To meet massive demand, standard models use multi-cavity molds to maximize efficiency; flagship models may use single-cavity molds paired with multiple mold sets to ensure ultimate quality and disperse production risk .

Material Regrind: Regrind material is used according to strict ratios, provided quality standards are met, to control costs .

LEDA Case Study: A TWS earbud charging case housing had dense internal ribs and snap-fits. Mold flow analysis accurately predicted sink risk opposite the ribs, which was eliminated by adjusting gate location and optimizing the packing profile. The exterior used a cavity polished with 2700-grit diamond paste for a mirror finish, with the parting line perfectly hidden in the hinge gap .

4.9.2 Medical Devices: Key Points for Biocompatibility, Cleanliness, and High-Reliability Design

The design principles for medical connectors stem from respect for life, with risk control at the core .

Biocompatibility and Cleanliness as the Baseline

Material Certification: Must use fully certified medical-grade materials with complete documentation and traceability, far beyond standard commercial grades .

Eliminate Contamination Traps: Product design must avoid any internal sharp corners, crevices, and uncleanable dead zones. All internal corners require adequate fillets (R ≥ 0.5mm) for thorough cleaning and sterilization. Mold steel must be high corrosion-resistant stainless steel (e.g., S136 ESR) to prevent rust and contamination over time .

Designed for Reliability and Manufacturability

Connection Strength: Innovative design is key to preventing insert molding separation. Designing mechanical interlock features (e.g., through-holes, grooves) inside the connector body allows the molded material to form "anchors," greatly enhancing bond strength and preventing cracking or detachment .

Precision and Stability: Medical connectors are often small, deep-cavity parts, demanding high mold cooling efficiency and venting. Conformal cooling channels inside cores and precise venting slots (0.015-0.03mm deep) are often necessary to prevent burns, short shots, or stress cracking from trapped air .

Process Validation: Every step of moldmaking and production must follow strict validation protocols (IQ, OQ, PQ), ensuring every shot meets predefined standards for consistent quality .

LEDA Case Study: A connector for a medical surgical robot had precise internal terminals. By designing over eight anchoring through-holes on the connector body, the molded material formed a mechanical interlock, increasing pull-out strength by 300% and eliminating the risk of insert cracking from frequent mating. The mold used a hot-to-cold runner pin-point gate for top feeding, ensuring terminal position accuracy and uniform plastic filling .

LEDA's Summary and Recommendation

Facing these two distinct fields, LEDA's advice is:

Consumer Electronics: Your design thinking should be "Outside-In." Start with Industrial Design (ID) driving the process, with mechanical design fully committed to realizing the ID intent, finding the optimal balance between cost, strength, and assembly efficiency. The core of DFM lies in the ability to achieve ultimate aesthetics and the stability/efficiency of mass production .

Medical Devices: Your design thinking must be "Inside-Out." First satisfy functional reliability and regulatory compliance; all designs revolve around risk reduction. Appearance serves cleanability and function. The core of DFM lies in process verifiability, product traceability, and reliability design that exceeds the norm .

At LEDA, we have extensive experience serving both fields. Whether for fashion- and efficiency-driven consumer electronics or safety- and reliability-critical medical devices, our engineering team provides end-to-end solutions from material selection, DFM analysis, mold design and manufacturing, to production control, ensuring your product meets the highest industry standards from concept to mass production .

Chapter 5: Die-Casting Design Guide

Die-casting is a precision metal forming technology characterized by high pressure, high speed, and high temperature. It involves injecting molten metal under high pressure into a precision mold cavity, where it rapidly cools to form metal parts with complex shapes, precise dimensions, and smooth surfaces. This process is particularly suited for the high-volume production of metal structural components that demand high precision and consistency, holding an irreplaceable position in industries such as automotive, consumer electronics, and aerospace .

The hallmark of die-casting is its extremely high production efficiency and exceptional part accuracy. Cold-chamber die-casting machines can typically achieve 50-90 cycles per hour, while hot-chamber machines can reach 400-900 cycles, far surpassing the productivity of traditional casting methods. Die-cast parts can achieve dimensional accuracy grades of IT13-IT15, with higher precision even reaching IT10-IT11, and surface roughness (Ra) can be as low as 3.2-1.6 μm, even 0.8 μm locally, allowing many castings to be used directly with little or no machining .

However, behind a successful die-cast part lies the precise coordination of materials science, mold engineering, and process control. The design of the die-cast part directly determines the complexity of the mold structure, production stability, and the quality of the final product. Excellent die-casting design is not just about "whether it can be formed" but about whether high-quality products can be produced efficiently, economically, and consistently. A rational die-cast structure can simplify the mold, reduce mold manufacturing costs by over 30%, and significantly decrease the incidence of defects like porosity and shrinkage .

This chapter will serve as your practical guide, systematically analyzing the key design points for die-casting. You will learn that by following scientific design principles, most common defects can be avoided, product quality enhanced, and production costs reduced. We will start with the properties and selection of the three main die-casting alloys (Aluminum, Magnesium, Zinc), delve into the golden rules of structural design — including wall thickness uniformity, rib design, and fillet optimization — and explore core factors determining mold feasibility like parting line selection, gating system design, and venting system layout. We will also analyze optimization strategies for side core-pulling mechanisms to control mold complexity, cover coordination requirements for post-processing and surface treatment, and discuss diagnosis and prevention of common defects .

Whether you are designing automotive components for lightweighting or consumer electronics housings for high precision, the principles in this chapter are universally applicable. Mastering this knowledge will enable you to communicate efficiently and accurately with mold engineers and manufacturers (such as LEDA Precision), working together to transform your ideas into competitive die-cast products .

LEDA Precision Practical Perspective: Based on our extensive experience in die-casting, this chapter will emphasize the collaborative optimization of design, material, and process. We strongly advise involving your manufacturing partner early in the project. Through CAE simulation analysis and DFM (Design for Manufacturability) assessment, we can ensure product manufacturability from the source, avoiding costly design changes later .

5.1 Die-Casting Alloys (Al, Mg, Zn) Characteristics and Selection Guide

The selection of die-casting alloy is the core bridge connecting product design with the final part's performance, cost, and reliability. It directly determines the part's lightweight potential, mechanical strength, environmental resistance, and mass production economy. Among the various alloys, aluminum, magnesium, and zinc alloys form the most widely used trio, each with unique property profiles. This section provides an in-depth analysis of these three alloy types and offers a systematic selection decision matrix based on performance, cost, and process feasibility to pinpoint the optimal material solution for your project .

5.1.1 Aluminum Alloys: The Choice for High Performance and Overall Cost-Effectiveness

Aluminum alloys are the most widely used and technologically mature family in die-casting, dominating the structural parts sector for automotive, telecommunications, and industrial equipment. Their success lies in achieving an excellent balance between lightweight, mechanical properties, processability, and cost .

Core Advantages:

High Specific Strength: Their density (~2.7 g/cm³) is only one-third that of steel, but through alloying and heat treatment, their specific strength can rival or even surpass some alloy steels, making them the primary choice for lightweighting.

Excellent Corrosion Resistance: A naturally formed dense oxide film provides basic protection, which can be further enhanced by surface treatments like anodizing, allowing adaptation to harsh environments like humidity and salt spray.

Superior Thermal and Electrical Conductivity: Their thermal conductivity is much higher than steel, making them ideal for components requiring good heat dissipation, such as engine parts, heat sinks, and electronic device housings.

Common Die-Casting Aluminum Alloy Series:

Al-Si Series (e.g., ADC10/A380, ADC12/A383): The most commonly used die-casting aluminum series. The addition of Silicon (Si) significantly improves fluidity, making it ideal for complex, thin-walled parts and effectively reducing shrinkage tendency.

Al-Mg Series (e.g., 5xxx series): These alloys offer better strength and corrosion resistance, particularly resistance to seawater corrosion, but their castability is slightly inferior to Al-Si alloys, making them more suitable for applications with specific strength and corrosion resistance requirements.

Design Considerations:

Good Applicability: Aluminum die castings exhibit high dimensional accuracy, low surface roughness, and good interchangeability.

Process Flexibility: Supports various processing methods like extrusion, casting, and stamping, enabling complex cross-section profiles.

LEDA Recommendation: For most structural components (e.g., engine housings, transmission cases, 5G base station heat dissipation units), Al-Si series alloys are the default choice due to their excellent overall performance. LEDA can provide material certification data for various grades to aid precise selection .

5.1.2 Magnesium Alloys: The Choice for Extreme Lightweighting

Magnesium alloys are the lightest metallic structural materials used in engineering applications, with a density of about 1.8 g/cm³, which is two-thirds that of aluminum and one-quarter that of steel. When weight reduction is the primary goal, magnesium alloys are the irreplaceable solution .

Core Advantages:

Unparalleled Lightweighting Effect: Their high specific strength and high specific stiffness give them an absolute advantage in aerospace, high-end consumer electronics, and sports equipment. Reducing a car's weight by 10% can effectively lower energy consumption.

Excellent Damping Capacity and Heat Dissipation: Their damping capacity is more than 15 times that of aluminum alloys, effectively absorbing vibration and noise, improving product stability and user experience. Magnesium alloys also have good thermal dissipation properties, beneficial for electronic device cooling.

Good Electromagnetic Shielding (EMS): Can provide over 50dB of EMS effectiveness, making them ideal for 3C product housings to protect internal 精密 components.

Design Considerations:

Corrosion Resistance Needs Attention: Magnesium is relatively reactive chemically. Appropriate surface protection measures (e.g., micro-arc oxidation, electrophoretic painting, coating) during production, assembly, and use are necessary to ensure long-term durability.

Higher Cost: Raw material costs and specific requirements for the production environment (e.g., protective gas for melting) typically result in a higher per-part cost compared to aluminum alloys.

LEDA Practice: Magnesium alloys are ideal for applications extremely sensitive to weight and heat, such as projectors, drone gimbals, and high-end laptop housings. LEDA has mature expertise in magnesium die-casting, enabling effective risk control and leveraging its ultimate performance .

5.1.3 Zinc Alloys: The Choice for High Precision and Excellent Surface Quality

Zinc alloys dominate the manufacturing of parts requiring high precision, complex details, and excellent appearance, thanks to their unrivalled casting fluidity and superior surface quality .

Core Advantages:

Exceptional Fluidity and Forming Precision: Low melting point and excellent fluidity allow perfect replication of mold details, enabling the formation of thin walls and complex structures better than Al/Mg alloys. Castings have stable dimensions, suitable for precision parts like small gears and connectors.

Excellent Surface Quality and Decorability: Die castings have a high surface finish, suitable for secondary processing like plating and painting, easily achieving high-end appearances like mirror finishes and chrome plating, widely used in bathroom hardware and high-end consumer goods.

Good Mechanical Properties and Wear Resistance: Zinc alloys possess relatively high strength, hardness, and wear resistance, meeting the mechanical requirements of many structural functional parts.

Design Considerations:

Density and Weight: Their density (~6.6 g/cm³) is significantly higher than Al and Mg, making them unsuitable for weight-critical applications.

High-Temperature Performance: Mechanical properties degrade noticeably at high temperatures; the maximum service temperature is generally not recommended to exceed 100°C.

LEDA Recommendation: When your design focuses on micro-precision, complex appearance, and excellent plating effects (e.g., door locks, high-end model parts, smart lock panels), zinc alloy is the best choice. Its efficient forming cycle also benefits large-scale production .

5.1.4 Selection Decision Matrix Based on Performance, Cost, and Process

To aid your decision-making, the following matrix synthesizes core performance, cost, and process factors .

Decision Dimension	Primary Choice: Aluminum Alloy	Primary Choice: Magnesium Alloy	Primary Choice: Zinc Alloy
Core Objective	Pursues overall cost-effectiveness, balancing strength, corrosion resistance, and good heat dissipation.	Pursues extreme lightweighting, superior vibration damping, or efficient electromagnetic shielding.	Pursues ultra-high forming precision, complex details, and top-tier surface decorability.
Key Properties	High specific strength, good corrosion resistance, good	Lowest density, very high specific strength, good damping/heat	Exceptional cast fluidity, dimensional stability, high surface finish, good

Decision Dimension	Primary Choice: Aluminum Alloy	Primary Choice: Magnesium Alloy	Primary Choice: Zinc Alloy
	thermal conductivity.	dissipation, excellent EMS.	mechanical properties.
Cost Consideration	Balanced material and process costs, offering the best overall value.	Higher material cost, stricter production control requirements, typically highest total cost.	Material cost fluctuates with zinc price, but high production efficiency can offer cost advantages at high volumes.
Typical Applications	Automotive structural parts, engine housings, communication base station radiators, power tool housings.	Aerospace components, high-end laptop housings, camera bodies, drone parts.	Precision gears, bathroom faucets, door locks, high-end models, decorative parts.
LEDA Selection Advice	Default starting point. If uncertain, start evaluation with aluminum alloy; it's rarely wrong.	Strategic choice. Use when saving 1 gram far outweighs the cost, or when there are extreme requirements for heat dissipation/damping.	Functional/Decorative choice. Use when the product's selling point lies in exquisite appearance or micro-complex structures.

Recommended Decision Process:

Define Requirements: First, identify the product's core performance indicators (is it strength, weight, or appearance?) and the target cost range.

Preliminary Screening: Use the table above to narrow options to 1-2 alloys based on the core objective.

In-depth Validation: Collaborate with LEDA's engineering team for CAE simulation analysis (e.g., strength, flow field) and prototype trials for the initially selected materials to validate selection feasibility from the manufacturing end.

LEDA Summary: There is no single "best" material, only the most appropriate choice. Aluminum, Magnesium, and Zinc form a complete spectrum from cost priority to performance extremes. The value of LEDA Precision lies in helping you navigate complex technical parameters with our rich material database and engineering experience, making rational decisions that best align with your product's commercial goals .

5.2 The Golden Rules of Die-Casting Part Structural Design

The structural design of a die-cast part is the core bridge connecting product functionality with manufacturability. An excellent design not only achieves the product's performance targets but also significantly reduces mold complexity, enhances production stability, and controls overall costs. This section will delve into the six golden rules for the structural design of die-cast parts, providing you with comprehensive guidance from principle to practice .

5.2.1 Uniform Wall Thickness Design: Balancing Filling, Strength, and Weight Reduction

Achieving a thin and uniform wall thickness is the primary principle of die-casting design. This aims not only for weight reduction but also for higher comprehensive mechanical properties and better processability .

Advantages of Thin Walls: Die-cast parts with thin walls cool faster, resulting in a denser internal structure, which leads to higher strength and pressure resistance. Furthermore, it significantly shortens the molding cycle, saving material and energy .

Hazards of Non-uniform Walls: Variations in wall thickness cause differential cooling rates. Areas with thicker sections, known as "hot spots," cool slower, becoming the primary source of internal defects like shrinkage porosity and cavities, and are also the main cause of part warping .

Recommended Wall Thickness Ranges: Wall thickness should correspond to the part's projected area. The following is a general reference guide based on practical experience :

Projected Surface Area of Die Casting (cm²)	Recommended Wall Thickness (mm)
≤ 25	1.0 - 3.0
> 25 - 100	1.5 - 4.5
> 100 - 400	2.5 - 5.0
> 400	3.5 - 6.0

Generally, parts with wall thicknesses exceeding 6mm are not suitable for production using conventional die-casting processes .

Techniques for Achieving Uniform Walls:

Coring Out: For necessary thick sections, they should be cored out to maintain the macroscopic thick feature while becoming uniformly thin microscopically, fundamentally eliminating hot spots .

Gradual Transitions: When wall thickness variation is unavoidable, transitions must be gradual with a taper greater than 3:1, avoiding sudden sectional changes to reduce stress concentration and flow resistance .

LEDA Case Study & Suggestion: We assisted a client in optimizing a vehicle controller housing where the side wall was 3mm thick, but the mounting flange was 8mm. CAE mold flow analysis predicted a high risk of shrinkage porosity at the flange. We recommended redesigning the flange into an "I-beam" structure with ribs and coring, reducing the effective wall thickness to 4mm while ensuring mounting strength, and ultimately eliminating the defect by adding process fillets. At LEDA, we strongly advise conducting mold flow analysis early in the design phase to foresee and address risks from wall thickness variation visually .

5.2.2 Rib Design: The Art of Enhancing Stiffness and Preventing Deformation

When simply increasing wall thickness is insufficient to meet rigidity requirements, the rational design and layout of ribs are the most effective solution .

Core Functions of Ribs: Enhance the part's stiffness and strength, prevent warping, and can act as auxiliary channels for molten metal flow .

Golden Design Criteria for Ribs:

Parameter	Recommended Value or Rule	Design Rationale
Rib Thickness	50% - 60% of the main wall thickness (generally not exceeding	Excessively thick ribs cause sink marks on the opposite surface; too thin leads to filling issues and insufficient strength .

Parameter	Recommended Value or Rule	Design Rationale
	0.7 times) .	
Rib Height	Should not exceed 5 times the rib thickness .	Excessively high ribs have poor rigidity and are difficult for metal to fill.
Draft Angle	1° - 3° .	Ensures smooth demolding.
Root Fillet Radius	R ≥ 0.25 - 0.5 × Main Wall Thickness .	Eliminates stress concentration, improves metal flow, and increases part strength.
Layout Strategy	Follow the principal stress flow lines, using triangular stable structures .	Avoid simple orthogonal grids; use customized layouts based on stress analysis to maximize force transmission efficiency. Adding auxiliary ribs to form triangular structures on straight main ribs significantly enhances deformation resistance.

Innovative Design Case: For a vehicle's integrated die-cast rear floor rail, the traditional design used 52 ribs. Through an innovative custom layout following crash force transmission paths and changing some areas to triangular nested structures, the number of ribs was reduced to 24 while meeting higher crash standards, achieving a weight reduction of 3.5kg and a cost saving of 175 RMB, exemplifying the significant benefits of structural optimization .

5.2.3 Fillet Design: The Key to Eliminating Stress Concentration and Extending Mold Life

All junctions between walls on a die-cast part must have fillet radii. This is not an aesthetic requirement but a crucial technical specification related to structural reliability and manufacturability .

Three Core Functions of Fillets:

Eliminate Stress Concentration: Sharp internal corners are stress concentration points, origins of cracks under force or thermal shock. Fillets effectively distribute stress, preventing crack initiation .

Improve Molten Metal Flow: Fillets create smooth contours in the mold cavity, significantly reducing flow resistance, enabling smoother filling, avoiding turbulence and air entrapment, thus reducing defects like flow lines and oxide slag .

Extend Mold Life: Sharp corners in the mold cavity are prone to cracking under alternating thermal and mechanical stress. Fillet design effectively prevents this, greatly extending mold service life .

Fillet Radius Selection: The recommended fillet radius R is 0.5 - 1.0 times the wall thickness. Typically, the external fillet radius R1 = R (internal fillet radius) + wall thickness. The fillet radius for die-cast parts should generally not be less than 1mm .

5.2.4 Avoiding Thick Sections and Hot Spots: The Core of Preventing Shrinkage Porosity and Cavities

"Hot spots" are the last areas to solidify in a casting and are breeding grounds for shrinkage defects. Prevention is far superior to cure .

Defect Formation Mechanism: Thick sections dissipate heat slowly. When the exterior has solidified, the interior may still be liquid or semi-liquid. During solidification shrinkage, the external solid skeleton forms, and the interior cannot receive effective liquid metal feeding, leading to vacuum pores or fine shrinkage porosity .

DFM Preventive Measures:

Structural Coring: For features like bearing seats and mounting bosses, use spherical hollowing or pocket designs to turn solid thick walls into uniform thin-walled shells while ensuring functional dimensions .

Using Ribs Instead of Thickening: As mentioned, using rational rib layouts to enhance rigidity is the best practice to avoid blindly increasing wall thickness .

Targeted Cooling System Design: During mold design, prioritize the layout of cooling channels for areas with unavoidable thick sections to enhance cooling efficiency and minimize the temperature difference with thin-walled areas .

5.2.5 Design Specifications for Holes, Threads, and Bosses

Hole Design:

General Rule: The die-casting process can directly form holes, but its process characteristics must be considered. Through-holes are easier to form than blind holes .

Blind Hole Design: The depth of blind holes should not be excessive. Cores are prone to bending under the impact of high-pressure metal. The recommended blind hole depth is ≤ 4-5 times the hole diameter. For deeper blind holes, consider subsequent drilling .

Hole Spacing and Edge Distance: The distance from the hole edge to the part edge, and the distance between holes, should be at least 1.5 times the hole diameter to ensure strength and facilitate mold design .

Thread Design: Avoid directly die-casting fine threads. Thread roots are typical stress concentration points and prone to damage during demolding. The recommended approach is to die-cast a pilot hole and then tap the thread via machining for higher quality threads and simpler mold structure .

Boss Design: Bosses used for screw support or location should consider:

Root Reinforcement: The root requires a fillet transition and should be connected to the main wall with ribs for support .

Avoiding Hot Spots: The boss itself should also be cored out to prevent thick sections at the root .

Draft Angle: The outer wall of the boss must have a sufficient draft angle .

5.2.6 Draft Angles: The Minimum Requirement for Smooth Demolding and Surface Quality

Draft angle is the taper applied to the part wall along the demolding direction. It is crucial for ensuring the part ejects smoothly from the mold without surface damage .

Why Draft is Essential: Insufficient or no draft angle causes the part to shrink tightly onto the core after cooling, creating significant friction and vacuum force. This leads to ejection difficulties, ranging from surface scuffing to ejection pin marks, cracking, or even mold damage .

Recommended Values: The required draft angle depends on the material, surface texture depth, and cavity depth. The table below shows general minimum requirements :

Surface Type	Recommended Minimum Draft Angle
External Surface	0.25° (or 0.5%)
Internal Surface (Core)	0.5° (or 1.0%)
Deep Cavity or Textured Surface	Needs increasing, sometimes requiring 1.5° to over 2°

Note: Internal surfaces require a larger draft angle than external surfaces because the part shrinks onto the core after cooling .

LEDA Summary: These six golden rules are interconnected and complementary. At LEDA, we systematically apply these rules to your product design through DFM reviews. Our engineers use 3D modeling software and CAE simulation tools to thoroughly inspect your model and provide suggestions, ensuring the design is not only theoretically feasible but also achieves high quality, efficiency, and cost-effectiveness in production. We hope these detailed rules will be a valuable tool for designing high-performance, manufacturable die-cast parts .

5.3 The Core of Die-Casting Feasibility: Parting Line, Gating, and Venting System Design

The parting line, gating system, and venting system are the three core elements of die-casting mold design. They directly determine the molten metal filling behavior, the internal quality of the casting, and the complexity of the mold structure. Rational system design is the foundation for achieving efficient, stable, and economical production .

5.3.1 Parting Line Selection Principles: The Starting Point for Simplifying Molds and Reducing Costs

The parting line is the separation interface between the moving and fixed mold halves. Its selection is the primary decision in mold design, with a global impact on part demolding, mold machining, and cost control .

Core Selection Principles Include:

Ensuring Easy Part Demolding: The parting line must be at the part's largest contour. The selection should allow smooth part demolding after mold opening .

Guaranteeing Part Precision and Quality: For features requiring coaxiality, they should be on the same side of the parting line. Avoid placing the parting line on highly visible surfaces .

Simplifying Mold Structure and Machining: Avoid or minimize side cores. Prioritize planar parting surfaces over complex curved ones to reduce machining difficulty and cost .

Facilitating Venting: The parting line is a primary venting path. Its selection should facilitate the escape of gases from the mold cavity .

LEDA Practice: At LEDA, we use 3D software for demolding analysis to evaluate different parting line scenarios regarding demolding feasibility, slider count, and mold complexity .

5.3.2 Introduction to Gating System Design: Guiding Smooth Metal Filling

The gating system is the channel that guides molten metal from the injection chamber into the die cavity. Its design goal is to achieve smooth, orderly, and simultaneous cavity filling with minimal pressure and heat loss .

System Components and Design Points:

Ingate: The ingate is the final channel connecting the runner to the cavity. Its location should allow metal to fill the cavity smoothly and facilitate gas expulsion. The ingate area determines the filling velocity .

Runner: The runner distributes the molten metal. For multi-cavity molds, a naturally balanced runner layout is essential. The runner cross-section should be designed to minimize heat loss .

Overflow Wells: Overflow wells serve to trap cold or oxidized metal and aid in venting. They are strategically placed to ensure complete filling and improve weld line quality .

5.3.3 Venting and Overflow System Design: The Key to Solving Gas Porosity

The ability of gases inside the cavity to escape promptly is crucial to avoid defects like porosity, short shots, and burns .

System Design and Parameters:

Vents: Vents are the main passages for gas escape. They must be located at the last areas to fill. The vent depth is critical and must balance effective venting with preventing metal splash .

Overflow Wells: Overflow wells work in synergy with the vents. Their design should ensure they are not prematurely sealed off by metal, allowing vents to function effectively .

Auxiliary Venting Means: In special areas, other methods like venting pins or porous metal inserts can be used for micro-venting .

LEDA Summary: The parting line, gating, and venting systems are not isolated but closely linked. At LEDA, we use CAE mold flow analysis software to simulate the filling process, temperature field changes, and the distribution of gases and weld lines during the design stage. This allows us to predict and optimize these system designs accurately before mold manufacturing, avoiding production risks at the source and ensuring the feasibility and robustness of the mold solution .

5.4 Side Core-Pulling Mechanisms and Mold Complexity Assessment

Side core-pulling mechanisms are key designs for resolving the demolding of features like side holes and undercuts on die-cast parts. A reasonable side core-pulling design can significantly enhance mold reliability and production efficiency but also increases mold complexity and cost. This section will systematically analyze the working principles and optimization strategies of side core-pulling mechanisms and their impact on overall mold costs, providing practical guidance based on LEDA's engineering experience .

5.4.1 Working Principles of Sliders and Core-Pulling Mechanisms

The core function of a side core-pulling mechanism is to retract the lateral forming components (e.g., cores or sliders) first during mold opening via mechanical, hydraulic, or electric drives to avoid damaging the casting. They can be categorized into mechanical, hydraulic, and manual types based on the power source .

1. Angle Pin Mechanism (Mechanical Type)

This is the most common core-pulling method, utilizing the mold opening force for actuation. The inclination angle (α) of the angle pin typically ranges from **12° to 25°**. An excessively large angle increases bending risk, while too small an angle requires a longer mold opening stroke. During mold opening, the relative movement between the angle pin and the inclined hole in the slider converts the vertical opening force into the lateral movement force of the slider. The wedge angle of the lock block should be **2°–3° larger** than the angle pin's inclination to resist injection pressure. This mechanism is compact and reliable, but the core-pulling force and distance are limited by the strength of the angle pin .

LEDA Practice: For angled core-pulling requirements in automotive lens molds, LEDA employs a front-end design with a **3° insertion angle**, significantly improving mold life and stability .

2. Hydraulic/Pneumatic Mechanism

Driven by external hydraulic or pneumatic cylinders, this type offers **high core-pulling force and long stroke** (can reach several meters), and its actuation timing can be controlled independently. It is suitable for deep cavities or complex core-pulling scenarios but requires an additional power system, resulting in higher costs .

3. Dog-Leg Pin and Inclined Slider Mechanisms

Dog-Leg Pin Mechanism: Features a rectangular cross-section with high bending strength, allowing inclination angles up to **30°–40°**. It can achieve delayed or variable-speed core-pulling.

Inclined Slider Mechanism: Integrates lateral core-pulling with the ejection action. During ejection, the inclined slider moves along an inclined guide slot, performing lateral parting while ejecting the part. Suitable for parts with shallow undercuts; compact but requires high guiding precision .

4. Key Design Parameter Calculations

Core-Pulling Distance: Must be **2–3 mm greater** than the depth of the side hole to ensure the core fully disengages from the casting.

Core-Pulling Force: Depends on the casting's shrinkage force on the core, the core's surface area, and the alloy's shrinkage rate. LEDA uses CAE to simulate the distribution of shrinkage force, optimizing core surface roughness (**Ra ≤ 0.8 μm**) to reduce resistance .

Angle Pin Dimensions: The diameter (d) is determined by the bending force; the length (L) must satisfy L = L1 + L2, where L2 is the effective length required for core-pulling .

LEDA Case Study: For a motor housing die-casting mold, a **"angle pin + delayed slider"** combined mechanism was used. By precisely calculating the core-pulling sequence, interference between multiple cores was avoided, increasing yield by 15% .

5.4.2 Avoiding Unnecessary Side Core-Pulling through Design Optimization

Adding side core-pulling mechanisms significantly increases mold complexity and cost. During the DFM phase, LEDA prioritizes optimizing the product structure to reduce or simplify core-pulling requirements at the source .

1. Undercut Feature Optimization

Increase Draft Angles: Change vertical undercuts to **5°–10° drafts**, utilizing the alloy's elasticity (e.g., ZL102) for forced ejection.

"Volcano" Structure: For undercuts at the root of bosses like screw posts, design a conical transition to align the demolding direction with the main direction.

Part Decomposition: For complex multi-directional undercuts, split the part into assemblies, reducing mold complexity through subsequent assembly .

2. Slider Mechanism Simplification Strategy

Prefer Angle Pins: This is the lowest-cost solution when the core-pulling force is **≤5 kN** and the stroke is **≤60 mm**.

Avoid Motion Interference: Use UG motion simulation to check the timing between sliders and ejector pins, ensuring collision-free reset. LEDA experience shows that installing early return mechanisms can prevent 90% of interference risks .

3. Innovative Structure Applications

Lifter Mechanism: For internal undercuts, use lifters instead of sliders to simplify the mold structure.

Movable Inserts: For very low-volume production, use manual inserts to avoid complex core-pulling structures .

Optimization Strategy	Applicable Scenario	LEDA Practice Case & Effect
Increase Draft Angle	Flexible alloys (e.g., ZL102)	A connector housing: draft increased to 8°, slider eliminated, mold cost reduced by 20%.
Geometric Reshaping	Deep internal undercuts	Redesigned the bearing seat support structure, converting side core-pull to top/bottom core-pull, simplifying mold structure.
Material Selection & Combination	Parts allowing secondary assembly	Split a complex part into a combination of a die-casting and a stamping, reducing total cost by 15%.

LEDA DFM Principle: In the early project stages, we conduct demolding analysis using 3D software to propose modifications, striving to meet functional requirements with the simplest mold structure .

5.4.3 Impact Assessment of Mold Complexity on Cost and Cycle Time

Introducing side core-pulling mechanisms significantly affects mold cost, manufacturing cycle time, and production stability. Based on historical project data, LEDA summarizes the following assessment framework :

1. Quantitative Analysis of Cost and Cycle Time

Complexity Dimension	Specific Impact on Cost/Cycle Time	LEDA Quantitative Reference
Mold Manufacturing Cost	Increases costs for machining and heat treatment of specialized parts like sliders, guide rails, and lock blocks.	Each added angle pin slider increases mold base cost by 20%–30%; a hydraulic core-pull system can double the total cost.
Manufacturing Cycle Time	Sliders require high-precision processes like slow wire EDM, extending assembly and debugging time.	Mold manufacturing cycle time with sliders is 30%–50% longer than for molds without sliders.
Maintenance Cost	Moving parts wear and require regular replacement; poor lubrication can easily cause jamming.	Failure rate of hydraulic core-pulling mechanisms is 15%–20% higher than mechanical mechanisms, requiring regular seal checks.

2. Complexity Control Principles

Minimize Core-Pulling Distance: Design according to the formula **S = h + (2~3) mm** to avoid over-engineering.

Material and Heat Treatment: Slider body uses H13 steel (hardness ≥55 HRC); angle pins use T8A (hardness 55 HRC).

Standardized Modules: LEDA has established a standard parts library for sliders to reduce non-standard design and shorten processing cycles .

3. LEDA's Assessment Process

DFM Phase: Use CAE to simulate molten metal flow and shrinkage stress, predicting core-pulling resistance and optimizing mechanism parameters.

Prototype Testing: Use 3D-printed slider assemblies for motion verification to identify interference risks early.

Mass Production Monitoring: Embed sensors to monitor slider load, providing early warnings for jams and reducing downtime losses .

LEDA Summary: Side core-pulling mechanisms are essential for realizing complex functions in die-casting molds but must be balanced against the associated costs and risks. Our experience shows that **30% of unnecessary core-pulling mechanisms can be eliminated** through upfront optimization, while modular design reduces maintenance costs. For example, an automotive steering housing project increased mold life to over 500,000 shots through optimized slider layout .

5.5 Post-Processing and Surface Treatment Guide

Post-processing and surface treatment of die-cast parts are crucial steps for enhancing product appearance, functional reliability, and service life. Scientific post-processing can significantly improve surface properties, while appropriate surface treatment provides corrosion resistance, wear resistance, and aesthetic qualities. This section details deburring methods, machining allowance control, and design coordination points for surface treatment processes.

5.5.1 Deburring and Flash Removal Methods and Considerations

After die-casting, flash and burrs inevitably form on the parting line, ejector pin holes, and hole edges. These excess metal remnants not only affect assembly accuracy but can also cause system failures if they break off during use, necessitating thorough removal .

Primary Deburring Method Selection Strategy:

Depending on burr characteristics, part material, production volume, and cost considerations, different deburring processes can be chosen. The table below compares common methods :

Deburring Method	Technical Principle	Applicable Scenario	Advantages	Limitations
Vibratory Finishing	Grinding via relative motion between parts and abrasive media induced by vibration.	High-volume small castings, simple structure, small burrs.	High efficiency, low cost, batch processing.	Poor effectiveness for complex internal cavities, deep holes, cross-holes; may cause secondary scratches.
Magnetic Abrasive Finishing	Magnetic field drives magnetic abrasive to form a "brush" that polishes the surface.	Complex shapes, precision parts (e.g., gears, threaded holes).	Good accessibility, uniform treatment, preserves accuracy.	Higher equipment cost; limited effectiveness for non-magnetic materials.
Electrochemical Deburring (ECD)	Selective anodic dissolution of burrs via electrochemical reaction in an electrolyte.	Hard-to-reach areas like hidden cross-holes, complex internal cavities (e.g., hydraulic valve bodies).	Excellent accessibility for complex structures; no mechanical stress; high production speed (seconds to tens of seconds).	Electrolyte is corrosive; parts require post-cleaning to prevent rusting. May affect dimensions/gloss near the burr.
Thermal Energy Deburring (TED)	Instantaneous ignition of hydrogen-oxygen gas mixture in a sealed chamber burns off burrs with high heat.	Precision parts with complex internal cavities/cross-holes (e.g., automotive fuel injectors).	Effective on any material, especially complex structures difficult for other methods.	Very expensive equipment; creates oxide layer requiring removal; may cause part distortion/annealing.
High-Pressure Water Jet Deburring	Removes burrs using high-pressure water jet	Components requiring high cleanliness (e.g., automotive	Eco-friendly (no pollution), combines	High equipment cost; limited effectiveness on large, hard

Deburring Method	Technical Principle	Applicable Scenario	Advantages	Limitations
	(tens of MPa).	hydraulic control systems).	cleaning.	burrs.
Chemical Deburring	Selective corrosion of burrs using specific chemical solutions.	Micro-burrs on tiny precision parts (e.g., medical devices, connectors).	Can handle very small internal burrs; no mechanical stress.	Chemicals require proper disposal; potential slight corrosion of substrate.
Cryogenic Deburring	Flash-freezes burrs to brittle state using liquid nitrogen, then removes them with projectile impact.	Flash on flexible materials like rubber, plastic, and Zn/Mg alloys.	High efficiency, doesn't affect part properties.	Equipment cost ~$20k-30k; mainly for thin flash/small burrs on small parts.
Manual Deburring	Manual removal using files, sandpaper, scrapers, etc.	Low-volume prototypes, simple structures, or supplementing automated deburring.	Simple, flexible tools, low cost.	Low efficiency, inconsistent results, labor-intensive, poor for complex structures.

LEDA Practice Recommendation: At LEDA, we formulate a tiered deburring strategy based on the structural complexity, volume, and cost targets of the die-cast part. For example, for automotive transmission housings (high volume, complex structure), we prioritize combined processes like **high-speed robotic milling + magnetic abrasive finishing** to ensure precision and efficiency for key areas. For medical device chambers (many internal cross-holes, micro-burrs), we recommend **electrochemical or chemical deburring** to ensure complete removal without damaging the substrate. We ensure deburring consistency through process validation .

5.5.2 Machining Considerations: Why Machining Should Be Minimized and Allowance Controlled

Die-castings should aim for **Near Net Shape** to avoid or minimize subsequent machining. Machining not only adds cost but can also destroy the dense surface layer of the die-casting, exposing internal porosity and affecting performance .

Negative Impacts of Excessive Machining Allowance:

Destroys Surface Integrity: The die-casting surface is a dense, fine-grained layer formed by rapid cooling, offering the best mechanical properties and corrosion resistance. Machining removes this layer, potentially exposing internal defects like porosity, significantly reducing corrosion resistance and fatigue strength.

Introduces Residual Stress & Distortion: Cutting heat and mechanical stress can cause local temperature rise and plastic deformation. For instance, machining slender shafts with large allowances leads to long machining times and high heat, causing bending deformation due to thermal expansion constraints, resulting in geometric errors after cooling.

Increases Cost & Cycle Time: Larger allowance directly increases tool wear, man-hours, and energy consumption, reducing production efficiency.

DFM Measures to Minimize Machining Allowance:

Optimize Mold Design: Improve casting dimensional accuracy and geometric tolerances at the source using high-precision molds, rational gating systems, and process parameter optimization to reduce reliance on machining.

Precise Datum Definition: If machining is necessary, preset machining datums in the design and ensure alignment with the die-casting mold's datums to avoid error accumulation from datum conversion causing uneven allowance distribution.

Scientific Allowance Setting: Set the minimum necessary allowance based on part size, structure, and machining requirements. For example, a **0.3mm unilateral allowance** may suffice for reaming a hole in an Al alloy casting, much less than the traditional 0.5mm+.

Adopt Micro-Cutting Processes: For high-precision features (e.g., threaded holes), use precision boring or honing instead of standard drilling to control unilateral allowance within **0.1-0.2mm**.

LEDA Summary: In LEDA's DFM process, we use CAE flow analysis and tolerance simulation to accurately predict casting distortion trends and apply **reverse deformation compensation** during mold design, thereby minimizing machining allowance. Our goal is **zero machining for non-critical fit surfaces and micro-finishing for critical fit surfaces**.

5.5.3 Design Coordination Requirements for Common Surface Treatments (Spraying, Plating, Anodizing)

Surface treatments aim to enhance corrosion resistance, wear resistance, decoration, or specific functions. Different processes have specific requirements for part design .

1. Spraying (Painting, Powder Coating)

Design Points:

Avoid Sharp Edges: All external edges need a radius of **R0.5mm or more** to prevent thin coating at edges which reduces protection.

Avoid Paint Pooling: Avoid deep grooves and dead ends in the design to prevent paint accumulation and sagging. Incorporate drain/vent holes (**φ1.5-2mm**).

Reserve Hanging Points: Design hidden hanging points or process edges to ensure uniform coating without affecting appearance.

LEDA Material Selection: Aluminum and zinc alloys have the best paint adhesion. For outdoor or high-weather resistance parts, LEDA recommends **fluorocarbon coatings (PVDF)**, with a service life exceeding 20 years .

2. Plating (Nickel, Chrome, Zinc Plating)

Design Points:

Uniform Wall Thickness: Plating uniformity relates to electric field distribution. Avoid sharp section changes to prevent excessive plating at 凸起 (projections) and insufficient plating in recesses.

Radius Corners: All edges must have a radius of **R≥1.0mm** to avoid plating buildup at sharp points.

Avoid Deep Blind Holes: Blind holes with an aspect ratio >1:1 are difficult to plate uniformly; use through-holes or add process holes.

Substrate Finish: Plating amplifies substrate defects. Surfaces to be plated should ideally have **Ra≤0.8μm**.

LEDA Practice: For high-gloss chrome-plated zinc alloy faucet handles, LEDA requires mold surfaces to be **super-mirror polished (Ra ≤ 0.025 μ m)** and uses a multi-layer **Copper-Nickel-Chrome plating system** before plating to ensure Level 1+ standards .

3. Anodizing (Primarily for Aluminum Alloys)

Design Points:

Control Alloy Composition: High-silicon (Si>12%) die-casting alloys like ADC12 turn dark gray after anodizing with an uneven layer. For decorative anodizing, use high-purity aluminum or low-silicon alloys (e.g., specially treated A380).

Mind Contact Points: Contact points for racking will not develop an anodized layer. Design them on non-appearance surfaces or plan for subsequent removal.

Account for Fit: Anodizing adds 5-25μm thickness, increasing part dimensions. For shaft-hole fits, adjust tolerances precisely based on the coating thickness, typically using a base-hole system and appropriately enlarging the hole tolerance.

LEDA Innovative Solution: To address the darkening issue of ADC12 anodizing, LEDA developed a combined process of **"mechanical polishing + hard anodizing"**, adjusting electrolyte parameters to achieve a uniform, wear-resistant dark gray surface successfully applied to industrial drone gimbal brackets .

General Design Coordination Summary:

Pre-treatment Compatibility: All surface treatments require pre-treatment (degreasing, derusting). Part design should facilitate the flow and drainage of cleaning and chemical solutions; avoid enclosed cavities.

Texture and Appearance: If the part has a texture (grain) before surface treatment, inform the supplier, as the texture can trap chemicals, potentially requiring process adjustments.

Environment and Safety: LEDA prioritizes **trivalent chromium passivation** over toxic hexavalent chromium processes and uses water-based coatings, complying with RoHS and REACH directives .

5.6 Analysis of Common Die Casting Defects and DFM Prevention Strategies

Defects in die-castings have complex causes, but over 80% can be avoided at the source through optimized Design for Manufacturability (DFM). The table below summarizes the core characteristics and design prevention strategies for three common defect types, providing a quick overview to help build a comprehensive understanding .

Defect Type	Core Characteristics	DFM Prevention Golden Rules
Gas Porosity & Flow Lines	Gas Porosity: Round or oval pores with smooth inner walls, often located inside or near the surface of the casting. Flow Lines: Irregular, meandering streaks or depressions on the casting surface.	Optimize Gating & Overflow System: Use stepped runners and large overflow wells to guide smooth metal filling and orderly venting. Strengthen Venting System: Install vents with appropriate depth in the last areas to fill.
Shrinkage Porosity/Cavities	Shrinkage Cavity: Irregularly shaped cavities with rough, dull inner walls, mostly located in hot spots like thick sections, ribs, and bosses. Shrinkage Porosity: Dense, dispersed micro-porosity.	Uniform Wall Thickness: Strictly control wall thickness uniformity, avoiding thick sections. Use "spot cooling" and "volcano" coring designs at thick areas to eliminate hot spots fundamentally.
Sticking & Soldering	Signs of material tearing on the casting surface, or presence of drag marks; severe cases lead to incomplete castings or distortion.	Strengthen Mold & Lubrication: Optimize draft angles; apply high-standard polishing in key areas; precisely control die release agent spraying. Optimize Ejection System: Ensure balanced ejection to avoid localized stress concentration.

Next, we will explore the specific causes and systematic prevention strategies for each defect in detail.

5.6.1 Causes and Prevention of Gas Porosity and Flow Lines

Gas porosity and flow lines are among the most common defects in die casting. Their essence is the entrapment of air, gas from the cavity, or volatilized die release agent during metal filling, which fails to escape in time, resulting in internal pores or surface quality issues .

Cause Analysis

Improper Gating System Design: Excessive gate velocity causing turbulent filling easily entraps air. Severe changes in runner cross-section or improperly located/ sized vents (too deep causing metal leakage, too shallow causing poor venting) prevent gas from escaping smoothly, leading to porosity .

Poor Mold Venting System: Insufficient number or poor placement of vents is the main cause. Deep cavities, slider joints, and the last metal meeting points are venting challenges, prone to flow lines or internal porosity due to trapped air .

Mismatched Process Parameters: Excessive injection speed, low die temperature, or excessive/improperly volatilized die release agent can introduce large amounts of gas, worsening porosity and flow lines .

Air Entrapment & Cold Shuts: Two metal flows meeting asynchronously or impact with cores creating vortices can trap air, forming not only porosity but also flow lines at the junction .

DFM Prevention & Optimization Strategies

Optimize Gating System: This is fundamental. Use fan gates and progressive tapered runners to ensure stable filling and avoid turbulence. LEDA practice shows that using CAE mold flow analysis to accurately predict flow patterns and last-to-fill areas is the most effective method for guiding rational gate and vent placement .

Strengthen Venting System: Provide sufficient and appropriately sized vents at the last areas to fill (typically behind overflow wells). Vent depth is critical: generally 0.05-0.15mm for aluminum alloys, slightly deeper for zinc alloys. For areas difficult to vent, use venting inserts or pins .

Refine Process Parameters: Adopt a "slow-fast-slow" multi-stage injection process. Start with low speed to push air out of the runners before the metal passes the gates, then switch to high speed to fill the cavity. Precisely control die temperature and die release agent concentration/spray volume to ensure complete volatilization .

5.6.2 Causes and Prevention of Shrinkage Porosity and Cavities

Shrinkage porosity and cavities are macroscopic or microscopic voids formed in the last solidifying areas of a casting due to liquid and solidification contraction that cannot be effectively compensated by adjacent liquid metal .

Cause Analysis

Structural Hot Spots: This is a core design issue. Non-uniform wall thickness and localized thick sections (e.g., boss bases, rib-to-wall junctions) cool slower. When these areas solidify last, they cannot be fed, leading to shrinkage porosity or cavities .

Insufficient Feeding Pressure: Low intensification pressure or short intensification time cause the plunger to stop before the casting solidifies, preventing effective feeding from the biscuit .

Premature Gate Solidification: Gates with too small cross-sectional area solidify before the casting, cutting off the feeding path .

Improper Fillet Design: While fillets avoid stress concentration, excessively large fillet radii can create local thick sections, forming new hot spots and causing shrinkage .

DFM Prevention & Optimization Strategies

Pursue Uniform Wall Thickness & Structural Coring: This is the most important principle. Use topology optimization to avoid sudden wall thickness changes. For necessary thick features like mounting bosses or screw posts, use coring to change them from solid to uniform thin-walled shells, fundamentally eliminating hot spots .

Optimize Gates & Cooling System: Place gates near thick sections and ensure sufficient thickness and longer solidification time to establish effective feeding channels. Cooling channel layout should focus on hot spot areas to achieve directional solidification, starting from the area farthest from the gate and ending at the gate .

Rational Design of Fillets and Ribs: Internal fillet radius is typically 0.5-1 times the wall thickness. Avoid excessively large radii creating thick sections. Rib thickness should be controlled at 50%-60% of the main wall thickness to prevent shrinkage at the root .

5.6.3 Causes and Prevention of Sticking and Soldering

Sticking/soldering refers to microscopic welding or mechanical interlocking between the casting surface and the mold cavity, causing tearing of the casting surface during ejection, leading to surface damage or even incompleteness .

Cause Analysis

Mold Surface Issues: Scratches, pits, or poor polish on the cavity surface cause mechanical interlocking. Insufficient mold steel hardness or improper heat treatment leads to surface softening under high temperature/pressure, increasing metal adhesion .

Poor Ejection System Design: Insufficient draft angles (especially for deep cavities or textured surfaces) or unreasonable ejection system layout (ejector pins, ejector plates) cause unbalanced ejection, resulting in high local stress and dragging .

Alloy-Mold Material Affinity: Aluminum alloys, especially those with low iron content, have high affinity with mold steel (e.g., H13), making them prone to soldering .

Improper Process Operation: Excessive die temperature, insufficient or ineffective die release agent, or excessive cooling time leading to high shrinkage force and tight wrapping .

DFM Prevention & Optimization Strategies

Ensure Adequate Draft Angles: This is key. Use the maximum value within the allowable range. For external surfaces, typically not less than 0.25° (or 0.5%); for internal surfaces (cores), typically not less than 0.5° (or 1.0%). For textured surfaces, additional draft must be added based on texture depth .

Mold Surface Strengthening & Optimization: Apply high-standard polishing (e.g., mirror polish) to cavities and cores. Use surface treatments like nitriding in prone areas to improve hardness, wear resistance, and anti-sticking properties. Select high-quality mold steel. All wall junctions must have fillet transitions, which not only avoid stress concentration but also significantly improve metal flow and mold life .

Optimize Ejection System: Ensure the ejector pin layout is balanced and powerful for uniform stress distribution during ejection, avoiding localized stress concentration. For deep-cavity parts, consider auxiliary ejection methods like air ejection .

LEDA Summary: The most economical and effective method to solve die-casting defects is prevention during the product design stage, rather than correction during mass production. At LEDA, we integrate CAE mold flow analysis, tolerance simulation, and DFM review as standard procedures. Using digital tools to accurately predict potential defects early in the design phase, combined with our extensive material database and engineering experience, we optimize product structure, gating/venting systems, and process plans to ensure manufacturability and stability from the source, laying a solid foundation for high-quality mass production .

Chapter 6: Sheet Metal Stamping and Bending Design Guide

Sheet metal stamping and bending are among the most widely used forming technologies in modern manufacturing. This process involves the cold working of metal sheets with uniform thickness (typically below 6mm) through techniques like shearing, punching, bending, stretching, and forming to efficiently produce metal parts with complex structures and excellent strength. This combined "subtractive" and "forming" process is particularly suited for high-volume, high-consistency production needs and holds an irreplaceable position in fields such as enclosures and cabinets, automotive components, electronic equipment, and new energy applications .

The design quality of sheet metal parts directly determines the complexity of the mold structure, production stability, and the final product's quality and cost. Excellent sheet metal design must not only meet the product's functional, strength, and appearance requirements but also fully consider its manufacturability. A seemingly simple bend angle or the layout of a hole directly impacts mold life, production efficiency, and product yield. By following scientific design principles, you can :

Avoid Common Defects: For example, effectively prevent cracking at bends through reasonable minimum bend radius design and the application of relief notches.

Ensure Dimensional Accuracy: Control the safe distance between locating holes and bend edges, and ensure part accuracy and assembly reliability by understanding and compensating for springback behavior.

Simplify Mold Structure and Reduce Costs: Optimize the part profile to avoid sharp corners and complex protrusions, and use fillet transitions to reduce mold wear, extend its life, and improve material utilization.

Enhance Production Efficiency and Quality Stability: Reasonable design of process reliefs and locating dimples can streamline the production process, reduce adjustment time, and facilitate automation.

This chapter will serve as your practical guide, systematically analyzing how to design sheet metal stamping and bending parts that are easy to manufacture. We will start by understanding sheet material properties and selection, the foundation of all design decisions. Subsequently, we will delve into the detailed design rules for blanking (punching, blanking) and bending, which form the core of this chapter. We will also examine design for assembly, surface treatment selection, and cost optimization through design .

Whether you are designing a server chassis requiring good electromagnetic shielding, a communication equipment structural part requiring high precision, or an EV charger enclosure needing to withstand harsh outdoor environments, the principles outlined in this chapter are universally applicable. Mastering them will enable you to communicate efficiently and accurately with mold engineers and manufacturers (such as LEDA), working together to transform your ideas into high-quality, low-cost sheet metal products .

LEDA's Experiential Perspective: Based on our extensive experience in sheet metal processing, this chapter will emphasize the collaborative optimization of design, material, and process. We strongly advise involving your manufacturing partner early in the project. Through DFM (Design for Manufacturability) analysis, conduct preliminary simulation and verification of material selection, bending sequence, and tolerance fits to avoid production risks at the source and prevent costly design changes later. For example, accurately calculating springback and compensating for it in the design can significantly improve bend angle accuracy and reduce debugging time .

6.1 Sheet Material Selection and Thickness: Balancing Performance, Processability, and Cost

Material selection for sheet metal parts is the cornerstone connecting product design and manufacturing processes, directly determining the part's functional realization, production cost, and quality reliability. Rational selection requires a fine balance between the material's mechanical properties, process adaptability, and overall cost .

6.1.1 Characteristics and Application Scenarios of Common Sheet Materials (Cold Rolled Steel, Galvanized Steel, Aluminum Plate, Stainless Steel)

The four mainstream sheet metal materials each have distinct characteristics and optimal application scenarios. The table below provides a quick reference for selection .

Material Type	Core Characteristics	Advantages	Limitations	Typical Application Scenarios
Cold Rolled Steel (SPCC)	Low cost, easy to form, moderate strength, but uncoated surface is highly prone to oxidation and rust.	Good stamping and bending performance, cost-effective, suitable for surface coating (e.g., powder coating, baking paint).	Poor corrosion resistance, requires surface treatment (e.g., painting, plating) for use.	Internal structural parts for enclosures/cabinets, internal frames for home appliances – structures with low appearance requirements.
Galvanized Steel (SECC/SGCC)	Cold-rolled steel base with zinc coating for corrosion resistance. SECC is electrogalvanized, SGCC is hot-dip galvanized.	SECC has a smooth, flat surface with good fingerprint resistance; SGCC has a thicker coating for stronger corrosion resistance and longer life.	SECC has poorer weldability; SGCC is harder, less ductile than SECC, not suitable for deep drawing designs.	SECC: Computer cases, appliance panels. SGCC: Building components, outdoor cabinets.
Aluminum Plate (e.g., 5052, 6061)	Low density (lightweight), corrosion-resistant, good electrical/thermal conductivity, but generally lower strength and hardness than steel.	Excellent strength-to-weight ratio, good corrosion resistance and formability; some grades can be heat-treated for strengthening.	Higher cost compared to carbon steel, lower hardness makes it prone to scratching, requires specific welding techniques.	Aerospace components, consumer electronics housings (e.g., laptops), heat sinks, lightweight vehicle structures.
Stainless Steel (SUS304/SUS301)	High strength and hardness, excellent corrosion and heat resistance.	SUS304 has good overall performance, widely used; SUS301 can achieve higher strength and elasticity	High material cost, difficult to process (high tool wear), significant	SUS304: Medical devices, food equipment, kitchenware. SUS301: Spring components, EMI shielding

Material Type	Core Characteristics	Advantages	Limitations	Typical Application Scenarios
		through cold working.	springback affects bending accuracy.	covers.

LEDA Experience: At LEDA, we recommend clarifying the product's use environment, life cycle, and key performance indicators early in the project. For example, for frequently inserted/removed data center server hard drive trays, even when cost-sensitive, SECC is recommended over SPCC because its inherent rust resistance significantly enhances long-term product reliability .

6.1.2 Impact of Key Process Properties (Plasticity, Surface Quality, Tolerances) on Design

Material selection directly influences the feasibility and quality of subsequent manufacturing processes. Three key process properties require focus :

Plasticity (Formability)

Plasticity determines the material's ability to undergo permanent deformation without cracking, directly impacting processes like bending and stretching.

Bending Performance: Sheets requiring bending should have sufficient plasticity and a low yield limit. Materials with good plasticity like low-carbon steel, brass, and rust-proof aluminum are easy to bend with little springback. Brittle materials like high-carbon steel and hard aluminum require a larger relative bending radius (R/t) to avoid cracking.

Stretching Performance: Deep drawing is a challenging process in sheet metal, requiring excellent material plasticity. Materials with a low yield ratio ($\sigma s/\sigma b$), like pure aluminum sheet and 08Al steel, have good stamping performance. Designs should avoid deep drawing or complex forming for materials with poor plasticity.

Surface Quality

The surface quality of the sheet determines the finished product's appearance and the effect of certain surface treatments.

Base Material: SECC and BA sheets come with excellent surfaces and are often used as appearance parts without needing heavy treatment. SPCC oxidizes easily and is typically used as a base for painting.

Processing Impact: Laser-cut edges have an oxide layer; CNC punching can create burrs around holes, requiring post-processing. For products requiring high gloss or a mirror finish, the initial surface quality of the material and the choice of cutting process are crucial.

Tolerances and Stiffness

Dimensional Tolerances: Cold-rolled sheets typically have smaller thickness tolerances and better flatness than hot-rolled sheets. This is critical for precision components and stable assembly.

Stiffness (Rigidity) Misconception: A common design mistake is switching from low-carbon steel to high-carbon steel/stainless steel, or from standard aluminum to hard aluminum, to increase part stiffness. However, for the same base material, heat treatment and alloying can significantly increase strength and hardness but change stiffness very little. The effective way to increase stiffness is to change the part's shape design (e.g., adding beads, flanges) or change the material (e.g., using steel instead of aluminum), as stiffness primarily relates to the material's elastic modulus and geometric structure .

6.1.3 Selection Decision Matrix: Striking a Balance between Performance, Processability, and Cost

To make a scientific selection decision, performance, processability, and cost must be weighed together. The table below provides a multi-dimensional decision matrix .

Consideration Dimension	Priority Option	Decision Rationale & Trade-off Strategy
Cost Priority	SPCC (requires surface treatment)	SPCC is the lowest-cost option when basic functionality is met. However, necessary painting or plating costs must be included. It is the first choice for internally structural parts with low corrosion requirements and extremely limited budgets.
Corrosion Resistance Requirement	Outdoor/Humid: SGCC, Aluminum, Stainless Steel Indoor/General: SECC	Choose based on environmental severity. Stainless steel is the most expensive, suitable for high-demand fields like medical and food; SGCC and aluminum are common choices for outdoor cabinets and curtain walls.
Lightweight Requirement	Aluminum Plate, Magnesium Alloy (not focus here)	Aluminum's density is about 1/3 that of steel, making it the primary choice for lightweighting. Its strength and stiffness must be evaluated for suitability, and its higher material cost accepted.
High Strength/Hardness Requirement	Stainless Steel, High-Strength Steel Plate	Ideal for load-bearing structures or wear-resistant parts. However, challenges like difficult processing, fast tool wear, and significant springback must be addressed.
Good Conductivity Requirement	Aluminum Plate, Copper Plate (T2 Copper, H62 Brass)	Preferred for busbars, electrical contacts, etc. Copper has excellent conductivity but is expensive and has low strength; H62 brass is often used as a substitute to balance cost and performance where conductivity requirements are met.
Appearance and Decorative Requirement	SECC, Aluminum (Anodized), Stainless Steel (Brushed/Mirror)	SECC has a uniform coating and good painting base; aluminum can be anodized for rich colors; stainless steel can be directly brushed or mirrored for a high-end look.

LEDA's Selection Process Recommendation:

Define Requirements: First, define the product's core needs: Is it cost-sensitive, weight-sensitive, or reliability-sensitive?

Preliminary Screening: Based on the primary need, use the table above to identify 1-2 candidate materials.

Process Feasibility Verification: Collaborate with manufacturing engineers (e.g., the LEDA team) to evaluate the candidate materials' bend radius, weldability, and surface treatment compatibility. For example, for an aluminum alloy housing with a narrow flange bend, verify if its minimum bend radius is feasible.

Cost Calculation: Perform a total cost calculation, including material cost, processing fees (e.g., laser cutting, bending), surface treatment cost, and potential mold costs.

LEDA Summary: Material selection is not about pursuing the optimal single indicator but finding the most economical and manufacturable balance point that meets the core function. At LEDA, we use material databases and DFM analysis software to provide clients with quantitative comparisons based on historical project data, helping you avoid selection pitfalls and make the best decision .

6.2 Blanking and Hole Feature Design: Ensuring Quality and Mold Life

Blanking is one of the most fundamental and critical processes in sheet metal fabrication. Its design quality directly determines mold life, production costs, and part accuracy. Scientific design of blanking and hole features can significantly enhance production stability and economy while meeting functional requirements .

6.1.4 Profile Design: Avoiding Sharp Corners and Complex Protrusions, Using Fillet Transitions

The profile design of a blanked part is the foundation of the mold structure and should follow the principles of "simplicity, robustness, and ease of manufacturing" .

Avoid Sharp Corners, Use Fillet Transitions: All sharp corners on internal and external contours are stress concentration points that significantly accelerate mold edge wear and chipping during stamping. All junctions between straight lines or curves must be designed with fillet transitions. The recommended minimum fillet radius is **R ≥ 0.5t** (where t is material thickness), and generally not less than 0.8mm. This not only protects the mold but also allows for smoother material flow and improves edge quality .

Avoid Complex Protrusions and Narrow Slots: The design of excessively long and narrow protrusions and slots should be avoided. The corresponding parts of the mold (punches or dies) for such features have low strength and are prone to damage. Generally, the width B of a protrusion or slot should be ≥ **1.5t**. If unavoidable due to functional constraints, their width should be maximized to enhance the strength of the mold edge .

Strive for Simple, Symmetrical Shapes: Simple, symmetrical profiles facilitate efficient material nesting, improve material utilization, and reduce scrap rates. Furthermore, symmetrical molds experience more balanced forces, effectively reducing offset loads, extending mold life, and helping to ensure part accuracy .

LEDA Practice Suggestion: At LEDA, we utilize CAD software for mold cavity pressure analysis to predict the force distribution on the mold during the blanking process already at the product design stage. For unavoidable sharp corners or protrusions, we advise customers on optimizations or provide early warnings about mold risks, enabling collaborative development of more robust mold solutions .

6.2.1 Hole Design: Safety Standards for Minimum Hole Size, Edge Distance, and Spacing

Holes are the most common features on sheet metal parts. Their design must strictly adhere to safety standards to ensure manufacturability and part quality .

Minimum Hole Diameter: The punching size is strictly limited by the strength and rigidity of the punch. Excessively small holes require thinner punches, which are prone to breakage due to poor stability. The minimum hole diameter is closely related to the material type and thickness. The table below provides a reference for the minimum punch diameter for common materials :

Material Type	Minimum Round Hole Diameter (d)	Minimum Square Hole Side Length or Short Side Width of Rectangular Hole
High Carbon Steel	d ≥ 1.3t	≥ 1.0t
Low Carbon Steel, Brass	d ≥ 1.0t	≥ 0.7t
Aluminum	d ≥ 0.8t	≥ 0.5t

Note: t is material thickness. Generally, punched hole sizes are not recommended to be smaller than 0.3mm; otherwise, alternative processing methods like laser cutting should be considered .

Edge Distance and Hole Spacing: This refers to the distance from the hole edge to the part's outer profile edge, and the distance from the edge of one hole to the edge of another. If this distance is too small, it can cause material distortion or tearing during stamping. Safety standards require :

Edge Distance: When the hole edge is parallel to the part edge, the minimum distance should be ≥ **1.5t**; when not parallel, the minimum distance should be ≥ **t**.

Hole Spacing: The minimum distance between two holes is recommended to be ≥ **t** to ensure sufficient strength in that area .

Hole Locations on Bent/Stretched Parts: When punching holes on bent or drawn parts, sufficient distance must be maintained between the hole wall and the part's straight wall to prevent uneven force on the core and hole deformation during punching. If high part accuracy is required, a process sequence of **forming first, then punching** should be adopted.

6.1.5 Relief Notches and Process Reliefs: Preventing Stress Concentration and Corner Tearing

When a bend line extends to the sheet boundary or is adjacent to another bend, the material undergoes significant internal stress during deformation,Prone to tearing (very likely to tear) starting from sharp corners or weak points. Relief notches and process reliefs are key designs for releasing stress and guiding crack propagation .

Relief Notches: Primarily used to prevent cracks from propagating into non-deformed areas during bending. Common forms include rectangular and circular slots.

Size Specification: The width of the relief notch should typically be **> t** (material thickness), and its length/depth should be **> 1.5t**. The end of the slot should be rounded to avoid new stress concentration points .

Application Scenario: When a bent flange extends to an internal edge of the blank, a relief notch or process hole must be designed in advance at the end of the bend line .

Process Holes/Reliefs: When local bending is required from within the sheet, or when bending is adjacent to the sheet edge, process reliefs must be pre-designed to provide space for material flow.

Partial Bending: When only part of an edge is bent, process reliefs should be designed at both ends of the bend line, with a recommended width ≥ **1.5t**, to prevent deformation and cracking .

U-shaped Bends: Design process locating holes as benchmarks for multiple bends to reduce cumulative errors and ensure product quality. These holes are particularly important for asymmetrical U-shaped bends .

Design Principles:

Anticipate Deformation Zones: Use CAE software at the design stage to simulate material flow and accurately predict high-stress areas.

Proactive Guidance, Not Reactive Fixes: Treat relief notches and process cuts as preventive design features, not as afterthoughts.

Precise Sizing: Strictly design the relief size according to the material thickness ratio; too small is ineffective, too large weakens the part.

LEDA Summary: Blanking and hole feature design combine rationality and experience. At LEDA, we incorporate DFM analysis as a standard procedure, conducting a comprehensive manufacturability review of your sheet metal model, including minimum spacing checks, sharp corner identification, and bend interference analysis, and provide a detailed optimization report. Through early collaborative design with you, we can avoid production risks at the source, achieving the optimal balance between quality, efficiency, and cost .

6.3 Bending and Forming Design: Core of Accuracy and Manufacturability

Sheet metal bending is a key process that determines part accuracy, strength, and assemblability. Scientific design can significantly improve product quality and production efficiency, while inadequate consideration can lead to cracking, springback, and dimensional errors. This section systematically analyzes the key points of bending design.

6.1.6 Bending Fundamentals: Minimum Bend Radius, Springback Control, and Bend Allowance

Minimum Bend Radius

The minimum bend radius is the smallest inside radius to which a material can be bent without failure. It is primarily determined by material type, condition, and thickness .

Design Guideline: The inner bend radius of the part should not be less than the material's allowed minimum. If the design radius is too small, excessive stretching of the outer fibers will cause cracking or significant weakening .

Reference Data: For low-carbon steel like SPCC, the minimum bend radius can be 0.4 times the material thickness; for stainless steel like 304, it increases to 0.8-1 times; for brass H62, it's about 0.8 times. Aluminum sheet (e.g., 5052) typically requires 0.5-1 times the material thickness .

Springback Control

Springback is the elastic recovery of the material after the bending moment is removed, causing changes in the bend angle and radius. It is the primary factor affecting bending accuracy .

Influencing Factors: Higher material yield strength, lower elastic modulus, and larger relative bending radius (R/t) lead to greater springback .

Control Strategies:

Die Compensation: During mold design, compensate for the predicted springback angle Δα by reducing the die angle accordingly .

Overbending: Adjust the machine program so the upper die stops slightly below the nominal bottom dead center, using slight over-bending to counteract springback .

Bottoming: Apply sufficient pressure at the end of the bending stroke to induce plastic deformation in the inner fibers of the bend area, reducing elastic recovery .

Design Stiffening Ribs: Pressing ribs in the bend area can effectively increase local stiffness and suppress springback .

Bend Allowance (Minimum Flange Length)

The bend allowance refers to the minimum straight height required for the bent flange to ensure a stable and smooth bending process .

Basic Requirement: The straight height L of the bent edge should not be too small; typically, **$L \geq 2t + R$** (t is material thickness, R is inside radius) is required. If the height is less than this value, the flange may not be reliably clamped by the tooling, leading to unstable forming or inaccurate dimensions .

Special Case: If a smaller straight height is structurally necessary (e.g., $L \leq 2t$), consider pre-machining a shallow groove in the deformation zone before bending, but this sacrifices some strength .

6.1.7 Key Structural Design: Minimum Straight Height, Edge Relief, and Process Reliefs for Local Bending

Minimum Straight Height

As mentioned, ensuring sufficient straight height is a basic requirement for bend manufacturability. For flanges with tapered edges, the minimum straight height needs to be appropriately increased, typically requiring **$h \geq (2\sim4)t$**, and greater than 3mm .

Edge Relief (Bend Relief)

When the bend line is not parallel to or intersects the sheet edge, a suitable relief (or "clearance cut") should be designed at the edge to avoid stress concentration and tearing at the sharp corner during bending. The relief size should ensure **S ≥ R** .

Process Reliefs for Local Bending

Process reliefs or relief notches must be designed when local bending is required in the middle of a plate, or when the bend line extends to the sheet boundary .

Function: Release stress, guide material flow, and prevent cracks from propagating into non-deformed areas.

Design Specifications:

Slot Width: K ≥ t (material thickness)

Slot Depth/Length: L ≥ t + R + K/2 (R is the inside radius)

The end of the relief should be rounded to avoid new stress concentrations.

6.1.8 Relative Position Design of Holes, Pressed-in Nuts, and other Features Relative to Bends

Distance from Hole to Bend Line

If a hole is too close to the bend zone, the material stretching during bending will deform the hole.

Design Principle: The distance from the hole edge to the inner surface of the bend must not be too small.

Safe Distance: Empirically, this distance should be ≥ **1.5t + R**. For critical mounting holes, it should be ≥ **2t + R**, or consider secondary processing (bend first, then punch) to ensure hole position accuracy .

Protruding Features like Pressed-in Nuts

Protruding features such as pressed-in nuts and screw studs must be located away from the bend deformation zone.

Safe Distance: The distance from the edge of these features to the inner surface of the bend typically needs to be ≥ **3t + R**, adjusted specifically based on the protrusion height. Insufficient distance will cause bending interference, crushing of the feature, or sheet distortion .

6.1.9 Enhancing Stiffness and Accuracy: Locating Holes, Stiffening Ribs, and Tolerance Labeling Key Points

Locating Holes

For parts requiring multiple bends or high-precision positioning, process locating holes must be designed.

Function: Provide a unified datum for all subsequent bending and processing operations, greatly reducing cumulative errors and ensuring product quality.

Design: Typically, two round holes not collinear are used as primary and secondary datums. The hole diameter is generally φ3.0 ~ φ5.0mm. Locating holes should be away from bend and weld zones to ensure their own accuracy stability .

Stiffening Ribs

Pressing stiffening ribs on sheet metal planes is an effective method to increase rigidity, suppress resonance, and reduce distortion of appearance surfaces, while also helping to control springback.

Design Points:

Rib Width: Generally 0.3 ~ 0.5 times the material thickness.

Rib Depth: Should not exceed 3 ~ 5 times the material thickness; excessive depth may cause visible marks on the outer surface.

The ends of the ribs should be designed with a gradual transition to avoid stress concentration .

Tolerance Labeling Key Points

Reasonable tolerance labeling is key to ensuring functionality and controlling costs.

Basic Principle: Follow GD&T (Geometric Dimensioning and Tolerancing) principles. Prioritize functional tolerances for key mounting and fit dimensions. Apply general tolerances to non-critical external dimensions.

Bend Angle Tolerance: Typically ±1°. For high-precision requirements, ±0.5° can be specified, but this increases adjustment costs. The angle measurement standard must be clearly specified on the drawing.

Linear Dimension Tolerance: Differentiate between non-bent dimensions and bent dimensions. For critical hole locations and edge-to-edge distances, clearly specify their tolerance requirements and measurement datums .

LEDA Suggestion: Clearly specify requirements such as burr direction, bend direction (up or down) in the drawing notes. For subsequent treatments like plating and painting that affect dimensions (usually causing thickening), this should be considered when labeling tolerances .

LEDA Summary: Bending design is a combination of art and science. At LEDA, we use the sheet metal modules of 3D CAD software for design and employ DFM analysis to simulate the bending process in advance, calculate blank sizes, and warn of potential defects. We strongly recommend communicating with our engineering team early in the project to collaboratively optimize the design, ensuring your sheet metal part achieves the best balance between function, manufacturability, and cost .

6.4 Connection and Assembly Design: Achieving Robust and Efficient Combinations

The design of connections and assemblies for sheet metal parts is a critical link in ensuring product structural integrity, functional reliability, and production efficiency. Scientific design can significantly reduce assembly difficulty, minimize component count, and control tolerance stack-up, thereby achieving robust manufacturing and high-quality product output .

6.1.10 Welding Design: Weld Layout, Groove Design, and Common Welding Process Selection

Welding is the most common permanent joining method for sheet metal structures. Excellent design must balance joint strength, process feasibility, and cost control .

1. Golden Rules for Weld Layout

Symmetrical Arrangement, Avoid Concentration: Welds should be laid out as symmetrically as possible to balance welding stress and distortion, preventing part warping. Absolutely avoid welds converging or overlapping at a single point, as this creates significant stress concentration points prone to structural cracking .

Accessibility for Welding and Inspection: Weld locations should allow easy access for the welding torch and subsequent quality inspection. Sufficient operating space must be reserved, avoiding designs in enclosed corners or areas inaccessible to line of sight .

Avoid Critical Areas and Appearance Surfaces: Welds should be placed on non-primary load paths and non-appearance surfaces. For exposed surfaces, continuous welds can be used to improve aesthetics, but welding distortion must be controlled .

2. Groove Design

For thicker plates (typically > 3mm), edge preparation (grooving) is necessary on the edges to be welded to ensure penetration and weld strength.

Groove Types: Common types include I-butt (no groove, for thin sheets), V-groove, Y-groove, U-groove, etc. Selection depends on plate thickness, welding method, and accessibility .

Design Points: The groove angle and root gap require precise calculation to balance penetration depth and filler metal volume (affecting efficiency and cost) .

3. Common Welding Process Selection

The most appropriate welding process must be selected based on material type, plate thickness combination, production volume, and quality requirements. The table below provides a clear guide based on :

Welding Process	Applicable Thickness (t, mm) & Materials	Core Characteristics & Application Scenarios	Design Considerations
Spot Welding (Resistance)	t ≤ 2.5mm (cold-rolled steel, stainless steel, and t1 - t2 ≤ 0.5mm)	High efficiency, high automation, suitable for thin sheet lap joints. Best processability, recommended first.	Spot weld flange width must be ≥ 15mm to provide operating space for the electrode .
Tungsten Inert Gas (TIG)	t ≤ 2.5mm (cold-rolled steel, stainless steel); t ≤ 3.0mm (stainless steel, thickness difference ≤ 2.0mm)	High weld quality, beautiful appearance, suitable for thin sheets and stainless steel requiring good appearance.	Suitable for long welds (l ≥ 150mm) without waterproofing requirements or full penetration welds .
Metal Active Gas (MAG)/CO_2	1.0mm ≤ t ≤ 2.5mm (Steel plate)	High efficiency, lower cost, suitable for semi-automatic/automatic welding of medium-thickness steel plates.	Suitable for long/short welds with strength requirements, but relatively high spatter .
Shielded Metal Arc Welding (SMAW)	t > 2.5mm (Steel plate)	Simple equipment, flexible, suitable for thick structural parts, repair, and on-site construction.	High heat input, significant distortion and residual stress, requires careful control .

4. Structural Reinforcement and Process Optimization

Stiffening Rib Design: For door panels wider than 500mm or parts with large open areas (≥20% of panel area), Ω-shaped stiffening ribs must be added to increase stiffness. The rib length should ideally extend to contact the bent edges at both ends of the part .

Butt Weld Optimization: For butt welds required due to material size limitations, using a backing plate with spot welding is recommended instead of traditional intermittent or full penetration butt welds. This enhances product grade and production efficiency .

6.1.11 Riveting and Threaded Installation: Press-in, Riveting Standard Part Selection and Design Key Points

Riveting and threaded installations are essential methods for detachable connections, suitable for joining different materials or areas requiring frequent maintenance .

1. Press-in Nuts/Studs

Applicable Thickness: Press-in standard parts (e.g., PEM nuts SO, studs BSO) have strict requirements on sheet thickness. Designers must consult standard part manuals to ensure the used thickness is within the recommended range .

Pilot Hole Design: The pilot hole diameter must precisely match the standard part specification. Too small a hole causes difficult pressing or plate cracking; too large a hole leads to insufficient clamp load and loosening in service .

Layout and Avoidance: Sufficient space must be reserved around the press-in point to avoid interference with dies or other part structures. Also, press-in locations should avoid bend lines and weld zones, generally requiring a distance greater than 1.5 times the rivet height from the bend line .

2. Blind Riveting

Characteristics & Application: Suitable for enclosed or semi-enclosed structures accessible only from one side. Requires pre-drilling, using a rivet gun to pull the mandrel, expanding the rivet body to achieve fastening .

Design Points: Sufficient operating space must be reserved for the riveting tool .

3. Thread Forming (Extruded Holes)

Applicability: Suitable for relatively thin sheets (typically ≤ 2.0mm), forming a boss of a certain height by hole flanging, then tapping inside the boss to create threads. Thick plates are not suitable for thread forming via hole flanging .

Design and Material: Adequate distance must be maintained between flanged holes themselves, and between flanged holes and the sheet edge. Otherwise, it can easily cause sheet distortion, affecting thread strength and performance .

6.4.1 Design for Assembly: Datum Definition, Poka-Yoke, and Tolerance Stack-up Control

The core of Design for Assembly (DFA) is to simplify operations, prevent errors, and improve efficiency .

1. Datum Design

Unified Datum: A unified locating datum (e.g., "one plane, two holes") should be established and used during part design and assembly. This effectively reduces tolerance stack-up from datum conversion and improves assembly accuracy .

Process Datum: Process datums should be clearly marked on part drawings, such as using bend lines or specially punched process holes as references for subsequent assembly positioning .

2. Poka-Yoke (Error-Proofing) Design

Geometric Poka-Yoke: Design asymmetric mounting holes, different sized connectors, or unique snap-fit structures so that parts can only be assembled in the one correct orientation and way, fundamentally preventing reversal or misassembly .

Visual Aids: For similar parts or installations with directional requirements, clear markings, symbols, or color codes can be added to drawings or parts to guide correct operator action .

3. Tolerance Stack-up Control

Dimensional Chain Analysis: Perform dimensional chain analysis on critical assembly dimensions to calculate the stack-up effect of tolerances at each stage, ensuring final assembly accuracy. Where function allows, assign appropriately loose tolerances to non-critical dimensions .

Accounting for Surface Coating Thickness: Coatings like painting add thickness (approximately 60-90μm per side). When designing dimensional chains, especially for assemblies with multiple mating surfaces, the cumulative thickness of the coating must be considered and pre-compensated in the design (e.g., subtract 0.2mm per mating surface), otherwise it may lead to excessive assembly gaps or failure to assemble .

Assembly Clearance Design: Scientifically design assembly clearances based on product structure and size. For example, the clearance between a hinged door and its frame needs to increase with door height (H<1m, clearance 1.5mm; H>1.8m, clearance 2.5mm). The design of bolt clearance holes must also consider welding distortion and assembly adjustment allowances .

LEDA Summary: Excellent connection and assembly design is key to successful product mass production. At LEDA, we use 3D assembly simulation and tolerance analysis software to virtually verify assembly sequence, interference, and tolerance matching during the product design phase, identifying and resolving potential issues early. We strongly advise involving your manufacturing partner in collaborative reviews as early as possible in the design process, integrating assemblability considerations into every detail to achieve efficient and highly reliable production .

6.5 Corrosion Protection and Surface Treatment Design

Surface treatment is a critical step in the sheet metal manufacturing process. It provides functional protection against corrosion and wear and is also a core means of achieving appearance texture and brand identity. Scientific design can significantly enhance product value, lifespan, and market competitiveness .

6.1.12 Introduction to Common Surface Treatments (Electroplating, Electroless Plating, Painting, E-Coating)

Different surface treatment processes have their own physicochemical principles, application scenarios, and cost structures. The table below compares the core characteristics of four mainstream processes to help you quickly establish a selection framework .

Treatment Process	Technical Principle	Core Characteristics	Typical Application Scenarios	Cost Positioning
Electroplating	Metal cations from a solution containing the plating metal are deposited onto the cathode (sheet metal part) via electrochemical principles.	Uniform coating, high gloss, thickness can be precisely controlled. Sensitive to part shape, difficult to plate inner surfaces of deep holes/recesses; wastewater treatment cost is	Screws, connectors, bathroom hardware, automotive door handles − decorative or functional parts.	Medium

Treatment Process	Technical Principle	Core Characteristics	Typical Application Scenarios	Cost Positioning
		high.		
Electroless Plating	Deposits a metal coating (e.g., Nickel-Phosphorus) on the part surface via chemical reduction without external current.	Extremely uniform coating, no edge effect, perfect coverage of deep holes/complex internal cavities. High hardness, good wear resistance, but cost is usually higher than electroplating.	Hard disk substrates, precision gears, complex valve bodies, engineering parts requiring high uniformity and wear resistance.	Medium-High
Painting	Includes powder coating and liquid painting. Powder coating uses electrostatic adsorption followed by high-temperature curing; liquid painting uses spray guns to atomize the paint.	Powder coating: solvent-free, eco-friendly, thick coating (60-120μm), strong protection. Liquid painting: rich colors, special effects like metallic finish possible, but may have VOC emissions.	Enclosures/cabinets, appliance housings, outdoor facilities – large appearance parts.	Economic to Medium
E-Coating (Electrodeposition)	Similar to plating, but deposits paint resin. The part is immersed in the paint bath; electrification causes charged resin particles to move and deposit on the part surface, forming a dense coating.	Excellent uniformity, covers inner/outer walls, edges, welds of complex structures, no sagging, good edge coverage. Typically used as a primer (e.g., E-coat primer), requires mid-coat/topcoat.	Automotive bodies, chassis, complex frames – parts requiring high rust prevention.	Medium

LEDA Experience: At LEDA, we often use "combination processes" for optimal results. For example, for outdoor cabinets, a "E-coat primer + Powder coating" scheme is used, where e-coating ensures protection of internal cavities and welds, and powder coating provides excellent appearance and weather resistance, achieving a 1+1>2 effect .

6.5.1 Design Requirements of Different Surface Treatments on Sheet Metal Structures

Surface treatment processes directly influence product design. To ensure manufacturability and yield, their process characteristics must be fully considered during the structural design stage .

Design Requirements for Electroplating and Electroless Plating

Avoid Sharp Edges: Plating builds up at sharp corners and edges ("edge effect"), causing excessive thickness there and thin plating in internal corners. All external edges must be chamfered or rounded in design, recommended radius R≥0.5mm .

Blind Hole Design Specs: Blind hole depth should not be excessive, otherwise plating quality at the bottom is hard to guarantee. Depth should ideally not exceed half the hole diameter, and high color requirements for the bottom should be avoided .

Ensure Uniform Wall Thickness and Strength: The plating process occurs in a 60-70 ° C environment; parts may distort during racking due to heat or internal stress. Recommended wall thickness ≥1.5mm; thin-walled parts (t<1.2mm) need stiffening ribs to prevent distortion .

Reserve Racking Points: Plating requires racking. Design must consider reserving threaded holes (M6 or larger) or special holes (e.g., 10mm×10mm) for racking. The location should stabilize the part in the tank and avoid important appearance surfaces .

Design Requirements for Painting

Avoid Sharp Edges, Add Radii: Paint tends to pull away from sharp edges due to surface tension before curing, causing thin coating at edges. All sharp edges must be rounded, minimum radius R≥0.5mm .

Design for Masking and Drainage: For areas not requiring paint (e.g., threaded holes, precision mounting surfaces), design should consider the feasibility and operating space for tape masking. For enclosed or semi-enclosed cavities (e.g., chassis), drainage holes (φ≥6mm) are essential to prevent air pockets during powder/liquid painting and allow exhaust gas escape after curing .

Address Distortion Control in Large Flat Parts: Painting (especially powder coating) requires high-temperature curing (180-200 ° C). Large, flat sheet metal parts are prone to thermal distortion. Design should include pressed stiffening ribs (depth 0.3-0.5t) or folded hemmed edges to significantly increase rigidity .

Special Considerations for Electroless Plating (e.g., Electroless Nickel)

Leverage Throwing Power Advantage: Electroless nickel provides uniform coverage, suitable for complex deep holes, blind holes, and internal threads. If the product requires this, it can be prioritized, considering cost .

Beware of Hydrogen Embrittlement Risk: Acid pickling activation pre-treatment for electroless nickel may introduce hydrogen. For elastic parts or load-bearing components made of high-strength steel (tensile strength ≥1000 MPa), dehydrogenation annealing at 190-230°C for at least 3 hours must be performed within 4 hours after plating .

Design Coordination for E-Coating

Optimize Part Racking Orientation: The part's orientation in the e-coat tank affects film thickness distribution. Discuss the best racking points with the process engineer during design to ensure the part is oriented favorably for ion exchange and uniform film formation .

Avoid Enclosed Cavities: The part structure should avoid completely enclosed cavities, otherwise air pockets form during e-coating, preventing the inner surface from being coated. Suitably sized and numerous process holes must be designed for solution flow and gas escape .

LEDA Summary: Surface treatment is a combination of science and aesthetics. At LEDA, we incorporate surface treatment 方案评审 into the essential DFM (Design for Manufacturability) process. Through experience-based judgment and physical prototyping, we ensure your design possesses excellent processability while meeting functional and aesthetic requirements, laying a solid foundation for efficient, high-yield mass production .

6.6 Cost Optimization and Design for Assembly (DFA)

In sheet metal design, cost optimization is deeply coupled with assemblability. Excellent design not only reduces the direct manufacturing cost of the part but also significantly lowers the total cost by simplifying the assembly process and reducing operational steps. Statistics indicate that 70% of a product's total cost is locked in during the design phase, while only 7% of the actual cost has been incurred by then, highlighting the criticality of early design decisions. This section will systematically elaborate on three core optimization strategies.

The table below summarizes the key points and benefits of the three core cost optimization strategies.

Optimization Strategy	Core Method	Key Benefits	LEDA Practice Case
Design Simplification	Reduce part count, consolidate functions, minimize bend 次数, adopt common tooling designs.	Reduces mold complexity, decreases assembly labor hours, improves production stability.	A chassis side plate: integrated 8 riveted parts into 1 stamped and bent part, cutting assembly time from 15 minutes to 3 minutes, reducing cost by 35%.
Standardization	Unify sheet specifications, bend radii, fastener types; establish a corporate standard parts library.	Reduces material SKUs, simplifies processes, improves interchangeability, lowers procurement and management costs.	Unified 36 types of mounting brackets into 3 standard models, saving 500,000 CNY annually in mold costs.
Material Utilization	Optimize nesting layout, adopt common-cut (micro-joint) cutting, implement remnant management, accurately calculate material thickness.	Directly reduces raw material cost (typically 40%-60% of total), decreases scrap disposal costs.	Through auto-nesting software and common-cut technology, material utilization increased from 68% to 85%, saving 1.2 million CNY annually.

6.6.1 Design Simplification: Reducing Process Steps and Adopting Common Tooling

The core of design simplification is "**if not necessary, do not add entities**," aiming to achieve the required functionality using the fewest possible parts and process steps.

Reduce Part Count: Each independent part implies corresponding mold costs, processing time, assembly operations, and quality risk points. **Functional integration**, consolidating functions that would otherwise require multiple parts into a single part, is a fundamental cost-reduction measure.

Case Study: An electrical cabinet door lock initially required 8 parts (lock cylinder, faceplate, linkage, etc.) assembled via riveting and screws. After DFA optimization, it was redesigned as a single stamped and bent structure with integrated snap-fits formed in one stamping operation. Assembly time dropped from 15 minutes to 3 minutes, also eliminating screws and washers .

Reduce Number of Bends: Each bend is an independent process step requiring machine setup, positioning, and quality control. Design should scrutinize the bending sequence:

Combine Adjacent Bends: Through rational structural design, combine two perpendicular bends into one complex bend with an angled flange.

Evaluate Welding as an Alternative: For extremely complex multi-directional bends, evaluate whether using simpler bent parts combined with welding might be more cost-effective overall.

Use Standard Profiles: For certain frame structures, directly purchasing standard profiles (e.g., angle iron, U-channel) can be more economical than bending from sheet metal.

Adopt Common Tooling and Compound Processes: Completing multiple operations in a single press stroke within the die can greatly enhance efficiency.

Common Tooling: Using clever strip layout in the same die set to produce multiple identical or different parts in one press stroke. This requires sophisticated strip layout design but significantly increases production efficiency.

Compound Dies: Using a compound die allows multiple operations like punching, blanking, and drawing to be completed simultaneously in one press stroke, suitable for regularly shaped, high-volume parts.

LEDA Practice: At LEDA, we use DFA analysis software during the concept stage to quantitatively assess the "theoretical minimum number of parts" and use this as a target for structural optimization, simplifying the product architecture from the source .

6.6.2 Standardization: Unifying Sheet Specifications, Bend Radii, and Fasteners

Standardization is the cornerstone for reducing variation, achieving economies of scale, and improving quality and reliability. It manifests primarily at three levels:

Part and Feature Standardization:

Establish a Corporate Standard Parts Library: Define a preferred list for fasteners like screws, nuts, and press-in inserts, preventing designers from arbitrarily using non-standard parts. For example, standardize on M3, M4, and M5 screws company-wide, rather than a wide range from M2.5 to M6.

Feature Standardization: Standardize geometric features like hole sizes, hole pitches, and bend radii. For example, unify all internal hole sizes to a series like φ3.3, φ4.3, φ5.3, facilitating the use of standard tools and reducing tool change time.

Material Standardization:

Streamline Material Types: Create a "Preferred Materials List," recommending a limited number of sheet types based on different corrosion resistance and strength requirements, e.g., SPCC (general structure), SECC (general protection), SUS304 (high corrosion resistance).

Unify Sheet Thickness: Minimize the variety of different sheet thicknesses used in a single product. Prioritize thicknesses commonly stocked in the warehouse during design, e.g., 1.0mm, 1.5mm, 2.0mm, to avoid extra costs and lead times from sourcing special thicknesses.

Process Standardization:

Unify Bend Radii: Clearly specify in the design specification that for different thicknesses, the inside bend radius (R) should preferentially use a standard set of values (e.g., R0.5, R1.0, R1.5). This allows the workshop to standardize machine settings, reduce adjustments, and helps control springback.

LEDA Practice: LEDA establishes a standardized checklist for clients, mandating checks for compliance with established standards during design reviews. For instance, our system prompts designers if non-standard hole sizes or non-preferred fasteners are used, ensuring standardization is embedded in the process .

6.6.3 Material Utilization: Optimized Nesting and Nesting

Material cost typically constitutes 40%-60% of the total cost of a sheet metal part, making improved material utilization the most direct and effective lever for cost control.

Optimized Nesting Design: This is the primary task for improving utilization.

Dense Nesting: Utilize the automatic nesting function of CAD software to arrange parts on the sheet in the most compact way possible, minimizing the gap (bridge) between parts.

Common-Cut (Micro-Joint) Cutting: When the distance between parts is very small, common-cut technology can be used, where two parts share a cutting line. This saves material and reduces the cutting path length, improving laser cutting efficiency.

Mixed Nesting: Nesting parts of different shapes and sizes on one large sheet maximizes the use of blank areas, especially suitable for low-volume, high-mix production.

Remnant Management and Reuse:

Establish a Remnant Library: Number, record dimensions, and material of large remnants generated after production, incorporating them into a management system.

Design Collaboration: When designing small parts, designers should prioritize checking the remnant library to see if suitable-sized remnants are available for use, turning waste into treasure.

Precise Calculation and Thickness Optimization:

Select Material as Needed: Through calculation and simulation, select sheet thickness based on actual load-bearing requirements, avoiding waste caused by "over-engineering." Choosing a slightly thinner gauge that meets stiffness/strength requirements saves material and reduces weight.

Precise Blank Size Calculation: Accurate blank development size is the starting point for material saving. LEDA uses professional sheet metal unfolding software and calculates based on its own accumulated K-factor library to ensure accurate blank sizes, avoiding material waste from trial-and-error corrections .

LEDA Summary: Cost optimization is a systematic project requiring collaboration between design, process, procurement, and production departments. At LEDA, through collaborative DFM

(Design for Manufacturability) and DFA analysis, we front-load cost control to the source of product development. Our goal is not just to create a "workable" design, but an excellent design with optimal total cost throughout its life cycle .

6.7 Common Defects and DFM Prevention Strategies

Quality defects in sheet metal processing directly impact product accuracy, strength, and yield. This section, based on LEDA's manufacturing experience, systematically analyzes the causes of three core defects and provides specific prevention and optimization strategies from the design source (DFM).

6.7.1 Causes of Bending Springback and Dimensional Inaccuracy and Design Countermeasures

Cause Analysis

Bending springback is the phenomenon where the bend angle and radius of curvature change after the bending moment is removed, due to the recovery of internal elastic deformation. The root cause is that plastic bending is always accompanied by elastic deformation. Main influencing factors include:

Material Properties: Higher material yield strength and lower elastic modulus lead to greater springback. For example, stainless steel has significantly more springback than low-carbon steel.

Relative Bending Radius (r/t): A larger r/t ratio indicates less material deformation and a greater proportion of elastic deformation, leading to more significant springback. This is the main reason springback is difficult to control in large-radius bends.

Bending Method: Free bending has much greater springback than coining (bottom bending), which has a corrective effect.

Die Clearance: Excessive clearance between the punch and die reduces constraint during bending, increasing springback.

DFM Prevention and Optimization Strategies

Design Compensation (Most Common): Based on empirical data or CAE-simulated springback prediction, apply reverse compensation during die design. For example, if a 2° springback angle is predicted, the punch and die angles are machined at 88° so the part springs back to 90° after unloading. This is a key measure that can be anticipated and planned during the product design stage .

Optimize Product Structure:

Avoid Large Radii: Where function allows, use smaller bend radii to increase the degree of plastic deformation. For large radii required for appearance, adding stiffening ribs or embossed beads inside the arc can effectively suppress springback.

Use Complex Sections: U-shaped channels have less springback than V-bends; box-shaped sections have even less springback than U-channels due to mutual constraint. Adding flanges or hemmed edges can also effectively reduce springback.

Process Optimization:

Apply Coining: At the end of the bending stroke, sufficient pressure is applied to slightly thin the bend area, placing both inner and outer fibers in tension, thereby significantly counteracting springback. In practice, LEDA prioritizes coining for high-precision parts.

Select High-Performance Materials: Where strength requirements allow, prioritize materials with low yield strength and high elastic modulus, e.g., SPCC is better than SUS304.

LEDA Practice: At LEDA, we use the sheet metal modules of 3D CAD software and CAE springback analysis software to accurately simulate the bending process and predict springback during the design stage. For example, through software analysis, we successfully controlled the springback angle error of a server chassis side panel from 1.8° to within 0.5°, significantly improving assembly accuracy .

6.7.2 Causes of Bending Cracking and Surface Scratching and Prevention

Cause Analysis

Bending Cracking:

Excessively Small Bend Radius: Cracking occurs when the relative bend radius (r/t) is less than the material's minimum allowable bending radius, causing excessive stretching of the outer fibers.

Poor Material Condition: Poor material ductility, work hardening, or the material's rolling direction being parallel to the bend line can easily cause cracking.

Edge Quality: Having the burr side (from blanking) facing outward during bending acts as a stress concentration point, initiating cracks.

Harmful Elements and Inclusions: High levels of sulfur, phosphorus, or large inclusions in the steel severely degrade bending performance.

Surface Scratching:

Rough Die Surfaces: Small punch/die radii or surfaces with scratches/insufficient finish.

Unreasonable Bending Clearance: Excessive clearance between punch and die can scrape the part surface during bending.

Defects or contaminants on the raw material surface.

DFM Prevention and Optimization Strategies

Design Guidelines to Prevent Cracking:

Set a Reasonable Minimum Bend Radius: The inner bend radius R should not be less than the material's minimum allowed value during design. For low-carbon steel, R≥0.4t; for stainless steel, R≥ 0.8t-1.0t. Consult material manuals for specific values.

Optimize Bend Direction: Orienting the bend line perpendicular to the material's rolling direction significantly improves crack resistance.

Ensure Edge Quality: Clearly specify on drawings that the burr side should be placed on the inside of the bend. For high-requirement parts, specify deburring before bending.

Design Relief Notches/Process Slots: When a bend line extends to an edge or for local bends, pre-design relief notches at the ends of the bend line to release stress and prevent crack propagation. Slot width K≥t, depth L≥t + R.

Measures to Prevent Surface Scratching:

Optimize Dies: Appropriately increase punch radius, ensure uniform punch-die clearance. Specify high-polish finishes for dies used for high-gloss parts.

Material and Protection: Use high-quality sheets. For parts with extremely high appearance requirements, consider applying protective film before bending.

6.7.3 Control and Optimized Design for Welding Distortion

Cause Analysis

Welding distortion stems from non-uniform heat input. The weld zone metal expands upon heating, constrained by surrounding cooler metal, causing compressive plastic deformation. Upon cooling, this zone contracts, leading to various forms of distortion like shrinkage, bending, and warping.

DFM Prevention and Optimized Design Strategies

Controlling welding distortion must be planned from the product design stage.

Structural Optimization Design:

Reduce Weld Number and Size: Optimize the product structure to eliminate unnecessary welds where strength and function allow. Use the smallest sufficient weld size for necessary welds.

Aim for Symmetrical Layout and Balanced Heat Input: Position welds symmetrically about the structure's neutral axis where possible, allowing heat input and shrinkage from each side to counteract each other. Avoid highly asymmetric or concentrated weld patterns.

Use Stamped Parts Instead of Welded Assemblies: For certain structures, like stiffeners, using a single stamped stiffening rib is far better for controlling distortion than welding a rib plate.

Process Design Optimization:

Allow for Shrinkage and Use Anti-Distortion Technique: Pre-allocate shrinkage allowance during blanking and fixturing based on material, thickness, and weld size. A more effective method is pre-deforming the workpiece in the opposite direction of the predicted distortion before welding to counteract post-weld distortion.

Define a Scientific Welding Sequence: The principle is dispersed, symmetrical welding to ensure uniform heating of the structure. For example, use the backstep sequence for long welds; for symmetrical structures, use two welders working simultaneously and symmetrically.

Utilize Jigs and Fixtures: Proper welding jigs and supports that rigidly fix the workpiece can effectively suppress angular and bending distortion. Note that this may increase internal stresses.

LEDA Summary: At LEDA, through **"DFA Reviews"** and **"Welding Simulation Analysis"**, we assess weld feasibility and distortion risks during the product design phase. For instance, for large enclosures, we might recommend clients divide large panels into modular components, optimizing lap joint design and welding sequence to control overall flatness within 3mm/m, thereby avoiding subsequent extensive correction work and achieving efficient, high-quality production .

Chapter 7: 3D Printing (Additive Manufacturing) Design Guide

Additive Manufacturing (AM), commonly known as 3D printing, is a cutting-edge manufacturing technology that constructs physical objects layer by layer based on 3D model data. Fundamentally different from traditional "subtractive" machining and "formative" mold-based manufacturing, it employs a digital, stack-forming strategy, radically 颠覆了 (subverting) design thinking and manufacturing paradigms .

Currently, the 3D printing industry is experiencing rapid growth. It is predicted that China's 3D printing market size will exceed 63 billion RMB by 2025, representing approximately 30-fold growth over the past decade . This expansion is driven by its unique advantages in integrated manufacturing of complex structures, highly personalized customization, and rapid product iteration, enabling its shift from prototyping towards direct part manufacturing and establishing it as a significant driver of new productive forces.

For product design and manufacturing, the core value of 3D printing lies in its liberation of design freedom. Complex internal channels, lightweight lattice structures, and integrated functional designs that must be avoided in traditional processes become feasible with 3D printing . It allows designers to focus on the optimal form for functional realization, rather than compromises for manufacturability, thereby creating products with superior performance, lighter weight, and higher material utilization . Examples range from topologically optimized engine blades in aerospace to custom implants perfectly matching patient anatomy in healthcare.

However, this "freedom" is not without boundaries. To obtain high-quality parts efficiently and economically, it is essential to deeply understand and adhere to Design for Additive Manufacturing (DfAM) principles . This chapter will systematically analyze the entire process design guide for 3D printing, from process selection and structural innovation design to process implementation and post-processing, helping you master the key capabilities to transform ideas into quality products.

7.1 Additive Manufacturing Process Selection Guide: SLM, SLS, SLA, FDM Comparison and Application Scenarios

Selecting the appropriate additive manufacturing process requires comprehensive consideration across multiple dimensions. The following provides a comparison of the core characteristics of four mainstream processes, offering a clear framework for selection .

7.1.1 Core Characteristics Comparison Matrix of Four Major Processes

I. Accuracy and Surface Quality: Application Boundaries from Micron to Millimeter Level

Different AM processes vary significantly in accuracy and surface quality, which directly determines their suitable applications .

SLM (Selective Laser Melting) typically offers dimensional accuracy between **±0.05mm to ±0.2mm** and a surface roughness (Ra) of about **5-15µm** . The minimum feature size achievable is around **0.1mm**, making it suitable for precise metal parts with complex internal structures.

SLS (Selective Laser Sintering) technology has comparable dimensional accuracy to SLM, but its unique advantage is that it **does not require support structures** as the unsintered powder naturally provides support . This makes SLS particularly suitable for extremely complex geometries, although surfaces typically have a granular texture.

SLA/LCD (Stereolithography/LCD Curing) offers the highest dimensional accuracy, potentially **±0.1mm or better**, with layer thicknesses as low as **0.05-0.1mm** . Surface quality is excellent, ideal for display models and precision parts.

FDM (Fused Deposition Modeling) is a process with relatively lower accuracy, typically **±0.2mm to ±0.5mm**, with visible layer lines on the surface . Its accuracy is significantly influenced by nozzle diameter, layer height, and material shrinkage.

II. Material Systems and Functionality: Compatibility with Metals, Polymers, Ceramics, and Composites

The range of material choices directly determines the functional characteristics and application fields of the parts .

SLM is 专门用于 (specifically for) processing metal materials, including titanium alloys, aluminum alloys, nickel-based superalloys, stainless steel, etc. Parts made from these materials exhibit mechanical properties close to forged parts, suitable for high-strength, high-temperature environments.

SLS technology is compatible with various materials, including Nylon (PA), TPU elastomer, Polypropylene (PP), and metal-ceramic composites. Direct Metal Laser Sintering (DMLS) capabilities have also been developed more recently, further expanding its application range .

SLA/LCD primarily uses photosensitive resins, offering standard resins, engineering resins (high toughness, heat-resistant), casting resins, and other formulations. However, resin parts are often brittle and may age over time .

FDM has the widest material selection, including PLA, ABS, PETG, Nylon, TPU, as well as various engineering plastics and composites. In recent years, high-performance materials like PEEK and PEI have also been gradually applied in industrial-grade FDM equipment .

III. Efficiency and Cost Structure: Economic Analysis from Single Prototype to Batch Production

The economics of the production process directly impact return on investment and industrialization feasibility .

Build Speed: **FDM** is usually the slowest due to its point-by-point extrusion method; **SLS** and **SLM** use layer sintering/melting, offering medium speed; while the latest generation of **SLA/LCD** using area projection technology can achieve faster printing speeds.

Equipment Investment: Industrial-grade **FDM** equipment is relatively economical (tens of thousands to several hundred thousand RMB); medium-sized **SLS/SLA** systems range from several hundred thousand to a million RMB level; while large industrial-grade **SLM** systems can cost several million RMB.

Material Cost: **FDM** materials are the most economical; **SLA** resins are moderately priced; **SLS** powder and **SLM** metal powder costs are higher, especially for specialized alloy materials.

Table: Core Parameter Comparison of Four Major Additive Manufacturing Processes

Process Parameter	SLM	SLS	SLA/LCD	FDM
Accuracy Range	±0.05-0.2mm	±0.1-0.3mm	Within ±0.1mm	±0.2-0.5mm
Surface Roughness (Ra)	5-15µm	10-20µm	1-5µm	15-50µm
Typical Layer Thickness	0.02-0.1mm	0.08-0.15mm	0.025-0.1mm	0.1-0.3mm
Material Range	Metal Alloys	Polymer Powders, Some Metals	Photosensitive Resins	Thermoplastics
Support Requirement	Required	Not Required (Powder self-supporting)	Required	Required
Build Speed	Medium	Medium to Fast	Fast (Area Projection)	Slow
Equipment Cost	High	Medium-High	Medium	Low to Medium
Material Cost	High	Medium-High	Medium	Low

7.1.2 In-depth Analysis of Typical Application Scenarios

SLM (Metal): The Preferred Choice for High-Performance Functional Parts

SLM technology is particularly suitable for manufacturing lightweight aerospace components like engine blades and fuel nozzles, enabling complex internal cooling channels impossible with traditional processes. In the medical implant field, SLM can create orthopedic implants with porous structures that promote bone ingrowth. The mold industry uses SLM to create conformal cooling channels, improving cooling efficiency by 30%-50% .

SLS (Polymer/Metal): The Balance Point for Functional Integration and Small-Batch

Production

SLS technology is suitable for functional integration prototypes and small-batch custom parts, such as automotive intake manifold prototypes and UAV fuselages. Due to its support-free nature, SLS is ideal for products with complex internal channels and moving parts formed in one go. In small-batch production, SLS is widely used in customized medical devices, professional sports equipment, etc. .

SLA/LCD (Photosensitive Resin): High-Precision Appearance Validation and Rapid Prototyping

SLA/LCD processes are irreplaceable for producing high-precision appearance models, perfectly presenting product details and surface texture. In the medical model field, SLA can create high-precision anatomical models based on CT/MRI data for surgical planning and education. Precision casting patterns are another important application for SLA, especially suitable for jewelry and art casting .

FDM (Engineering Plastics): An Economic Solution for Concept Validation and Functional Tooling

FDM technology is most suitable for the concept validation and design iteration stages, allowing quick and economical verification of assembly relationships and basic 外形 (form). In manufacturing, FDM is widely used for creating jigs and fixtures, significantly reducing lead time and cost. With the emergence of high-performance materials, FDM is gradually being used for low-cost prototypes and certain end-use parts .

7.1.3 Selection Decision Process and LEDA Practice Case

Weighted Scoring Model Based on Performance, Accuracy, Volume, and Cost

Scientific process selection requires a systematic evaluation framework. LEDA employs a four-dimensional weighted scoring model to help clients quantitatively evaluate the optimal process route.

Step 1: Requirement Analysis

Performance Needs: Assess mechanical properties, temperature resistance, chemical stability, etc.

Accuracy Requirements: Determine dimensional tolerances and acceptable surface roughness range.

Volume Assessment: Clarify the required part quantity and subsequent production plans.

Cost Constraints: Set the project total budget and target cost per part.

Step 2: Process Screening

Establish a scoring matrix, rating each process's performance (1-5 points) against each requirement, and calculate a total score combined with weight factors:

Performance Weight (40%) + Accuracy Weight (25%) + Volume Adaptability Weight (20%) + Economics Weight (15%) = **Composite Score**

Step 3: Solution Validation

Validate the selected solution through small-batch trial production, collect data, optimize parameters, and finalize the mass production process route.

Case Study: Comprehensive Cost-Benefit Analysis of a UAV Bracket from FDM Validation to SLM Production

Project Background: An industrial UAV manufacturer needed to develop a high-performance carbon fiber composite UAV bracket requiring light weight, high strength, and environmental resistance.

Initial Solution: Initially used FDM technology to create prototypes with carbon fiber-reinforced nylon, quickly validating the mounting interface and basic structure at only 15% of the cost of a metal prototype.

Issues Identified in Performance Testing: The FDM prototype exhibited interlayer cracking during vibration tests, failed to meet fatigue life requirements, and showed significant anisotropy (Z-axis strength less than 60% of XY-axis strength).

Solution Optimization: Switched to SLM process using AlSi10Mg aluminum alloy, achieving 25% weight reduction while increasing stiffness through topological optimization and lattice structure design.

Cost-Benefit Analysis:

Mold Cost: Traditional machining required mold cost of ~50k RMB; SLM had no mold cost.

Production Lead Time: From design to prototype, the FDM+SLM combination took only 2 weeks, 60% shorter than traditional processes.

Cost Per Part: For small batches (<100 pieces), SLM cost per part was 128 RMB, significantly lower than machining (~350 RMB).

Performance Improvement: SLM part fatigue life was 5 times that of the FDM prototype, with 15% weight reduction.

Mass Production Decision: Based on the comprehensive evaluation, the client chose the SLM solution. Through LEDA's batch optimization and nesting, the number of parts per build increased from 8 to 15, further reducing the cost per part by 20%.

Through this systematic selection method, LEDA helps clients achieve optimal lifecycle costs while meeting technical requirements. This analytical approach has become a standard part of the LEDA DFM Guide, ensuring every project is built on a scientific process foundation from the start.

7.2 Core Design Thinking: Topology Optimization, Lattice Structures, and Integrated Functionality

The core advantage of Additive Manufacturing (AM) lies in its liberation of design freedom unattainable by traditional processes. Mastering the three key design philosophies — Topology Optimization, Lattice Structures, and Integrated Functionality — is essential for unlocking the full potential of 3D printing and achieving breakthrough improvements in product performance .

7.2.1 Topology Optimization: Algorithm-Driven Design for Performance and Lightweighting

Topology Optimization (TO) is a design method that uses mathematical algorithms to intelligently find the optimal material distribution within a given design space. Its core objective is to minimize material usage while meeting performance requirements (e.g., stiffness, strength), achieving ultimate lightweighting .

Software Workflow Based on Load Conditions

Taking mainstream software like Altair OptiStruct and ANSYS Topology Optimization modules as examples, the basic workflow is as follows :

Define Design Space: Import or create a 3D geometry representing the maximum possible volume where material can be placed.

Apply Loads and Constraints: Precisely define the real-world boundary conditions (e.g., fixed supports) and load cases (e.g., pressure, inertial forces) under which the part will operate.

Set Optimization Objectives and Constraints: A typical setup involves minimizing structural compliance (i.e., maximizing stiffness) as the objective, while constraining the material volume fraction (e.g., retaining only 30% of the material) .

Iterative Solving: The software performs iterative calculations based on methods like the Solid Isotropic Material with Penalization (SIMP) method, gradually removing material from low-stress areas, ultimately generating a material distribution cloud diagram .

Design for Manufacturability (DFM) Verification and Redesign Guidelines

The direct output of TO is often a rough "organic form" that requires manufacturability checks and redesign :

Geometric Reconstruction and Smoothing: Use software like ANSYS SpaceClaim or Materialise 3-matic to convert the optimized result into a smooth, usable CAD model. This process requires balancing the preservation of load paths with ensuring model quality.

Manufacturing Constraint Verification:

Minimum Feature Control: Ensure that generated beams and wall thicknesses are not smaller than the printer's minimum printable feature size (e.g., typically >0.3-0.5mm for SLM processes).

Overhang Angle Control: To minimize support structures, design features with angles preferably greater than 45° relative to the build plate. This can be input as a constraint before optimization or adjusted during post-processing.

Performance Validation: Conduct Finite Element Analysis (FEA) on the redesigned model to verify that its mechanical performance meets the optimization targets, avoiding the introduction of performance defects during redesign .

7.2.2 Lattice Structures: Precise Design for Lightweighting, Cushioning, and Heat Dissipation

Lattice structures are porous materials composed of microscopic unit cells arranged periodically in 3D space, enabling unique mechanical and physical properties . The table below compares common lattice types :

Lattice Type	Structural Characteristics	Mechanical Properties	Typical Applications
BCC (Body-Centered Cubic)	Node connection number is 8.	Bending-dominated, good energy absorption capability, and moderate stiffness.	Cushioning materials, shock absorbers, non-load-bearing filler.
FCC (Face-Centered Cubic)	Node connection number is 12.	Stretch-dominated, higher stiffness and strength, but lower toughness.	Load-bearing structures, lightweight brackets.
Gyroid	Triply periodic minimal surface.	Isotropic properties, uniform strength, also provides excellent fluid mixing characteristics.	Heat exchangers, implants (facilitating cell ingrowth).

Gradient Design of Lattice Parameters

The performance of lattice structures can be further optimized through gradient design :

Unit Cell Size Gradient: Use smaller, denser lattice units in high-stress areas to enhance strength; use larger units in non-critical areas to reduce weight.

Strut Diameter Gradient: Based on the stress contour map, thicken the struts of the lattice structure in high-stress areas and thin them in low-stress areas, achieving optimal material distribution and improving specific strength.

Research indicates that novel multi-biomimetic lattice structures (MHC), combining hollow design and edge optimization, can achieve multiples of improvement in specific energy absorption and specific stiffness compared to traditional BCC structures .

7.2.3 Integrated Functionality: From Multi-Part Assembly to Single-Piece Manufacturing

3D printing allows the integration of multiple parts into a single component, reducing assembly steps, improving reliability, and enabling new functionalities .

Conformal Cooling Channel Integration in Molds

In injection molds, traditional drilling can only create straight cooling channels. 3D printing can manufacture conformal cooling channels that follow the contour of the mold cavity surface, enabling uniform and efficient cooling. This can significantly shorten the injection molding cycle (up to 30%) and reduce part warpage .

Integrated Moving Structures: Design Considerations for One-Print Hinges and Springs

Assemblies containing moving parts, like living hinges and springs, can be printed in a single build.

Living Hinge Design: Commonly used in polymer printing (e.g., SLS, MJF). Requires selecting a material with good toughness (e.g., Nylon PA12). The hinge area should be thin (approx. 0.5-0.8mm) and designed with sufficient bend radius to avoid stress concentration.

Spring Mechanisms: Traditional coil springs or more complex leaf springs can be designed. The key is to reserve sufficient clearance, typically a unilateral clearance not less than 0.3mm, to prevent fusion after printing. Additionally, the print orientation should ensure that the interlayer bonding force is not the primary load-bearing direction.

7.2.4 LEDA Collaborative Design Support

At LEDA, we understand the challenges of translating these advanced design concepts into reliable products. Therefore, we provide comprehensive collaborative design support for our clients :

Simulation-Driven Design: We use professional software like Altair HyperWorks, ANSYS Mechanical, and nTopology to perform topology optimization, lattice structure performance simulation, and printing process simulation during the design phase, predicting and mitigating manufacturing risks early .

Design for Manufacturability (DFM) Review: Our engineering team, based on the specific 3D printing process (SLM/SLS/SLA, etc.), will conduct build orientation optimization, support generation strategy formulation, and distortion compensation analysis for your model, ensuring the design is not only high-performing but also efficient and economical to manufacture .

End-to-End Support from Design to Manufacturing: We are committed to acting as your extended engineering team, helping you transform ideas into high-quality, market-competitive products through professional technical support .

7.3 Design Golden Rules: Orientation, Self-Support, Wall Thickness, Holes, Fillets, and Detailed Design

Mastering the golden rules of 3D printing design is the critical bridge connecting creativity to successful manufacturing. These rules directly determine whether a part can be printed successfully, its accuracy, strength, and post-processing difficulty .

7.3.1 Print Orientation Optimization: Balancing Accuracy, Supports, and Mechanical Performance

Print orientation is the primary factor determining print quality, requiring an optimal balance between surface finish, support structures, build time, and mechanical performance .

Impact on Surface Quality: Upward-facing planes and near-horizontal curved surfaces have the smoothest finish and highest detail resolution. The bottom surface in contact with the build platform is typically rougher due to contact with the base plate or supports. Vertical surface quality falls in between. Therefore, the most critical aesthetic surface (e.g., the front face of a product housing) should be oriented facing upward along the Z-axis.

Impact on Support Structures: Orientation determines the amount of support needed. Adhering to the "**45-degree rule**" is key: when the overhang angle exceeds 45 degrees, supports are typically needed; angles less than 45 degrees might be self-supporting as each layer has sufficient underlying material, potentially avoiding supports. Supports increase material consumption, post-processing time, and can leave marks on the model surface after removal.

Impact on Mechanical Performance: 3D printed parts are anisotropic, meaning their mechanical properties differ depending on the direction. The interlayer bonding strength along the XY-axis (horizontal) is usually better than along the Z-axis (vertical). Therefore, for parts bearing significant stress, ensure the primary load direction is parallel to the layer deposition direction (XY plane), not perpendicular to it, to enhance part strength and reliability .

LEDA Practice Suggestion: At LEDA, we use specialized pre-processing software for multi-orientation simulation analysis, quantitatively evaluating support volume, estimated build time, and surface quality for different orientations, providing clients with data-supported optimal orientation solutions .

7.3.2 Self-Supporting Design: The 45° Rule for Support-Free or Minimized Support

Excellent design should minimize or avoid supports whenever possible, saving costs, reducing lead time, and ensuring surface quality .

Design Limits for Overhangs: As mentioned, 45° is a critical threshold. Designs should aim to keep overhang angles within 45°. For necessary overhangs, consider designing them as gradual "Y"-shaped support structures, which are more stable than right-angled "T" shapes and often printable without supports.

Bridging Design: Bridging refers to horizontal unsupported structures connecting two points. Short bridges (e.g., length < 36mm in FDM) can often be printed successfully with slight sagging. Long bridges require evaluation of sag or consideration of adding support pillars in the middle.

Protrusion and Hole Design: For horizontal blind holes, the top can be treated as a short bridge. For vertical holes, the top surface creates an overhang. A practical technique is to modify the top/bottom shape of the hole to a teardrop or diamond shape, effectively avoiding the need for supports.

7.3.3 Key Feature Design Specifications

Key Feature	Core Design Specification	Typical Requirements by Process (Reference)	Common Pitfalls & Consequences
Wall Thickness	Uniform wall thickness is the primary rule. Transition areas must be smooth (gradual change or fillet), avoiding sudden changes.	FDM: ≥1.0mm SLA/DLP: ≥0.5mm SLS: ≥0.8mm Metal SLM: ≥0.4mm	Too thin: part fragility; too thick/uneven: shrinkage, warping, sink marks, cracking.
Holes	Through holes are better than blind holes. Blind hole depth should preferably be ≤ 4x the hole diameter. Consider teardrop/diamond shapes to avoid supports.	Hole Diameter: ≥1.0mm (metal can be as low as 0.5mm)	Deep blind holes: difficult support removal; top of round holes: prone to sagging.
Fillets	All internal and external corners must have fillets. Recommended radius R ≥ 0.5mm or 0.5x wall thickness.	Increase radius appropriately for key stress areas.	Sharp corners are stress concentration points, primary cause of cracking under load/heat.
Text & Small Features	Prefer embossed (raised) text. For engraved text, width should be greater than depth, and depth should not be excessive.	Line Width: ≥0.2mm Raised Height/Engraved Depth: ≤0.5mm	Text too small/deep: blurry, difficult support removal.

7.3.4 LEDA DFM Checklist

Before sending a model for printing, use this checklist for a quick self-review to avoid 90% of common design issues .

Model Integrity Check

[ ] Is the model "manifold" (watertight)? I.e., no holes, self-intersections, or invalid faces.

[ ] Are the normals of all model faces correct (facing outward)?

Printability Check

[ ] Is the model's maximum dimension within the printer's build volume?

[ ] Are all wall thicknesses greater than the minimum required for the chosen process and material?

[ ] Are there local thick sections? Have thick walls been "cored out"?

Support and Orientation Optimization

[ ] Has the print orientation been adjusted to minimize support usage?

[ ] Have all overhangs been designed within the 45° rule where possible?

[ ] Does the hole design avoid unnecessary supports?

Detail and Safety Check

[ ] Have all edges and corners been filleted (R≥0.5mm) to eliminate stress concentration?

[ ] Are small features like text and logos clear and within process capabilities?

[ ] For moving parts or assemblies, is sufficient clearance reserved (typically ≥0.3-0.5mm)?

LEDA Summary: This checklist is the starting point of LEDA's collaborative design process. We strongly advise sharing your design with us early in the project. Our engineers will use professional software and extensive experience to provide a detailed DFM analysis report, working with you to optimize the design and ensure a successful print .

7.4 Process Implementation: Model Orientation, Support Design, and Generation Strategy

The success of 3D printing relies not only on excellent design but also on pre-print process planning. Rational model orientation, intelligent support design, and fine-tuned slicing parameter settings collectively determine the final part's quality, accuracy, and cost.

7.4.1 Model Orientation Strategy: Balancing Accuracy, Efficiency, and Surface Quality

The orientation of the model on the build platform is the starting point of process planning. It requires finding the optimal balance among multiple dimensions: surface quality, print time, support consumption, and mechanical performance .

Orientation Principle for Critical Surfaces: The most important aesthetic or mating surfaces of the product should be oriented facing upward along the Z-axis or placed vertically. This is because the upward-facing surface typically achieves the smoothest finish and highest detail resolution in processes like FDM and SLA, while the downward-facing surface, in contact with the platform or supports, tends to have the poorest quality, often showing support marks or roughness . For example, a car model should be oriented with its front side facing up and the chassis bottom as the supported surface.

Minimizing Support Contact on Key Aesthetic Surfaces: The orientation should be adjusted to ensure that support structures only contact non-critical surfaces of the part or areas that will be machined away later. By rotating the model, large overhanging surfaces can be cleverly transformed into self-supporting angles (close to or greater than 45°), or support points can be concentrated on internal or secondary locations .

Considering Physical Effects of the Printing Process: In powder bed-based 3D printing, the recoater blade generates a pressure wave while spreading new powder layers. If a large flat surface of the model directly faces the recoater's movement direction, this pressure wave can push against the already printed part, potentially leading to print failure. Therefore, the long edges or large inclined surfaces of the model should be set at an angle to the recoater's movement direction to disperse the pressure .

Optimizing Print Time and Strength: Orientation directly affects the number of print layers. For a tall model, printing it vertically results in the most layers and the longest time; printing it horizontally results in the fewest layers and the shortest time. Additionally, the part's anisotropy must be considered. For load-bearing components, the primary load direction should be parallel to the layer deposition direction (XY plane) to achieve higher strength and avoid the risk of interlayer delamination .

7.4.2 Intelligent Support Design: Applicable Scenarios for Manual and Automatic Generation

Support structures act as "scaffolding" for overhanging parts. The core design goal is to ensure print success while being easy to remove and causing minimal damage to the model surface.

Selection and Design Points of Support Types:

Tree-like Supports: Resembling tree branches, they have few contact points and are particularly suitable for supporting dispersed, point-like overhangs or complex curved surfaces. Advantages include material savings, ease of removal, and minimal surface damage; disadvantages include relatively poor stability, making them unsuitable for large-area overhanging structures .

Block/Grid Supports: Traditional linear array supports offer high stability and provide uniform, reliable support for large horizontal overhangs. Disadvantages include high material consumption and large contact area with the model, leaving noticeable marks after removal .

Water-soluble Supports: Use materials like PVA that dissolve in water as supports, printed simultaneously with the model material (e.g., PLA). Ideal for extremely complex internal cavities, nested structures, or any situation where tool access for support removal is difficult. The advantage is thorough support removal and good surface quality; disadvantages include higher material costs, slower printing speeds, and the requirement for a dual-extruder printer .

Design of the Support-Part Interface: This is the essence of support design, directly affecting removal difficulty and surface finish.

Easy-to-Detach Interface Design: By setting a Z-distance (typically 0.2-0.3mm) between the support and the model in the slicing software, a tiny gap is created, making the support easier to peel off the model .

Contact Point Shape and Density: For automatically generated supports, the density of the contact area at the support top can be adjusted. Reducing density can minimize adhesion. For manually added supports, use "point contact" or small area contact instead of large-area bonding .

Support Strategy for Complex Textured Surfaces: For overhanging bottom surfaces with fine textures (e.g., leather grain, mesh), directly adding supports can damage the texture and make cleaning difficult. An innovative method involves first generating a smooth, offset virtual surface based on the textured surface in 3D CAD software, then generating supports based on this virtual surface. This way, the top of the support is flat, easy to generate, and maintains a uniform distance from the actual textured surface, providing necessary support while avoiding the dilemma of support points embedding directly into the micro-texture, which is hard to remove .

7.4.3 Slicing Parameter Optimization: Comprehensive Tuning of Layer Height, Infill Density, and Speed

Slicing is the critical step of converting a 3D model into printer-executable instructions (G-code). Parameter settings determine the print's accuracy, strength, speed, and surface quality.

Core Slicing Parameters and Their Effects:

Parameter Objective	High Precision/Quality Mode	Balanced/Standard Mode	High Speed/Draft Mode	High Strength Mode
Layer Height	0.1 mm or smaller (Smoother surface, clearer details)	0.15 - 0.2 mm (Good balance of accuracy & speed)	≥ 0.28 mm (Fewer layers, faster print speed)	0.1 - 0.2 mm (Tighter interlayer bonding, slightly higher Z-strength)
Infill Density	15% - 20% (Saves material, reduces weight)	20% - 30% (For most general-purpose parts)	Below 15% (Further increases speed/saves material)	40% - 100% (Core parameter, significantly increases strength)
Infill Pattern	Gyroid, Honeycomb (Good strength/weight ratio)	Grid, Lines (Fast printing speed)	Lines (Fastest speed)	Honeycomb, Grid (Good structural stability)
Print Speed	≤ 40 mm/s (Slower outer walls improve surface quality)	50 - 60 mm/s (Universal speed)	≥ 80 mm/s (Sacrifices surface quality for speed)	40 - 50 mm/s (Ensures good interlayer adhesion)
Shell/Wall Thickness	≥ 3x nozzle diameter (Increases wall thickness, improves surface feel)	2-3x nozzle diameter (Standard configuration)	2x nozzle diameter (Reduces time)	≥ 4x nozzle diameter (Key parameter, shell bears main load)
Adaptation Scenario	Display models, final parts	Prototype verification, functional test pieces	Concept validation, internal draft models	Gears, load-bearing structures, moving parts

Synergistic Effects Between Parameters:

Speed and Temperature: Increasing print speed usually requires synchronously increasing the nozzle temperature to ensure material has sufficient fluidity and bonds well with the previous layer.

Layer Height and Temperature: When using thinner layer heights, due to less extruded material per layer and faster cooling, the nozzle temperature might need slight reduction to prevent overheating.

Infill and Shell: Increasing shell thickness is one of the most effective ways to enhance part strength, often more significant than simply increasing infill density.

LEDA Practice Suggestion: At LEDA, we have established parameter preset libraries for different project types and materials. We strongly advise clients to conduct small test prints before formal printing, such as printing a small evaluation model containing overhangs, bridges, pillars, and text, to quickly verify the printer's status and the effect under the current parameter combination, thereby avoiding wasting significant time and material .

7.5 Post-Processing and Quality Assurance: Heat Treatment, Surface Finishing, and Inspection Considerations

Post-processing and quality assurance are decisive steps for unleashing the final potential of 3D printed parts and ensuring they meet usage requirements. Scientific post-processing can significantly enhance the part's appearance, mechanical properties, and environmental adaptability, while strict quality inspection is the cornerstone of product reliability.

7.5.1 Comprehensive Analysis of Post-Processing Techniques

Post-processing can be systematically categorized into three main classes: support removal, heat treatment, and surface finishing. The table below summarizes the key points of core processes to help build a comprehensive understanding quickly.

Post-Processing Category	Specific Process	Core Principle	Main Applicable Materials	Key Effects & Limitations
Support Removal	Manual Removal	Using pliers, scrapers, etc., for mechanical removal.	Universal	Simple and flexible, but inefficient and can easily damage the part.
	Water-soluble Support	Immersing parts with PVA supports in water to dissolve.	FDM (PVA supports)	Handles complex internal cavities, no mechanical force required, but time-consuming.
	Thermal Dissolution Support	Immersing parts in specific heated liquid to melt support material.	Special support materials	Relatively efficient; temperature control is crucial to avoid part deformation.
Heat Treatment	Stress Relief Annealing	Low-temperature heating to eliminate residual internal stress from printing.	Metals (e.g., stainless steel, titanium alloys), Polymers	Prevents subsequent deformation/cracking; standard for metal printing.
	Hot Isostatic Pressing	Eliminates internal pores/micro-cracks under high temperature and pressure.	High-performance metals (e.g., IN718, Ti6Al4V)	Increases density (can reach >99.9%) and fatigue life.
	Aging/Precipitation Hardening	Precipitates strengthening phases at specific temperatures to increase strength/hardness.	Heat-treatable alloys (e.g., AlSi10Mg, 17-4PH)	Key for strengthening functional parts.
Surface Finishing	Sandblasting/Bead Blasting	Impacting surface with high-speed micro-particles (plastic beads, sand).	Metals, Polymers	Achieves uniform matte finish; fast processing; difficult for deep holes.
	Vibratory Polishing	Tumbling parts with abrasive media in a vibratory tub.	Metals, Polymers	Suitable for batch processing, uniform smooth surface, but may round sharp edges.
	Vapor Smoothing	Solvent vapor slightly melts the surface, causing it to reflow and solidify smoothly.	Some polymers (e.g., ABS, PA/Nylon)	High gloss, sealed surface; may lose fine details.
	Polishing/Grinding	Step-by-step mechanical grinding from coarse to fine.	Metals, Resins, Ceramics	Can achieve mirror finish; labor-intensive, not suitable for

Post-Processing Category	Specific Process	Core Principle	Main Applicable Materials	Key Effects & Limitations
				complex geometries.
	Coating/Plating	Applying a layer of paint or metal coating.	Universal	Changes appearance, improves corrosion/wear resistance; requires good adhesion.

Support Removal: From Manual to Smart Dissolution

Support removal is the first step in post-processing. The choice of method directly impacts efficiency and part integrity.

Manual removal is the most basic method, requiring tools like specialized scrapers and needle-nose pliers. Force must be applied carefully at the junction between the support and the model to avoid damaging the model's features. Breakaway points can be designed into the supports to improve efficiency.

Water-soluble supports are ideal for complex internal support structures. Using water-soluble materials like PVA for supports during printing allows them to dissolve when the finished part is immersed in water. The advantage is no risk of mechanical damage, capable of handling extremely complex internal channels; the disadvantage is the long duration (can take several hours).

Thermal dissolution supports are applicable to specific material systems, such as Breakaway Support or some specialized support materials, where heating softens or melts the support material for separation .

Heat Treatment: Key to Enhancing Intrinsic Properties

Heat treatment improves mechanical properties by altering the material's internal structure through controlled heating and cooling processes.

Stress Relief Annealing: This is an almost essential step after metal 3D printing. The significant temperature gradients during printing create high residual stresses within the part, which can cause warping or cracking during subsequent machining or use. Stress relief annealing, typically performed at relatively low temperatures (e.g., 400-600 ° C for stainless steel), can eliminate most of the residual stress.

Hot Isostatic Pressing: Primarily used for high-quality metal parts, such as aerospace engine blades. It is performed under high temperature (up to 1300°C+) and high pressure (100-200 MPa) in an inert gas environment, effectively closing internal pores and micro-cracks, significantly improving the part's density and fatigue performance.

Solution Treatment + Aging: For alloys capable of precipitation hardening (e.g., aluminum alloy AlSi10Mg, maraging steel 17-4PH), this is the core strengthening method. Solution treatment dissolves alloying elements into the matrix, followed by rapid cooling (quenching) to form a supersaturated solid solution. Aging treatment at lower temperatures causes the uniform precipitation of strengthening phases, significantly increasing the part's strength and hardness.

Surface Finishing: Transformation from "Rough" to "Refined"

Surface finishing aims to improve appearance, tactile feel, or meet functional requirements (e.g., reducing fluid drag).

Sandblasting/Bead Blasting: Cost-effective and efficient, quickly achieving a uniform matte surface. Different media (e.g., glass beads, alumina sand) can adjust surface roughness. A disadvantage is difficulty in processing shielded areas like deep holes and narrow slots.

Vibratory Polishing: Suitable for batch processing small parts, achieving a very uniform smooth surface. However, it may round sharp edges, and fine features might be worn away.

Vapor Smoothing: Mainly applicable to solvents that dissolve specific polymers like ABS and Nylon (PA). It achieves excellent smoothness and makes the part waterproof. However, part size is limited, and overexposure can cause part deformation or loss of details.

Mechanical Polishing: Can achieve the highest surface finish (mirror effect) but relies entirely on skilled labor, is time-consuming, and not suitable for complex geometries or mass production.

Coating Techniques: Spraying (e.g., painting) can change color and provide basic protection; Plating (e.g., nickel plating, chrome plating) can impart a metallic look, high hardness, wear and corrosion resistance, but requires the substrate to be conductive and have good adhesion with the coating.

LEDA Practice: At LEDA, we formulate tiered post-processing strategies based on the part's application scenario. For example, for a UAV bracket (metal SLM), we use a "**Stress Relief Annealing + Sandblasting**" combination to ensure strength and weight reduction; for a medical surgical guide (resin SLA), we adopt a full process of "**Precision Support Removal + Medical-grade Chemical Polishing + Gamma Ray Sterilization**" to ensure biocompatibility and usage safety.

7.5.2 Quality Inspection and Standards

Strict quality inspection is the final checkpoint ensuring 3D printed parts transition from "prototype" to "product," covering dimensions, internal quality, and mechanical properties.

Dimensional Accuracy Inspection:

Coordinate Measuring Machine: The authoritative standard for dimensional and geometric tolerance inspection. It obtains 3D coordinates of points by probing the part surface and compares them with the CAD model, with accuracy up to micrometers. Suitable for inspecting key hole positions, mounting surfaces, and geometric tolerances, but relatively slow.

D Scanning: Uses laser or structured light technology for non-contact, rapid acquisition of massive point cloud data from the part surface. Highly efficient, suitable for full-dimensional inspection, free-form surface verification, and reverse engineering. Professional software performs chromatographic deviation analysis against the CAD model, intuitively displaying the distribution of dimensional deviations.

Internal Defect Detection:

Industrial Computed Tomography: Like giving the part a "CT scan," it can non-destructively clearly reveal internal defects such as pores, cracks, and lack of fusion, and their 3D distribution. It is the most powerful method for inspecting the effectiveness of HIP and assessing the internal quality of load-bearing parts, but the equipment is expensive, and inspection and data analysis are time-consuming.

Ultrasonic Testing: Detects defects using the principle of ultrasonic wave reflection at interfaces like cracks or delaminations within the material. Sensitive to planar defects like delaminations and cracks, often used for spot checks of internal quality in simple-shaped metal parts, but difficult for parts with complex internal cavities.

Mechanical Performance Verification:

Mechanical performance testing requires preparing standard test coupons (typically printed in the same batch as the part and undergoing identical post-processing) and testing them on a universal testing machine, strictly following international standards (e.g., ASTM, ISO).

Tensile Test: Obtains key parameters like yield strength, ultimate tensile strength, and elongation after fracture, evaluating the material's strength and toughness.

Impact Test: Evaluates the material's toughness under high-speed impact loads, crucial for parts operating in low-temperature or dynamic load environments.

LEDA Summary: At LEDA, we integrate post-processing and quality inspection into a digital quality management system. Each part has a unique "process ID" recording full-process data from print parameters and heat treatment curves to 3D scan reports, ensuring full traceability. We recommend clients define quality acceptance standards early in the project, allowing us to customize the most cost-effective and efficient quality control plan.

Chapter 8: Precision Mold and Tooling Design Considerations

Precision molds and tooling are the core foundation of modern manufacturing, hailed as the "Mother of Industry." In mass production, the quality of the mold directly determines the product's accuracy, consistency, and production efficiency. As manufacturing advances towards high-end and intelligent development, the design of precision molds is no longer just about replicating part geometry; it has become a systematic engineering task requiring comprehensive consideration of material properties, forming processes, production efficiency, and cost control .

Current manufacturing demands are increasingly stringent: part dimensional accuracy is entering the micron level, structural complexity is continuously increasing, and production cycle requirements are significantly shortened. For instance, the accuracy of precision molds has evolved from 5μm a decade ago to 2-3μm today, with 1μm accuracy gradually becoming a reality. In this context, the traditional experience-dependent design model can no longer meet demands. Digital design tools (e.g., CAD/CAE), new material applications (e.g., high-performance mold steels, engineering plastics), and intelligent manufacturing technologies are becoming key drivers of mold technology transformation .

This chapter will systematically elaborate on the core considerations for precision mold and tooling design, aiming to equip you with the knowledge of how scientific design can significantly enhance mold life, optimize production efficiency, and reduce overall manufacturing costs while ensuring product function and quality. The content will cover everything from DFM (Design for Manufacturability) collaborative analysis to ensure mold feasibility, to mold structure optimization for high-volume production; from interface design for automated assembly, to systematic methods for 预留 tooling interfaces for quality inspection .

LEDA Perspective: In LEDA's engineering practice, we deeply recognize that a successful mold project results from the seamless collaboration of product design, mold engineering, and production processes. We strongly recommend initiating cross-department collaboration early in the project, using digital tools (e.g., mold flow analysis CAE) for preliminary simulation and verification, thereby mitigating risks at the source, shortening delivery cycles, and ensuring the first-time success rate of mold projects .

8.1 DFM Collaboration and Mold Feasibility Analysis

Mold DFM (Design for Manufacturability) collaborative analysis is a critical bridge connecting product design and manufacturing. Its core objective is to systematically identify and resolve manufacturability risks within the product structure through deep collaboration between the design team and the manufacturing team (mold engineers, process engineers) before machining the mold. A thorough DFM analysis can significantly reduce late-stage mold modification costs (industry statistics indicate reductions of up to 20% or more) and ensure the mold is capable of stable and efficient mass production .

8.1.1 Draft Angle Design Specifications and Surface Texture Adaptation Principles

The draft angle is a fundamental design feature ensuring that plastic parts can be ejected smoothly from the mold cavity without drag marks or damage. Its design is not a fixed value but requires comprehensive consideration of material, texture, and structural depth .

Core Principle: The fundamental purpose of the draft angle is to overcome the clamping force exerted on the core by the plastic after cooling shrinkage, and the friction between the part and the cavity surface. Insufficient draft angle leads to ejection difficulties, surface scuffing (white marks), ejection deformation, or even mold damage .

Material Differences: Plastics with different shrinkage characteristics require different draft angles. Hard plastics (e.g., PC, ABS, PS) require smaller angles, typically 0.5°~1.5° per side. Soft plastics (e.g., PP, PE) require larger angles, typically 0.5°~3° per side, to prevent vacuum suction during ejection due to their strong elastic recovery .

Surface Texture Adaptation: This is a critical yet easily overlooked point. A smooth surface finish requires the minimum draft. Once texturing (e.g., etching, EDM texture) is applied, additional draft must be added based on texture depth. A general rule is: for every 0.025mm increase in texture depth, add approximately 1° of draft. For example, a part with a VDI 3400 #A25 (medium) texture needs an additional 1°~2° of draft on top of the base value for a smooth surface .

Structural Depth and Location: The draft angle for cores (internal surfaces) should be greater than for cavities (external surfaces) because shrinkage tightens the part onto the core. For deep cavities or ribs exceeding 50mm, additional draft (e.g., add 0.5° for every 25mm increase in depth) is needed to compensate for the longer friction path .

Design Specifications:

Unified Datum: The drawing must clearly specify the datum edge for the draft angle (cavity surfaces often use the "small end" as the datum, cores use the "large end" as the datum).

LEDA Practice: At LEDA, we strongly recommend performing draft analysis on all 3D models, setting a check angle (e.g., 0.5°), using color maps to visually identify non-compliant areas, and confirming the minimum feasible draft for critical appearance surfaces with the client .

8.1.2 Uniform Wall Thickness Design and Local Thick Section Optimization

Wall thickness is the "soul" of injection molding design. Uniform wall thickness is the primary principle for avoiding defects like sink marks, warpage, and voids .

Core Principle: Strive for uniform wall thickness. Non-uniform walls cause differential cooling rates, leading to flow orientation, differential shrinkage, internal stress, and are a primary cause of part warpage. Sudden changes in wall thickness create stress concentration points, which can become initiation points for cracking .

Recommended Range: The reasonable wall thickness for most plastics is between 1.0mm and 4.0mm. Excessively thick walls (>4mm) lead to long cooling times, sink marks, and voids; excessively thin walls cause filling difficulties and insufficient strength .

Optimization of Local Thick Sections: For thick areas necessitated by functional requirements (e.g., screw bosses, mounting pads), they must not be simply made solid. "Coring" or "volcano" designs should be used—hollowing out the solid boss so its effective wall thickness matches the main wall thickness (typically achieved by connecting ribs to the boss)—fundamentally eliminating hot spots .

Rib Design Guidelines: Ribs are for increasing stiffness, not wall thickness. Their root thickness should not exceed 50%-60% of the main wall thickness (empirical value), their height should not exceed 3 times the main wall thickness, and the root must have a fillet transition (R ≥ 0.25-0.5t). Violating this rulevery easy causes sink marks on the opposite side of the rib .

8.1.3 Fillet Transitions and Stress Concentration Control

All sharp internal and external corners on the product are potential failure points and must be replaced with fillets .

Core Functions:

Eliminate Stress Concentration: Sharp internal corners are stress concentration points. Under impact or cyclic loading, cracksvery easy initiate here. Fillets effectively distribute stress, significantly improving the part's impact and fatigue strength.

Improve Material Flow: Fillets create smooth cavity contours, significantly reducing melt flow resistance, facilitating smoother filling, helping to reduce vortices and air traps, and improving pressure transmission efficiency within the mold cavity.

Enhance Mold Life: Sharp corners in the mold cavity are the origin of cracks under repeated thermal and mechanical stress. Fillet design avoids stress concentration, extending mold service life.

Design Specification: All internal and external corners should have fillets. The recommended internal fillet radius is R ≥ 0.5 times the wall thickness. The external fillet radius is R1 = R (internal radius) + wall thickness. Generally, all fillet radii should not be less than 0.5mm .

8.1.4 Ejection System Layout and Part Integrity Assurance

The ejection system is the final step in the molding cycle. Its design rationality directly determines whether the part can be ejected completely and undamaged .

Layout Principles: The ejection system (ejector pins, sleeve ejectors, ejector plates, air ejectors) must be laid out based on the principles of balance and sufficient force. Ejection force should act on areas with the best rigidity and highest clamping force (e.g., under cores, ribs, bosses), ensuring uniform force during ejection to prevent part warping or "ejector pin blushing" .

Avoiding Defects:

Blushing/High Marks: Improper pin location or insufficient area causes localized stress, whitening the material or creating raised marks.

Puncture: Pins that are too small in diameter or have excessive stroke can pierce through the part.

Location Avoidance: Pin locations must avoid critical appearance surfaces, functional mating surfaces, and weak structures (e.g., thin walls). The pin layout must be reviewed with the mold engineer to ensure no interference with cooling channels or internal cores .

Ejector Pin Type Selection:

Round Ejector Pins: Most common, cost-effective.

Blade Ejector Pins: Suitable for ejecting deep, narrow ribs.

Sleeve Ejectors: Used for ejecting cylindrical deep features (e.g., screw bosses), ensuring concentricity.

Air Ejection: Suitable for deep-cavity thin-walled parts, using compressed air to assist demolding and prevent vacuum distortion.

LEDA Summary: DFM collaboration is not a one-time audit but an iterative, cross-functional communication process. At LEDA, we use professional DFM reports, rich with graphics and text (including 3D model screenshots, wall thickness analysis cloud diagrams, problem point annotations), to conduct multiple rounds of review with clients, ensuring all potential risks are agreed upon and optimized before mold machining. Our goal is to resolve issues on the drawing board, not in the mold .

8.2 Mold Engineering Optimization for High-Efficiency Production

Efficient mold engineering optimization is a critical bridge connecting design and stable mass production. Its goal is to maximize production efficiency, extend mold life, and control overall costs while ensuring product quality. Systematic optimization across the following four dimensions is core to achieving this goal.

Optimization Dimension	Core Objective	Key Implementation Path
Structural Simplification	Improve reliability, reduce failure rate	Slider integration, undercut avoidance, motion simulation
Standardized Design	Shorten cycle time, reduce costs	Standard selection of mold bases, ejector pins, fasteners
Cooling System Optimization	Shorten cycle time, ensure quality	Water channel layout, thermal balance, conformal design
Material & Heat Treatment	Extend lifespan, ensure stability	Performance matching, surface strengthening, microstructure control

8.2.1 Mold Structure Simplification Strategy: Slider Integration and Undercut Avoidance

Complex mold structures are a primary source of production instability and failures. The core of simplification lies in reducing the number of moving parts and avoiding internal interference.

Slider Integration: For parts with multiple undercuts in similar directions, priority should be given to integrating several independent sliders into one compound slider. For example, using angled wedge mechanisms or hydraulic core-pulling systems to achieve "one movement driving multiple movements," significantly reducing mold complexity and assembly difficulty. Dynamic interference simulation must be performed during the design phase to ensure no collision risk for all moving parts throughout the mold opening/closing cycle.

Undercut Avoidance: The most effective simplification is to avoid undercuts at the product design source. Collaborative review with the client, by increasing draft angles or splitting a single part into two assemblable components, can fundamentally eliminate the need for side core-pulling mechanisms. If undercuts are unavoidable, prioritize using lifters instead of external sliders to handle internal undercuts, as they offer a more compact structure.

"Strong Ejection" Design: For non-critical appearance or functional surfaces, within allowable limits, smaller draft angles or even zero draft can be used, relying on the plastic's elasticity for "strong ejection." However, this requires precise evaluation of material toughness and selection of high-hardness mold steel (e.g., hardness HRC52-56) to prevent cavity scuffing.

8.2.2 Standardized Design: Mold Base / Ejector Pins / Fastener Selection Standards

Standardization is the cornerstone for improving design efficiency, ensuring mold quality, and reducing procurement and maintenance costs.

Mold Base Standardization: Prefer globally mainstream standard mold bases from suppliers like LKM, DME, or HASCO. This ensures mold base rigidity, guiding accuracy, and interchangeability, while shortening procurement lead times by approximately 60%-70%.

Ejector Pin Standardization: Ejector pin specifications should be unified, commonly using standard series like diameters Φ4mm, Φ6mm, Φ8mm, etc. The layout must be balanced to ensure uniform ejection force distribution. For ejecting deep ribs or around bosses, sleeve ejectors can be selected to ensure uniform force and avoid pin blushing.

Fastener Standardization: The specifications of fasteners (e.g., screws) for plate connection should be unified as much as possible. For instance, main plate fixation can uniformly use M10 or M12 screws. The screw engagement depth into the plate should be 1.5 to 2 times the screw diameter to ensure a secure connection. All screw hole positions should be fully detailed during the design stage and machined by CNC, avoiding on-the-spot fitting by toolmakers to guarantee precision and efficiency.

8.2.3 Cooling System Optimization: Water Channel Layout and Thermal Balance Design

The cooling system directly determines the production cycle time and part quality. Its optimization goal is to achieve rapid and uniform heat exchange.

Water Channel Layout Principles:

Balance: Cooling channels should uniformly surround the cavity. The distance from the channel centerline to the cavity surface is generally 1.5-2 times the channel diameter (typically maintained at 10-12mm) to ensure the cavity surface temperature variation can be controlled within 2-3°C.

Parallel Circuits Preferred: Using parallel circuits instead of series circuits ensures consistent flow rate and cooling intensity in each branch, avoiding insufficient cooling at the far end. The length of a single channel should not be excessively long, recommended to be controlled within 1.5 meters.

Conformal Cooling Channels: For molds with complex curved surfaces (e.g., irregular cores), conformal cooling channels manufactured using 3D printing technology are the optimal solution. They allow the channel shape to follow the cavity surface contour, achieving uniform cooling without dead zones. Research indicates that compared to traditional straight channels, conformal cooling can improve cooling efficiency by up to 40% and significantly reduce part warpage.

Precise Parameter Control: To ensure cooling effectiveness, the coolant should be in a turbulent flow state (Reynolds number $Re \geq 6000$). The inlet-outlet water temperature difference for precision molds should be controlled within 2-3°C, and temperature fluctuations should be kept within ±0.5°C via mold temperature controllers.

8.2.4 Mold Material Selection and Heat Treatment Process Planning

Materials are the skeleton of the mold, and heat treatment is the means to strengthen it; together they determine the mold's lifespan and stability.

Material Selection Based on Need:

General Plastic Parts (e.g., PP, ABS): Pre-hardened steels like P20 (3Cr2Mo) can be used, with a hardness of about HRC30. They can be machined directly without heat treatment, offering lower cost.

High-Appearance/Corrosion-Resistant Plastics (e.g., PC, PMMA): Select stainless steels like S136 (4Cr13), which offer excellent corrosion resistance and polishability, achieving a mirror finish.

High-Wear/Long-Life Production (e.g., glass-filled materials): Use hot-work tool steels like H13 (4Cr5MoSiV1), which, after heat treatment, possess high toughness, red hardness, and thermal fatigue resistance.

Heat Treatment Processes:

Vacuum Quenching + Multiple Tempering: This is the standard process for high-end tool steels. Heating in a vacuum state prevents surface oxidation and decarburization, followed by quenching and then 2-3 tempering cycles at different temperatures to stabilize the microstructure, eliminate internal stress, and achieve uniform hardness (e.g., H13 steel can reach HRC48-52) and high toughness.

Surface Strengthening Treatments: To further enhance wear resistance, surface treatments like nitriding (gas or ion) can be applied to form a high-hardness (up to HV1000 or more) nitride layer, increasing mold life by over 30%. PVD coatings (e.g., TiN, TiCN) can also significantly reduce the coefficient of friction and prevent sticking.

LEDA Practice: At LEDA, we establish a material and heat treatment file for each project. For example, for a gear mold expected to exceed one million cycles, we would specify H13 steel and implement a full suite of strengthening processes: vacuum quenching + double tempering + cryogenic treatment + PVD coating, ensuring dimensional stability and durability under long-term, high-speed operation.

8.3 Design for Adaptability to Automated Production

In automated production, the quality of part design directly determines whether the production line can achieve efficient, stable, and reliable operation. Scientific design can significantly reduce reliance on complex dedicated fixtures, increase production tempo, and fundamentally minimize risks like misloading, interference, and collisions. This section delves into the core design principles for automated production.

8.3.1 Datum System Design: Primary/Secondary Datum Coordination Principle

The positioning datum is the "anchor point" for precisely locating a part on a fixture or equipment. Its rational design is the cornerstone for achieving high-precision, repeatable assembly.

Core Principles:

Datum Coincidence and Unification Principle: This is the most important principle. Datum coincidence means making the fixture's locating datum coincide with the part's design datum (the origin point for dimensions on the drawing) as much as possible, eliminating errors from datum mismatch. Datum unification means using the same set of datums for positioning across multiple processes as much as possible to reduce cumulative errors from datum conversion and simplify fixture design.

Classic "One Plane, Two Pins" Model: This is the most common locating method in automated fixtures. The "one plane" (primary datum plane) restricts three degrees of freedom. The "two pins" (one round pin and one diamond pin) restrict the remaining three degrees of freedom. The primary datum plane should be the largest, most rigid, and flattest surface on the part to ensure stability.

Primary/Secondary Datum Coordination: The primary datum provides the main support and location, while secondary datums (e.g., side faces or holes) are used to restrict the remaining degrees of freedom. Secondary datums should be away from the primary datum to increase the moment arm, improving positioning accuracy and stability against torsion.

Design Key Points:

Datum Accessibility and Stability: Datum features must be easily accessible to fixture elements (e.g., locating pins, blocks), avoiding internal or hard-to-reach areas. The datum surfaces themselves must have sufficient rigidity and stability to prevent deformation under clamping force or gravity.

Avoid Over-constraint: It must be ensured that the locating system only restricts the necessary degrees of freedom without redundantly restricting the same degree of freedom, which could cause the part to be "locked" or induce assembly stress. For example, when using "one plane, two pins," one pin must be designed as a diamond pin to compensate for hole pitch errors and avoid over-constraint.

LEDA Practice: At LEDA, we strongly recommend clearly specifying primary and secondary locating datums during the part design stage. For example, on sheet metal parts, we design and call out three non-collinear process holes (typically two round holes and one slot) as the unified locating datum, used throughout all automated processes from stamping and welding to final inspection, ensuring accuracy consistency across the entire workflow.

8.3.2 Poka-Yoke (Error-Proofing) Design: Geometric Poka-Yoke and Sensor Integration Strategy

Poka-Yoke design aims to physically prevent the possibility of incorrect part installation or assembly, which is key to enhancing the reliability of automated production lines.

Geometric Poka-Yoke: Designing the part's geometry so it can only be installed in the one correct orientation and way.

Principle: Utilize asymmetry. For example, make a set of symmetric mounting holes different sizes or distances; design locating pin holes with different diameters; create asymmetric tabs or slots on the part contour or connectors.

Advantage: Very low cost, extremely high reliability, a "set-and-forget" error-proofing method. It requires no added electronics, eliminating the possibility of error at the physical level.

Sensor Integration Strategy: When geometric Poka-Yoke is not fully achievable, use sensors for verification during the assembly process.

Application Scenarios: Used to detect if a part is installed correctly, if the model is correct, if screws are tightened, if clips are engaged, etc.

Common Sensors:

Photoelectric/Fiber optic sensors: Detect part presence or position.

Proximity switches: Detect the position of metal parts.

Vision systems: Use cameras to read barcodes, markings, or specific features on the part to verify the product model; most powerful but higher cost.

Force control sensors: Integrated on the robot end-effector to monitor force/torque changes in real-time during assembly, ensuring a smooth process and preventing damage from excessive force.

The table below compares the characteristics and applications of different Poka-Yoke strategies:

Poka-Yoke Strategy	Implementation Principle	Advantages	Typical Applications
Geometric Poka-Yoke	Uses asymmetric design of the part itself or tooling to make incorrect installation impossible.	Zero operational cost, high reliability, no maintenance.	PCB connector 防反 (e.g., USB), cabinet panel misalignment prevention, connector reversal prevention.
Sensor Detection	Sets detection points in the assembly 流程; uses sensor signals to judge correctness.	Flexible, can detect incomplete installation, data recordable.	Checking screw tightening torque, part presence signal, product model verification via scanning.
Process Poka-Yoke	Uses PLC program logic to ensure subsequent steps cannot proceed if previous steps are incomplete.	Systematic control, avoids missed steps.	Machine won't start if safety doors are open; defective part from previous station isn't released.

LEDA Summary: The highest level of Poka-Yoke design is "preventing disease before it occurs." We recommend a combined strategy of "**Geometric Poka-Yoke first, sensor verification as supplement**." For example, when designing a plug-in module, first use asymmetric guide pins for geometric Poka-Yoke, then set a micro-switch at the insertion endpoint to detect full engagement, creating a double insurance.

8.3.3 Feeding Compatibility: Strip Layout Design and Grasping Feature Optimization

To adapt to automated feeding equipment (e.g., bowl feeders, conveyors, robots), part design must consider ease of separation, orientation, and grasping.

Avoid Tangling and Nesting: Part design should avoid hook-like, spring-like structures that easily interlock. For thin sheet parts, designtiny bumps or ribs in appropriate locations to reduce contact area and prevent adhesion due to oil film or vacuum suction.

Design a Stable Orientation: The part should have a clear center of gravity and a stable resting surface, facilitating smooth orientation in bowl feeders and stable transportation on conveyors. Avoid completely symmetrical structures like spheres; provide a clear reference surface for robots or grippers.

Optimize Grasping Features:

For Vacuum Cups: Provide sufficiently large, flat, and smooth surfaces for vacuum cup attachment. For non-planar parts, design dedicated process tabs as pickup points, which can be broken off or machined away after completion.

For Mechanical Grippers: Reserve clamping areas or grooves to ensure that functional surfaces are not damaged or deformed during gripping. The clamping area must have sufficient rigidity.

8.3.4 Tolerance Stack-up Control: Dimensional Chain Analysis and Tolerance Allocation

In automated assembly, errors from multiple parts accumulate along the assembly dimensional chain, potentially causing the final product to fail assembly or function. This cumulative effect must be controlled through scientific tolerance design.

Dimensional Chain Analysis: First, diagram the dimensional chain starting from the datum part, through all related parts, to the key assembly feature (e.g., fit clearance). This helps visualize the path of error propagation.

Tolerance Allocation: Assign appropriate tolerances to each link in the dimensional chain while meeting the final assembly's functional requirements. The basic principle is: assign loose tolerances to non-critical dimensions to reduce cost, and tightly control critical fit dimensions.

Tolerance Optimization Strategies:

Reduce Links: Minimize the number of links in the dimensional chain where functionally possible. Fewer links naturally lead to smaller stack-up.

Adjust Tolerance Grades: Based on machining capability, assign tight tolerances to dimensions that are easy to control, and appropriately loosen tolerances for difficult-to-machine dimensions where function allows.

Use Asymmetric Tolerances: Based on the fit type, use asymmetric tolerance zones. For example, for clearance fits, bias the tolerance zone towards increasing the clearance, relaxing manufacturing requirements while ensuring minimum clearance.

Introduce Adjustment Rings: Deliberately include an adjustable element (e.g., a shim) in a critical dimensional chain to compensate for the stack-up from preceding links during final adjustment, a common method for ensuring high-precision assembly.

LEDA Practice: At LEDA, we use 3D tolerance analysis software (e.g., CETOL 6 Sigma or VisVSA) for virtual simulation. By simulating the extreme material conditions (Maximum Material Condition, Least Material Condition) of parts in the computer, potential risks of assembly interference or excessive clearance can be accurately predicted and optimized during the design phase, significantly reducing assembly defect rates during mass production.

8.4 Test Tooling Interface Design

Test tooling serves as a bridge connecting product design and quality control. Excellent test tooling interface design not only enhances testing efficiency and accuracy but also guides production process optimization through data-driven feedback. This section will systematically elaborate on how to reserve interfaces from the design end for the smooth application of test tooling.

8.4.1 Datum Unification: Consistency with Manufacturing/Assembly Datums

Datum unification is the cornerstone for achieving precise and reliable measurement. Its core principle is to follow

the Datum Unity Principle: throughout the entire process of design, manufacturing, inspection, and assembly, the same set of datums should be used as much as possible to define the positional relationships of features on a part .

Core Value: Unified datums avoid cumulative errors from datum conversion at the source, simplify fixture design, and ensure that measurement results truly reflect assembly quality. If the inspection datums are not unified with the machining or assembly datums, the measurement data loses its guiding significance for the manufacturing process .

Implementation Path:

Clear Datum Definition in Design: The primary locating datums (typically following the 3-2-1 principle) and secondary datums should be clearly defined and annotated within the product's 3D model. These datums should prioritize features that are stable and reusable in both machining and assembly, such as large, flat mounting surfaces or precision-machined process holes .

Collaborative Design: The design of test tooling must be based on the product's design datums. The positions of locating elements (e.g., datum pins, support blocks) on the tooling must be strictly consistent with the locating datums used in the production line's assembly fixtures and machining centers .

LEDA Practice: At LEDA, we strongly recommend determining the datum strategy collaboratively through **"Concurrent Product and Process (CP) Meetings"** involving design, process, quality, and inspection engineers at the project initiation stage. This strategy is then mandated in drawings and technical specifications to ensure full-process datum unification .

8.4.2 Measurement Access Reservation: Probe Path and Sensor Space Planning

To ensure that inspection equipment (e.g., CMM probes, vision sensors) can access and identify measured features without obstruction, sufficient physical space must be reserved during the product design phase .

Space Planning: The working path of CMM styli or vision cameras should be virtually simulated within the product's 3D model. Focus on checking for interferences along the measurement path to deep holes, internal cavities, etc., ensuring the probe can approach the measurement point at the optimal angle and length without colliding with the part structure .

Avoidance Design:

For obstacles (e.g., side walls, bosses) in the measurement path, consider designing measurement access holes or relief slots.

For optical inspection (e.g., machine vision), ensure ample and unobstructed line-of-sight for cameras and light sources, considering the impact of reflections from surrounding structures on image quality .

Ergonomics: In addition to automated equipment, consider the operational space for manual gauges (e.g., calipers, plug gauges) to ensure operators can perform inspection actions conveniently and with minimal effort .

8.4.3 Critical Feature Measurability: Interface Design for Hole/Thread/Flatness Inspection

Different geometric features require different measurement methods; the design must match corresponding inspection strategies for their specific characteristics .

Hole Diameter and True Position:

Through holes are relatively simple to inspect and can be measured using a plug gauge or a Coordinate Measuring Machine (CMM). The design should ensure the hole depth is sufficient for the probe to enter smoothly.Blind Holes and Stepped Holes: More challenging. Require special styli (e.g., star probes or disc probes) or dedicated pneumatic plug gauges. The design should avoid complex shapes at the bottom of blind holes or excessively small diameters .

Thread Inspection:

Functional Inspection: Use thread plug gauges (GO/NO-GO) for rapid functional verification, determining if the thread size is within acceptable limits.

Parametric Inspection: Requires thread ring gauges or CMMs with specialized thread measurement software to obtain specific parameters like pitch diameter, pitch, and thread angle. The design must ensure the probe can access the complete thread profile .

Flatness and Profile:

Flatness measurement requires the probe to sample points uniformly across the entire plane. The design should ensure the measured surface is open and flat, avoiding protrusions or depressions that would hinder probe movement.

Profile measurement often requires 3D scanning. Consider adding auxiliary datum features or fiducial marks in non-critical areas to improve scan data registration accuracy and measurement efficiency .

8.4.4 Digital Inspection: DTS Matrix and Measurement Point Planning

Digital inspection is a trend in modern quality control, focusing on transforming inspection requirements into a structured, executable measurement plan .

DTS (Dimensioning and Tolerancing Specification) Matrix:

A DTS matrix is a structured table linking product key characteristics with measurement methods, tolerances, inspection frequencies, etc. It originates from product functional requirements and is the fundamental basis for guiding measurement point planning .

Each key dimension or tolerance in the matrix should have a corresponding, clear measurement plan .

Measurement Point Planning:

Function-Based: Measurement points should prioritize covering Key Product Characteristics (KPCs) / Critical Control Characteristics (CCCs) that affect product performance, assembly, safety, and appearance .

Scientific Distribution: The number and location of measurement points should accurately represent the feature's geometry. For example, measuring flatness requires evenly distributed points on the surface; measuring a round hole requires points on at least three different cross-sections .

Traceability: Each measurement point should have a unique identifier corresponding to an entry in the DTS matrix, ensuring traceability and analyzability of measurement data .

Digital Integration: Planned measurement points can be imported into CMMs or 3D scanners to generate automatic inspection paths. Results can be directly compared against the 3D CAD model using color deviation maps, visually highlighting out-of-tolerance areas and enabling closed-loop quality data management .

LEDA Summary: Test tooling interface design is the ultimate reflection of "Design for Quality." At LEDA, we utilize **MBD (Model-Based Definition)** digital design to embed inspection requirements, datums, measurement points, and other information directly into the 3D product model, achieving a seamless data flow from design to inspection. We assist clients in establishing DTS matrices and measurement point planning early in the product development phase, ensuring product measurability at the source and laying a solid foundation for building a robust and efficient quality control system .

Part 3: Special Topics in Universal Design and Collaboration

Chapter 9: Tolerance Analysis, Annotation, and Measurement Specifications – The Unified Language of Design and Manufacturing

In modern manufacturing, tolerance is the critical link connecting product design and physical production. It is a key technical language for ensuring parts achieve their intended functions in assembly and guaranteeing interchangeability. It is not merely the allowed dimensional deviation marked on a drawing; rather, it is a systems engineering task that coordinates design intent, manufacturing processes, and inspection methods .

The core value of tolerance design lies in balancing economy and precision. Overly loose tolerances may lead to assembly interference or functional failure, while excessively tight tolerances will exponentially increase manufacturing costs. Research indicates a significant positive correlation between the strictness of part tolerance requirements and manufacturing cost. Thus, rational tolerance design is key to optimizing product cost .

Traditional tolerance systems primarily relied on symbolic annotations on 2D drawings. With the development of digital manufacturing, **Model-Based Definition (MBD)** technology is gradually becoming mainstream. By directly embedding geometric tolerances, dimensions, and annotations (PMI) into the product's 3D model, it creates a single source of truth, enabling seamless collaboration and unambiguous transfer of information across design, process planning, manufacturing, and inspection stages .

This chapter will systematically analyze the complete chain of tolerance analysis, annotation, and measurement specifications. Starting from the rational application of Geometric Dimensioning and Tolerancing (GD&T), we will explain how it precisely conveys functional requirements. We will delve into how dimensional stack-up analysis quantifies and predicts accumulated assembly variation, thereby mitigating risks during the design phase. Finally, we will focus on LEDA's collaborative workflow, demonstrating how to translate design annotations into executable measurement plans, forming a digital closed-loop for quality control, and providing a methodological guarantee for achieving "right-first-time" production .

Rational Application of Geometric Dimensioning and Tolerancing (GD&T): From Functional Requirements to Drawing Specifications

Geometric Dimensioning and Tolerancing (GD&T) is a core technical language on engineering drawings for precisely communicating design intent and ensuring parts fulfill their intended functions in assembly. It defines the allowable variation in the form, profile, orientation, location, and runout of part features through a standardized system of symbols. The rational application of GD&T fundamentally avoids the dilemma of "the drawing is correct, but the parts cannot be assembled" and is the foundation for achieving efficient manufacturing and reliable quality .

I. Datum Reference Frame Establishment Principles and Assembly Function Simulation

The Datum Reference Frame (DRF) is the cornerstone of GD&T. It establishes a theoretically exact set of points, lines, and planes (the datums) on a part to provide a measurement origin for locating and orienting all other features within the geometric space. Its establishment process essentially simulates the part's assembly function within the final product .

II. Core Establishment Principles

3-2-1 Locating Principle (Six-Point Locating Principle): This is the most fundamental principle for establishing a datum reference frame. It constrains three degrees of freedom (one translation, two rotations) using three non-collinear points for the primary datum (usually Datum A); constrains two degrees of freedom (one translation, one rotation) using two points for the secondary datum (usually Datum B); and constrains the final degree of freedom (one translation) using one point for the tertiary datum (usually Datum C). This ensures all six degrees of freedom of the part in space are fully constrained, resulting in a unique and stable position .

Function-First Principle: Datum selection must be based on the part's assembly function. The primary datum (A) should be selected from the feature that has the largest contact area with the mating part and best determines its core position (e.g., a primary mounting surface). The secondary (B) and tertiary (C) datums should be selected from features that constrain the remaining critical degrees of freedom to ensure correct assembly orientation .

Stability and Repeatability: Features selected as datums must themselves have sufficient size and form stability, and be easy to locate stably and repeatably during manufacturing and inspection .

Assembly Function Simulation

When annotating datums on a drawing, designers should constantly ask: "How will this part assemble with its adjacent parts?" For example, when a chassis mounting plate assembles to a base, its bottom surface first contacts the base (simulating primary datum A), then one side surface aligns with a locating pin or another side (simulating secondary datum B), and finally perhaps an end surface completes the final location (simulating tertiary datum C). Establishing the datum reference frame is about precisely replicating this assembly process on the drawing .

I. Common Misconceptions to Avoid

Misconception 1: Improper Datum Selection. Avoid selecting features that are too small, lack shape stability, or are difficult to measure accurately (e.g., short bosses, free-form surfaces) as datums .

Misconception 2: Incorrect Datum Sequence. The datum sequence (e.g., |A|B|C|) defines the priority of constraint application. An incorrect sequence can lead to inspection results that do not match the assembly condition .

Misconception 3: Over-constraint. Not all situations require three datums. For features that only need to ensure position within a plane, perhaps only datums |A| and |B| are necessary, provided the function is satisfied .

II. Geometric Tolerance Annotation Standards: Core Application Scenarios and Misconception Avoidance

The following details three of the most commonly used and critical geometric tolerances: position, profile, and runout.

Tolerance Type	Core Function	Typical Application Scenarios	Annotation Key Points & Common Misconceptions
Position	Controls the true position of a feature's axis (e.g., hole, pin)	Bolt connection hole patterns: Ensure multiple	Key Points: Must be used in conjunction with True Position (Theoretically Exact Dimensions - TEDs, dimensions enclosed

Tolerance Type	Core Function	Typical Application Scenarios	Annotation Key Points & Common Misconceptions
	relative to one or more datums.	holes align simultaneously with the mating part. Pin locating holes: Achieve precise part positioning.	in a rectangular frame). The tolerance zone is a cylindrical or rectangular area. Common Misconceptions: 1) Not using TEDs for location leads to incorrect tolerance zone interpretation. 2) Incomplete datum system fails to uniquely define the feature's theoretical position. 3) When applying position tolerance to a threaded hole, it should be directed to its axis, not the thread form surface.
Profile	Controls the deviation of a part surface or profile line from its ideal shape.	Complex curved surface parts: e.g., automotive body panels, aircraft wing skins. Mating surfaces: Surfaces requiring strict sealing or contact.	Key Points: Divided into surface profile and line profile. Can simultaneously control a feature's form, orientation, and location. Common Misconceptions: 1) Misunderstanding the profile tolerance zone, which is a bilateral zone equally distributed on either side of the theoretical profile. 2) Incorrect datum usage: Profile without a datum only controls form; profile with a datum controls form, orientation, andlocation. 3) Using profile tolerance for regular features that should be controlled by simpler tolerances (e.g., flatness), resulting in over-design.
Runout	Controls the total variation in a specified direction when a feature is rotated one full revolution about a datum axis. It comprehensively controls errors like coaxiality, circularity, and straightness.	Rotating components: e.g., transmission shafts, bearing seats, gears. Ensures smooth operation and reduces vibration and wear.	Key Points: Divided into circular runout (single measurement cross-section) and total runout (entire feature surface). Common Misconceptions: 1) Misusing the datum: Runout must reference a datum axis. Referencing an incorrect datum (e.g., a plane) renders the requirement ineffective. 2) Confusing circular runout and total runout. Total runout is a more stringent requirement, controlling the composite error across the entire cylindrical surface. 3. Incorrectly applying runout tolerance to non-rotating parts.

LEDA Practice Recommendation: At LEDA, we strongly recommend conducting **"Virtual Assembly" analysis** early in the project phase. Utilizing the tolerance analysis modules within 3D CAD software, based on your defined GD&T, allows simulation of part assembly under extreme material conditions, thereby verifying the rationality of the tolerance design before manufacturing and mitigating the risks of interference or excessive gaps at the source .

9.1 Dimensional Chain Analysis and Assembly Stack-up Error Control: Worst-Case, Root-Sum-Square, and Monte Carlo Simulation

Dimensional chain analysis is a core engineering technique that connects product design with assembly accuracy. Its essence lies in systematically modeling the interaction between part dimensions and tolerances to quantitatively predict the accumulated variation in the final assembly, thereby identifying and resolving potential risks of interference, excessive gaps, or functional failure during the design stage. Scientific dimensional chain analysis significantly reduces costs and lead times during mold tryouts and debugging, serving as a key guarantee for achieving efficient, high-precision mass production .

Dimensional Chain Modeling Methodology: From 1D Linear Chains to 3D Spatial Tolerance Modeling

Establishing an accurate dimensional chain model is the foundation of the analysis. The core process follows principles from overall to local and from simple to complex.

Step 1: Identify the Closed Loop

Definition: The closed loop is the dimension that is naturally formed last in the assembly process or machining sequence. It is typically the key assembly gap, fit interference, or critical positional accuracy that needs to be controlled (e.g., the axial clearance of a gear assembly or the step difference between two housing surfaces after closing) .

Key Point: The closed loop is the target and output of the dimensional chain analysis. Its tolerance is determined collectively by the tolerances of all component links, making its resulting tolerance range the largest.

Step 2: Identify All Component Links

Definition: Component links refer to all part dimensions and their tolerances that have a direct impact on the closed loop. These dimensions must form a closed chain .

Method: Start from one end of the closed loop and trace along the assembly or part, identifying all directly related dimensions sequentially until returning to the other end of the closed loop. Ensure the path is unique and closed, avoiding omissions or introducing irrelevant dimensions.

Step 3: Determine Increasing and Decreasing Links

Increasing Link: When the size of this component link increases, it causes the closed loop dimension to increase in the same direction.

Decreasing Link: When the size of this component link increases, it causes the closed loop dimension to decrease in the opposite direction .

Technique: On the drawn dimensional chain diagram, mark all links with a single-direction arrow. Links whose arrows point in the same direction as the closed loop's arrow are decreasing links; those pointing in the opposite direction are increasing links.

Step 4: Model Extension from 1D to 3D

1 **D Linear Dimensional Chain:** All dimensions lie in the same direction or on parallel lines. This is the simplest and most basic model, suitable for axial assembly analysis of stacked parts.

2 **D Planar Dimensional Chain:** Dimensions lie within a plane but have different directions, involving angular dimensions and projection relationships. Trigonometric calculations are needed to resolve diagonal dimensions into components along rectangular coordinates before incorporating them into the calculation.

3 **D Spatial Dimensional Chain:** This is the most complex model, considering deviations in all six degrees of freedom (three translational, three rotational). This requires the use of vector chains, homogeneous transformation matrices, or specialized tolerance analysis software to establish a model that accurately simulates the actual variation accumulation of parts in space .

Tolerance Analysis Algorithm Selection: Applicable Scenarios and Precision Comparison of Worst-Case (WC), Statistical (RSS), and Monte Carlo Simulation (MCA)

Choosing an analysis algorithm is essentially a trade-off between risk (precision), cost, and efficiency. The table below clearly compares the characteristics of the three core algorithms.

Characteristic Dimension	Worst-Case (WC)	Statistical Method (RSS)	Monte Carlo Simulation (MCA)
Core Principle	Assumes all part dimensions are simultaneously at their limit deviations (Maximum/Least Material Condition).	Based on probability theory, assuming part dimensions follow a normal distribution centered on the nominal value.	Uses computer random sampling to perform thousands/millions of "virtual assemblies," simulating real production variation.
Calculation Assumptions	All parts (100%) fall at tolerance zone limits.	Dimensions are independent, normally distributed, mean coincides with nominal, process is controlled (e.g., SPC).	Can define any distribution shape (e.g., skewed, uniform), tolerance zone shift, and correlation for each dimension.
Result Meaning	Absolute extreme range of the assembly dimension.	Statistical distribution range of the assembly dimension at a specific confidence level (e.g., ±3 σ, 99.73%).	Simulated probability distribution of the assembly dimension; can calculate exact defect rate (PPM).
Main Advantage	100% interchangeability; simple	More aligned with high-volume	Highest accuracy; can handle any complex

Characteristic Dimension	Worst-Case (WC)	Statistical Method (RSS)	Monte Carlo Simulation (MCA)
	concept and calculation; absolutely safe.	production reality; allows looser part tolerances; cost-effective.	scenario (non-linear, non-normal, 3D spatial chains).
Main Limitation	Overly conservative; leads to overly strict part tolerances and high manufacturing costs.	Relies on ideal assumptions; underestimates extreme risk if production is unstable (mean shift, non-normal).	Computationally complex; requires specialized software, higher setup barrier, and consumes more resources.
Applicable Scenarios	Applications requiring extremely high safety, very few parts (n≤4), and 100% interchangeability (e.g., aerospace, medical implants).	High-volume, quality-stable (high Cp/Cpk) conventional products (e.g., consumer electronics, automotive parts).	Complex mechanisms, high-precision prediction needs, low-volume high-value products, and risk assessment when process capability is unknown or unstable.

Algorithm Selection Strategy and Engineering Practice:

Worst-Case (WC): Its calculation formula is: Closed Loop Tolerance (T_0) = Sum of all Component Link Tolerances (T_i), i.e., $\mathbf{T_0 = \Sigma T_i}$. This implies it pursues "absolute certainty," but at the cost of being the most expensive. It is used only when the consequences of failure are extremely severe .

Statistical Method (RSS): Its core formula is: Closed Loop Tolerance $\mathbf{T_0 = \sqrt{(\Sigma T_i^2)}}$. This method acknowledges the extremely low probability that all parts are simultaneously at their worst-case condition, thereby significantly reducing machining difficulty and cost while accepting a tiny, acceptable risk of failure (e.g., 0.27%). Implementing the RSS method requires the production process to have stable and reliable statistical process control (SPC) to ensure the dimensional distribution meets expectations .

Monte Carlo Simulation (MCA): This is the most powerful and realistic method. It does not rely on a single mathematical formula but directly calculates the assembly yield rate through simulation. MCA is the only tool that can provide accurate predictions for products involving kinematic mechanisms, flexible parts, or complex 3D spatial tolerances .

LEDA Practice Summary: At LEDA, our tolerance analysis follows a tiered decision-making process:

Preliminary Analysis: For most projects, prioritize using the RSS method for rapid assessment, balancing economy and risk.

Critical Verification: For critical dimensional chains related to safety and reliability, use the Worst-Case method for redundancy checks.

In-depth Simulation: For complex systems (e.g., vehicle door-to-body side matching, precision transmission mechanisms), we employ Monte Carlo Simulation combined with 3D tolerance analysis software for virtual assembly analysis. This accurately predicts production variation and allows for tolerance optimization during the design phase, enhancing product manufacturability and quality stability at the source.

By mastering the modeling methods and algorithm selection strategies above, you can effectively translate design intent into quantifiable and controllable manufacturing quality objectives, providing core support for robust design.

9.2 Balancing Cost and Tolerance: Process Capability-Based Tolerance Optimization Design

A core challenge in tolerance design is achieving the optimal balance between manufacturing cost and product quality (ensured by tolerance requirements). Excessively loose tolerances may lead to product functional failure or assembly difficulties, while excessively tight tolerances exponentially increase machining difficulty, inspection costs, and product defect rates. Scientific tolerance optimization design is essentially about making informed decisions based on quantified process capability data to maximize cost-effectiveness .

To help you quickly establish an overall understanding, the table below summarizes the core methodologies and key parameters discussed in this subsection.

Core Concept	Description	Relationship with Cost	Key Parameters/Standards
Process Capability Index (Cp)	Measures the inherent ability of a process, under statistical control, to meet technical specifications. Does not consider process mean shift.	A high Cp value indicates low process variation and high potential, but may imply high costs due to excessive precision pursuit.	Cp = T / (6σ) (T is the tolerance range, σ is the process standard deviation).
Process Capability Index (Cpk)	Measures the actual ability of a process output to meet technical specifications. Considers both process variation and mean shift.	Cpk is a key indicator for balancing cost and quality. Low Cpk implies high defect rates; excessively high Cpk often indicates unnecessary cost investment.	Cpk = Min[(USL-μ)/3σ, (μ-LSL)/3σ]. Typically, Cpk ≥ 1.33 is considered acceptable process capability.
Tolerance Optimization Allocation	A systematic method to tighten tolerances for critical dimensions and relax them for non-critical dimensions, based on process capability.	The goal is to achieve the lowest total cost (manufacturing cost + quality loss) while meeting product function.	Requires quantitative analysis combined with cost-tolerance models (e.g., exponential model, negative power model).

I. The Linkage Model Between Process Capability Indices (Cp/Cpk) and Tolerance Design

Process capability indices are the core tools for quantifying how well a manufacturing process's consistency matches tolerance requirements. They provide an objective data foundation for tolerance design, linking design intent with manufacturing reality .

Engineering Significance of Cp and Cpk

Cp (Process Precision Index): This index assumes the process mean is perfectly centered on the tolerance midpoint. It only reflects whether the inherent process variation (6σ) can be contained within the tolerance range (T). A high Cp value (e.g., >1.67) indicates high process potential with low variation, but it does not account for any shift in the process mean.

Cpk (Process Capability Index): This index considers both the process variation and the shift of the process mean relative to the tolerance center. Therefore, Cpk ≤ Cp, and it more accurately reflects the actual quality level of the process output. Cpk is the gold standard for evaluating whether an existing production process can stably meet drawing requirements .

Direct Link Between Cpk and Tolerance Design

Tolerance design determines the tolerance range width (T) and the center position (M). The Cpk calculation formula directly incorporates these elements:

Cpk = (1 - k) * Cp, where **k = |M - μ| / (T/2)**.

This formula clearly reveals their relationship:

Given a manufacturing process (i.e., fixed σ and μ): Relaxing the tolerance (increasing T) will directly lead to higher Cp and Cpk values. This means the same manufacturing equipment will exhibit higher process capability indices and expected yield rates when producing products with lower precision requirements.

Given tolerance requirements (fixed T and M): The tolerance requirements on the design drawing present a Cpk challenge target for the manufacturing process. To achieve the required Cpk value (e.g., 1.33), the process variation (σ) and shift (|M-μ|) must be controlled within corresponding limits.

Process Capability-Based Design Decisions

During the design phase, engineers should consult or evaluate the supplier's process capability data (Cp/Cpk). If historical data indicates that the manufacturing Cpk for a certain type of feature (e.g., precision hole spacing) is consistently low (e.g., <1.0), and this feature is not a critical fit dimension, then appropriately relaxing its tolerance is a more economical and robust choice. Conversely, for safety-critical dimensions, strict tolerances must be specified, and suppliers must demonstrate high Cpk (e.g., ≥1.67) .

II. Tolerance Allocation Strategy for Cost Optimization: The Art of Trade-offs Between Tight Control of Critical Dimensions and Relaxation of Non-Critical Dimensions

The core of tolerance optimization is **"using resources where they matter most,"** investing cost in areas critical to product function while releasing cost pressure in non-critical regions.

Identification and Tight Control of Critical Dimensions

What are critical dimensions? These are dimensions that directly affect the product's core function, safety, reliability, or primary assembly performance (e.g., fit bore diameter for a bearing, center distance for gear transmission, compression amount of a seal).

Tight Control Strategy: For critical dimensions, relatively tight tolerances should be specified during the design phase, and suppliers should be explicitly required to achieve high Cpk values (e.g., ≥ 1.33) during manufacturing. Although the cost of this single part may increase, it is a necessary investment to ensure the overall product quality and market reputation, avoiding significant after-sales maintenance costs and brand damage later.

Identification and Relaxation of Non-Critical Dimensions

What are non-critical dimensions? These mainly refer to cosmetic surfaces, non-mating surfaces, and structural dimensions that merely provide support or shielding. Minor deviations in these dimensions have no substantial impact on the product's main function or user experience.

Relaxation Strategy: While meeting the most basic assembly and structural strength requirements, relax the tolerances of non-critical dimensions as much as possible. This directly brings benefits:

Reduced machining costs: Looser tolerances allow for more economical processing techniques (e.g., standard machining instead of wire EDM), improving production efficiency and reducing scrap rates.

Increased supplier flexibility: More suppliers are qualified to undertake the order, enhancing supply chain resilience and potentially reducing procurement costs through competition.

Quantitative Trade-offs Based on Cost-Tolerance Models

Tolerance optimization can be analyzed quantitatively by establishing mathematical models. The basic idea is to minimize the total cost.

Cost Composition: Total Cost = Manufacturing Cost + Quality Loss Cost.

Manufacturing Cost: Typically increases sharply as tolerances become tighter (T value decreases).

Quality Loss Cost: Includes scrap, rework, after-sales repairs, etc., and decreases as tolerances become tighter.

Optimization Goal: Find the lowest point on the total cost curve. The tolerance corresponding to this point is the economic tolerance. Modern optimization algorithms (e.g., particle swarm optimization, genetic algorithms) can help solve optimal tolerance allocation schemes under complex dimensional chains .

LEDA Practice: At LEDA, we strongly recommend conducting cost-tolerance sensitivity analysis early in the project. By building mathematical models of key dimensional chains and inputting process capability (Cpk) and cost data for each link, we can simulate the impact of different tolerance allocation schemes on the total cost, providing clients with data-driven decision support to find the design scheme with the best cost-performance ratio .

9.3 Model-Based Definition (MBD) for Digital Collaboration: Integrating PMI and Tolerance Information in 3D Models

Model-Based Definition (MBD) is a technology leading a paradigm shift in manufacturing. Its core lies in the seamless integration of 3D annotations (Product Manufacturing Information, PMI) with the 3D geometric model, creating a single, complete, and machine-readable digital source for product definition. This initiative aims to completely replace traditional 2D drawings, enabling unambiguous data flow throughout the entire process from design, process planning, and manufacturing to inspection. It breaks down information silos and lays the foundation for efficient collaboration and automation .

9.3.1 MBD and PMI: Building a Single Source of Truth for Product Definition

MBD is not merely a 3D model; it is a methodology that uses an integrated 3D solid model to carry all product definition information. This information includes not only the geometric shape but also all Product Manufacturing Information (PMI) required for manufacturing and inspection .

Core Components of PMI: PMI encompasses all non-geometric information traditionally found on 2D drawings, associating it in a structured way with specific features of the 3D model. Key components include :

Geometric Dimensioning and Tolerancing (GD&T): Includes dimensions, tolerances, datums, and geometric tolerances (e.g., flatness, position), which are critical for ensuring part function and interchangeability.

Surface Texture and Technical Requirements: Such as surface roughness, heat treatment, coatings, etc.

Notes and Identifiers: Part identifiers, general notes, and process instructions for specific areas.

Management and Material Attributes: Such as part number, revision, and material specifications.

Value of the Paradigm Shift: Compared to the traditional "3D model + 2D drawings" mode, MBD offers fundamental advantages :

Eliminates Ambiguity: The "what you see is what you get" nature of 3D models makes design intent more intuitive, significantly reducing interpretation errors across different departments (process, manufacturing, inspection).

Improves Efficiency: Designers save significant time previously spent creating and annotating 2D drawings, allowing them to focus more on the product design itself.

Foundations for Automation: Structured and semantically complete PMI serves as a direct input source for driving downstream automated processes, such as Computer-Aided Manufacturing (CAM) programming and automated Coordinate Measuring Machine (CMM) program generation .

9.3.2 MBD Implementation Specifications for Unambiguous Data Transfer

To achieve unambiguous data transfer, strict corporate specifications must be established and followed, ensuring the quality and consistency of MBD models .

Adherence to International and National Standards: PMI annotations in the model must comply with standards such as ASME Y14.41, ISO 16792, or China's GB/T 24734. These standards define rules for the display, placement, and associativity of PMI in 3D space, forming the basis for correct interpretation of data by different software and teams .

Ensuring Semantic Integrity of PMI: Annotations must not be merely visual "notes"; they must establish precise, machine-readable associations with the model's geometric features. For example, a position tolerance must be clearly associated with the corresponding hole feature and the datum reference frame. Errors like "dangling annotations" or incorrect associations to reference geometry must be avoided .

Effective PMI Organization and Management: To prevent information overload and a "porcupine" effect on complex parts, strategies like grouping and using different views should be employed. Utilizing CAD software features like layers, view orientations, and combination states helps group, display, or hide PMI information relevant to specific process or inspection steps, ensuring a clear interface where personnel only see pertinent information .

9.3.3 LEDA Collaborative Process: The Digital Closed-Loop from MBD Model to Inspection Report

At LEDA, we have deeply integrated MBD into the collaborative workflow, building a digital closed-loop system from design to inspection feedback. This process significantly enhances quality and efficiency. Its core stages are summarized in the table below :

Process Stage	Core Activities & Technical Support	Key Outputs & Value
1. Model Quality Verification	Use MBD model quality validation tools to automatically check the released design model, ensuring PMI semantic integrity, standards compliance, and absence of geometric errors.	A high-quality, "right-first-time" MBD model, preventing manufacturing deviations caused by model errors at the source.
2. Inspection Planning Generation	Inspection engineers directly import the MBD model into CMM software. Using standards like QIF, the software automatically recognizes tolerance requirements and inspection features from the PMI, automatically or semi-automatically generating inspection paths and measurement plans.	Significantly reduced inspection programming time (can reach 80%-95%), eliminating manual programming errors and interpretation 偏差 .
3. Automated Inspection Execution	The CMM executes the inspection program derived directly from the MBD model, enabling automated part measurement.	An efficient, consistent measurement process reduces human intervention and improves inspection reliability.
4. Result Analysis & Visual Feedback	Measurement data is fed back to an analysis system via a format like QIF. The system automatically compares the actual measured point cloud data with the original MBD theoretical model.	Automated generation of chromatographic deviation analysis reports: intuitively displays the magnitude and direction of deviation between actual dimensions and theoretical values across the part using color mapping .

LEDA Practice Case: In a precision component project, the LEDA team employed the full MBD collaborative process. The design model contained complete GD&T PMI, which directly drove the CMM to generate the inspection program via a QIF interface. The chromatogram generated post-inspection clearly showed an out-of-tolerance trend (displayed in red) for the flatness of a critical mounting surface near its edge. This report was immediately fed back to the design team. Investigation traced the root cause to a minor deformation induced by the fixturing strategy. By optimizing process parameters, the issue was resolved within the first production batch, avoiding a mass quality incident. This process shortened the traditional design-manufacture-inspection feedback cycle from several days to just a few hours .

9.3.4 Implementation Path and Benefits Summary

A successful implementation of MBD digital collaboration typically follows a phased strategy :

Foundation Building Phase: Establish corporate MBD standards, train the design team, and introduce basic MBD design tools and model checking software.

Process Pilot Phase: Select a typical product project to pilot the complete workflow from MBD to automated inspection, validating technical feasibility and accumulating experience.

Full Promotion and Integration Phase: Promote the matured process to more product lines, and deeply integrate the MBD-QIF data flow with corporate PLM (Product Lifecycle Management) and MES (Manufacturing Execution System) to build a complete digital thread.

The core benefits achieved by implementing MBD-based digital collaboration include :

Elimination of 2D drawing conversion, improving design efficiency.

Reduction of manufacturing errors and rework caused by misinterpretation.

Automation of inspection, significantly shortening the inspection cycle.

Establishment of a precise, rapid, data-driven quality feedback loop for closed-loop correction.

Ultimately empowering enterprises to reduce costs, enhance efficiency, and achieve quality upgrades.

Chapter 10: Design for Assembly (DFA) – Simplification is the Ultimate Sophistication

Design for Assembly (DFA) is a critical core methodology in product development. Its core philosophy is to fundamentally simplify and optimize the assembly process through product design optimization. It requires designers to systematically consider how parts can be efficiently, economically, and reliably combined into a complete product, starting from the very first sketch. Successful DFA practice can significantly reduce assembly costs, shorten cycle times, and improve quality and reliability, serving as a bridge connecting design and efficient manufacturing .

DFA and Design for Manufacturing (DFM) are complementary, together forming the cornerstone of concurrent engineering. If DFM focuses on the excellent manufacturability of individual parts, then DFA focuses on the optimal efficiency and reliability of combining all parts into a product . Its ultimate goal is to achieve the minimum number of parts, the simplest assembly steps, and the lowest assembly cost while satisfying all functional, performance, and quality requirements . Research indicates that a significant portion of the total product cost is determined by the assembly 环节, and DFA can have a decisive impact on this through design optimization .

The concept that "simplification is the ultimate sophistication" is fully embodied in DFA. It does not seek simple reduction but aims to eliminate unnecessary complexity and redundancy through profound insight and innovation, achieving an efficient, robust, and elegant state . Its value is mainly reflected on three levels:

Economic Efficiency: Reducing the number of parts and assembly actions directly lowers material, inventory management, and labor costs .

Quality and Reliability: Fewer parts and simpler interfaces mean fewer potential failure points, higher product consistency, and reliability .

Efficiency: Simplified assembly processes shorten production cycles, improve production efficiency, and create favorable conditions for automated assembly .

This chapter will systematically analyze the principles and methods of DFA, guiding you on how to integrate "thinking for assembly" into the design DNA, thereby creating products that are not only functionally excellent but also easier to produce with high efficiency and quality.

10.1 Product Architecture Optimization and DFA Indexed Assessment

The core goal of Design for Assembly (DFA) is to significantly improve assembly efficiency, reduce costs, and enhance reliability by optimizing the product architecture. Its successful implementation relies on two pillars: first, adhering to concise and efficient design principles, and second, establishing a quantitative assessment model to provide objective data support for design decisions .

10.1.1 DFA Core Principles: Part Count Minimization and Functional Integration

The cornerstone of DFA is "Entities should not be multiplied without necessity." Its most fundamental principle is to minimize the number of parts to the greatest extent and integrate multiple functions into fewer parts .

Value Manifestation: Reducing the number of parts brings comprehensive benefits. It directly reduces the complexity of procurement, warehousing, and logistics management, lowering material costs. On the manufacturing side, it simplifies assembly operations, reduces the need for tools and fixtures, shortens assembly time, and also lowers quality risks caused by multiple part interfaces and potential assembly errors . Every superfluous part is a potential failure point and a cost-adding link.

Functional Integration: Through innovative design, functions originally achieved by multiple parts are integrated into a single part. For example, using injection molding technology, multiple structural components, snaps, and guiding features can be integrally formed, replacing traditional schemes involving "multiple metal parts + screw fastening." This not only reduces the part count but also eliminates screw assembly steps, greatly improving efficiency .

The Three Criteria for Part Consolidation: When evaluating the necessity of each part, the classic three principles can be applied :

Criterion of Relative Movement: Must the part move relative to its adjacent part(s)?

Criterion of Material Necessity: Must the part be made of a material different from its adjacent part(s)?

Criterion of Service/Access: Does the part need to be separate for disassembly, maintenance, or repair?

If the answer to all three questions is "No," then the part is a prime candidate for being combined with another.

10.1.2 Quantitative Assessment Model: The DFA Index

To move beyond empirical judgment and achieve scientific decision-making, a quantitative assessment model—the DFA Index—needs to be introduced. This index aims to quantify each part's contribution to assembly time, cost, and product failure rate, providing clear data guidance and prioritization for design optimization .

1. Core Evaluation Metrics

The DFA Index typically consists of the following key metrics, which can be systematically assessed using the table below:

Evaluation Dimension	Evaluation Metric	Description & Quantification Method	Design Optimization Goal
Part Contribution	Functional Value Ratio	Assesses whether the part is essential for achieving the product's core function. Can be judged based on the "Three Criteria for Part Consolidation" above.	Eliminate non-functional parts: Remove parts that do not contribute to core functions.
Assembly Efficiency	Assembly Operation Index	Measures the time and motion complexity required to assemble the part. For example, a part requiring orientation, alignment, rotation, and application of force to snap in has a much higher index than one that simply needs placing. Part symmetry is a key factor: fully symmetrical parts require no orientation, offering the highest assembly efficiency.	Simplify assembly actions: Design symmetrical parts, add guiding features, avoid hidden assembly paths.
Quality & Reliability	Failure Risk Factor	Predicts the probability of the part failing during assembly and use. The part's inherent structural fragility, assembly difficulty, and its performance under load or environmental conditions all influence this factor.	Design for robustness: Strengthen weak points, optimize load paths, employ mistake-proofing (Poka-Yoke) design.

2. DFA Index Calculation and Application

Based on the above metrics, a comprehensive DFA Index score can be calculated for each part. A higher score indicates a greater "burden" in terms of assembly efficiency, cost, and quality, giving it a higher priority for optimization .

Formula Framework: DFA Index = f (Assembly Operation Index, Failure Risk Factor, Functional Value Ratio)

Application Process:

Create a Bill of Materials: List all parts in the product.

Score Each Part: Score each part against the above metrics.

Calculate Index: Calculate a comprehensive DFA Index for each part based on predefined weights.

Identify Improvement Points: Focus on parts with the highest DFA Index scores as the primary targets for design optimization.

Through this quantitative analysis, the design team can concentrate their efforts on the "most critical" few parts, thereby maximizing the return on resource investment . For example, the analysis might reveal that a small, asymmetrical cover plate, which is only decorative but difficult to assemble and prone to damage, contributes more to the total index than a large main structural part, making it the primary candidate for optimization.

LEDA Practice: At LEDA, we incorporate DFA indexed assessment into our collaborative design review process. By using internally developed assessment tools and databases, we can quickly provide clients with a data-driven DFA analysis report for new designs, clearly indicating areas with the greatest optimization potential, and work with clients to explore optimal solutions, such as merging multiple sheet metal parts into a single stamped and bent part through structural innovation, or replacing complex screw fastening with efficient snap-fits .

10.2 Error-Proofing Design (Poka-Yoke): Geometric Error-Proofing, Sensor Intervention, and Assembly Path Optimization

The core goal of error-proofing design (Poka-Yoke) is to create a manufacturing environment where it is "difficult to make a mistake." Its application follows a key principle: prioritize fundamental error-proofing through physical design at the product design stage; use detection-based error-proofing with technologies like sensors in the manufacturing process as a secondary strategy .

1. Geometric Error-Proofing: The Most Effective Strategy

Geometric error-proofing uses physical design features to ensure parts can only be assembled in the one correct way, fundamentally eliminating the possibility of error .

Asymmetric Design: Deliberately design parts that are theoretically symmetrical to be slightly asymmetric. For example, use positioning pins and holes of different sizes or at different pitches to ensure the part cannot fit together in the wrong orientation .

Mandatory Guidance Design: Use features like chamfers, guide slots, and unique snap-fit structures to guide parts to naturally slide or snap into the correct position during assembly. The anti-insertion design of a USB port is a classic example of mandatory guidance .

Key Principle: Error-proofing features should be as obvious as possible. If asymmetry is necessary, it should be exaggerated so that operators or equipment can perceive it without close inspection .

2. Sensor Intervention Strategy: Real-Time Verification and Error Prevention

When geometric error-proofing cannot be fully achieved, sensors are introduced at key stations for real-time verification .

Photoelectric Sensors: Used to detect the presence/absence of a part or whether it is installed correctly (e.g., detecting if a screw is fastened or a cover is snapped shut) .

Vision Sensors/Machine Vision Systems: Use cameras to identify part features, barcodes, or markings to verify product model or assembly correctness. This is the most powerful method .

Force Control Sensors: Integrated on robot end-effectors to monitor force/torque changes in real-time during assembly, preventing damage from press fits or misalignment .

Application Logic: When a sensor detects an anomaly, the system immediately triggers an alarm (audio/visual), stops the production line, or locks the current station to prevent defects from flowing to the next process .

3. Assembly Path Optimization

Errors can be reduced by optimizing the assembly sequence and process. For example, using pick-to-light systems, where lights indicate the next part to pick and its installation location, simplifies operations and prevents picking errors or omissions .

10.3 Design for Adaptability to Automated Assembly

Product design must fully consider the operational characteristics of automation equipment (e.g., robots, AGVs) to ensure they can reliably pick, handle, and assemble parts .

1. Designing for Robot Gripping

Stable Gripping Surface: Provide suction cups with a sufficiently large, flat, and smooth surface. For non-planar parts, design dedicated process tabs as pickup points .

Clear Clamping Areas: Design grooves or flanges for robotic grippers to ensure stable clamping without damaging functional surfaces. The clamping area must have sufficient rigidity to prevent deformation .

Avoid Tangling and Nesting: Part design should avoid hook-like or spring-like structures that easily interlock. For thin sheet parts, design small bumps or ribs to reduce contact area and prevent adhesion caused by oil films or vacuum suction .

2. Designing for Machine Vision

Machine vision systems are the "eyes" of automation equipment and require clear visual cues .

Fiduciary Marks: Design dedicated fiduciary marks on non-functional areas of PCBs or parts. These should be high-contrast shapes (e.g., bright copper vs. dark substrate) with a flat surface. Their size should be determined based on the camera's field of view and accuracy requirements, typically 1mm to 3mm in diameter .

Feature Contrast: When using the part's inherent structural features (e.g., holes, corners) for positioning, ensure there is sufficient color or grayscale contrast between the feature and the background for easy recognition by image processing algorithms .

Avoid Reflections and Glare: Surface design should avoid large areas of highly reflective or mirror-like finishes, which can interfere with the vision system's imaging, leading to positioning failures .

The table below summarizes key design considerations for adapting to automated assembly.

Design Consideration Dimension	Core Objective	Key Design Points
Robot Gripping	Provide stable, reliable gripping	Provide flat suction surfaces or dedicated clamping features (e.g., tabs,

Design Consideration Dimension	Core Objective	Key Design Points
Adaptability	points.	grooves). Avoid geometries prone to tangling or nesting.
Machine Vision Adaptability	Provide highly reliable, high-precision visual positioning features.	Set high-contrast fiduciary marks. Ensure features used for positioning have clear edges. Avoid large reflective surfaces.
Assembly Path & Error-Proofing	Simplify assembly actions, integrate error-proofing.	Optimize part symmetry (aim for full symmetry or significant asymmetry). Design physically guided features (e.g., chamfers, specific snap-fits).

LEDA Practice Recommendation: At LEDA, we strongly recommend involving automation and manufacturing process experts for collaborative reviews early in the design phase. Using 3D digital simulation, potential issues with robot gripping paths, the recognizability by vision systems, and the complete assembly process can be identified and resolved before physical prototyping. This significantly shortens the production ramp-up cycle .

10.4 Standardization and Modularization: Universal Design of Fasteners, Interfaces, and Functional Units

Standardization and modularization are core DFA strategies aimed at significantly reducing production complexity and cost by minimizing variety and increasing reusability. Their core value lies in transforming point-based designs into reusable, scalable system resources .

1. Establishing an Internal Standard Parts Library

Core Content: Build and mandate the use of a unified standard parts library covering fasteners (e.g., screws, snaps), connectors, common components, and modular units. This greatly reduces part types, lowering procurement, inventory, and management costs .

Implementation Path:

Unify Specifications: For example, standardize screw specifications to M3 and M5, avoiding non-standard sizes like M2.5 or M4 in designs.

Create Library & Specifications: Establish a standard parts library within CAD software, accompanied by a "Standard Parts Selection Specification" defining a preferred list.

Process Assurance: Integrate the use of the standard parts library into the design review process to ensure compliance .

2. Defining Modular Interface Specifications

Core Value: Modular design decomposes a product into independent modules by function, connected via standard interfaces. This supports parallel development and testing, facilitates repair and upgrades, and enables rapid derivation of new products based on existing modules .

Design Key Points:

Interface Definition: Clearly define mechanical (e.g., locating pin holes, slot) and electrical (e.g., connector type, pin definition) interfaces between modules to ensure compatibility.

Functional Independence: Modules should be as functionally independent as possible to reduce interdependencies and the ripple effects of changes .

Case Reference: A medical device company improved assembly efficiency by 40% and reduced repair time by 65% after modularizing its device into five functional modules .

10.5 Quantitative Assessment and Benefit Analysis of DFA

The value of DFA must be demonstrated through quantitative assessment models and real benefit data. This provides a clear basis for design decisions and showcases the financial and efficiency improvements resulting from optimization.

1. DFA Indexed Assessment Model

A typical quantitative assessment system may include the following core metrics, used to evaluate and compare the strengths and weaknesses of different design schemes during the design phase:

Assessment Dimension	Key Metric	Description & Value
Design Efficiency	Theoretical Minimum Part Count (TM)	Determines the theoretically indispensable number of parts by applying the "Three Criteria for Necessity" to each part; it is the foundation for assessing simplification potential.
	Design Efficiency (η)	Calculated as η = (3 × Theoretical Minimum Part Count TM) / Total Estimated Assembly Time. A higher value indicates a design superior in simplicity and ease of assembly.
Assembly Cost	Total Estimated Assembly Time	Calculated by summing the standard times for each assembly operation; it is the core metric for evaluating assembly cost.
	Assembly Cost	Direct labor cost 折算 based on the total assembly time.
Quality Risk	Failure Risk Factor	Predicts the probability of failure for specific structures or connection methods during assembly and use, based on historical data or simulation analysis.

2. Benefit Analysis Cases

Through DFA optimization, companies can achieve significant improvements across multiple key performance indicators. The following cases illustrate potential improvement areas:

Optimization Area	Case Description	Quantitative Benefits
Part Count Reduction	A mechanical equipment control panel was optimized using DFA, reducing the number of parts from 43 to 30 by integrating brackets and mounting plates, and replacing some screws with snap-fits.	• 30% reduction in part count • Assembly time shortened from 18 minutes to 7 minutes • 85% reduction in assembly errors
Standardization	An electronic product unified 8 types of custom screws into 2 standard	• 60% reduction in fastener

Optimization Area	Case Description	Quantitative Benefits
Benefits	specifications and adopted self-tapping screw designs to reduce nut usage.	costs • Assembly error rate reduced from 5% to 0.5%
Modularization Benefits	An aerospace component used carbon fiber composite material to replace titanium alloy and integrated 5 parts into 1.	• 65% weight reduction • 80% reduction in assembly time

By implementing standardization, modularization, and conducting rigorous quantitative assessment, DFA can deliver multiple benefits in cost, efficiency, and quality from the source. It is a powerful tool for driving continuous product design optimization.

Chapter 11: Design for Cost (DFC) – Designing for Value

Design for Cost (DFC) is a systematic product development methodology. Its core principle is the precise control and optimization of a product's total cost throughout its entire lifecycle through proactive design interventions, while ensuring user requirements for function, quality, and performance are met. It requires designers to consider cost as a key design variable, equally important as performance and reliability, from the conceptual stage, rather than treating it merely as an accounting metric after the design is finalized.

In product development, 70%-80% of the final cost is locked in during the design phase. DFC aims to reverse the traditional passive model of "design first, calculate cost later," shifting to an active strategy of "designing for cost". The goal is not simply to reduce the price, but to pursue higher efficiency in value creation. This means using scientific design decisions to eliminate unnecessary redundancy and waste, ensuring that every unit of cost invested translates into product value perceptible to the user.

The implementation of DFC permeates all stages, including material selection, process planning, structural optimization, and even supply chain collaboration. For example, in CNC machining, optimizing part geometry to reduce machining time and avoiding special tools can directly lower manufacturing costs. It is essentially a concurrent engineering practice that requires collaboration between design, manufacturing, procurement, marketing, and other departments. By breaking down functional silos, the goal is to jointly find the optimal balance between cost and performance.

This chapter will systematically analyze the core principles and methods of DFC, guiding you on how to integrate cost thinking into the design DNA from the early stages of product development, thereby creating excellent products that are both market competitive and profitable.

11.1 Full Lifecycle Cost Analysis: Establishing a Cost Baseline

Full lifecycle cost analysis requires the systematic quantification of all relevant costs from the product concept stage to its end-of-life. The core of this analysis is building a transparent cost model that clearly shows the cost composition of each stage, such as tooling, materials, processes, assembly, and inspection, providing data support for design decisions.

Cost Components:

Design & R&D Stage: Design costs (personnel compensation, software tools), patent application fees, prototyping and testing validation fees.

Production & Manufacturing Stage: Direct materials (raw materials, purchased parts), direct labor, manufacturing overhead (equipment depreciation, factory rent, energy consumption), quality costs (inspection, rework, scrap), and tooling costs (amortization).

Operation & Maintenance Stage: Logistics, after-sales repair, spare part replacement, energy consumption, and periodic maintenance costs.

End-of-Life & Disposal Stage: Dismantling, environmental treatment costs, and material recycling residual value.

Role of Establishing a Cost Baseline: This baseline serves as a fundamental benchmark for measuring cost fluctuations caused by design changes. For instance, choosing a cheaper material might reduce direct material costs, but if it leads to an increase in scrap rate or maintenance costs, the total cost might actually rise. The baseline model allows for the quantitative assessment of the real impact of different design schemes on the total cost.

11.2 The Cost Leverage Effect of Design Decisions

Decisions made during the design phase have a tremendous leverage effect on the total product cost. The famous industry "**1:10:100 Rule**" illustrates this: if the cost to fix a problem during the design stage is 1 unit, the cost to fix the same problem if found during the tooling manufacturing stage skyrockets to 10 units, and if it flows to the mass production stage, the correction cost could be as high as 100 units.

Leverage Effect Analysis: This is because the design phase determines fundamental elements that constrain all subsequent manufacturing steps, such as material selection, basic structure, and process route. Once the design is frozen, approximately 70%-80% of the final cost is locked in.

Value of Design Intervention: Therefore, minor, low-cost design optimizations early on (e.g., reducing one part, optimizing a wall thickness to avoid sink marks) can have an exponentially positive impact on later production costs. For example, avoiding mold rework through DFM optimization early in the design phase can directly save significant costs and shorten the lead time by several weeks.

11.3 Value Engineering / Value Analysis (VE/VA): Function-Cost Trade-offs and Innovative Solutions

Value Engineering / Value Analysis (VE/VA) is a systematic methodology. Its core is to seek lower-cost solutions under the premise of meeting the user's essential functions, through creative thinking and functional analysis, to achieve the highest value (**Value = Function / Cost**).

Core Process and Tools (FAST Diagram):

Function Analysis: Clearly define the basic functions (Must Have) and secondary functions (Nice to Have) of the product and each of its components.

Function Analysis System Technique (FAST Diagram): Decompose product functions into a hierarchical structure using "How-Why" logic, visually displaying the logical relationships and support paths between functions.

Creativity and Evaluation: For functions with high cost but low value, organize cross-functional teams for brainstorming to find alternative solutions (e.g., replacing screws with snap-fits, custom parts with standard parts), and evaluate the technical feasibility and cost-effectiveness of new solutions.

Innovative Solutions: VE/VA encourages challenging the status quo. For example, not all housings require CNC machining; evaluating sheet metal bending or high-precision injection molding might achieve the same function at a lower cost. A complex transmission mechanism might be able to reduce material usage while improving performance through topological optimization.

11.4 Production Volume and Process Selection: Quantitative Evaluation of Economic Batch Sizes and Process Path Optimization

Production volume is a key factor in determining the optimal manufacturing process. Different processes (e.g., 3D printing, CNC machining, injection molding, die casting) have their economic volume ranges. Choosing the wrong process can lead to a sharp increase in costs.

The table below shows the economic analysis of processes under different production volumes:

Production Volume Scale	Recommended Process	Cost Drivers & Considerations	LEDA Practice Suggestions
Prototype / Low Volume (1-100 pcs)	3D Printing (SLS/SLA/FDM), CNC Rapid Prototyping	High mold cost; piece part cost mainly from machine time and material. No mold investment needed, flexible iteration.	Prioritize machining efficiency and material suitability over extreme optimization of material usage per part.
Medium & Low Volume (100-10,000 pcs)	Vacuum Casting (RIM, Urethane Casting), Multi-cavity CNC, Soft Tooling	Use low-cost molds (e.g., silicone rubber molds, aluminum molds) for medium/low volume replication, balancing mold cost and piece part cost.	Use "multi-cavity molds" or "family molds" (producing different parts in one mold set) to improve efficiency.
High Volume (>10,000 pcs)	Hard Tooling Injection Molding, Die Casting, Stamping	High mold cost is amortized over large volume, resulting in very low piece part cost. Automation level and production cycle become key.	Design must be optimized for high-speed automated production, e.g., considering automated feeding, robotic degating.

LEDA Summary: The successful implementation of DFC relies on data-driven decisions. At LEDA, we build parametric cost models incorporating material prices, process hours, equipment rates, etc., to simulate the impact of different design choices on the full lifecycle cost in real-time. We strongly recommend initiating cost modeling and VE/VA analysis at the beginning of a project, allowing cost awareness to permeate every design decision, thereby creating truly market-competitive, high-value products.

Chapter 12: Design for Reliability and Verification (DFR/DFV) – Building Intrinsic Product Robustness

In today's highly competitive market, product reliability has evolved from a competitive advantage to a cornerstone for corporate survival and development. An unreliable product, regardless of how innovative its functions or how low its cost, will ultimately lead to high after-sales repair costs, loss of customer trust, and irreversible damage to brand reputation. True reliability is not "passively screened" through late-stage inspection and testing; rather, it is an inherent property proactively "built-in" from the very beginning of a product's life through forward-looking, systematic design philosophies and methods.

Design for Reliability (DFR) and Design for Verification (DFV) represent precisely such a systematic methodology. Their core philosophy is that **"Design Determines Reliability"**. Statistics indicate that approximately 80% of product reliability issues have their roots in the design stage. This means that design decisions—such as component selection, circuit architecture, layout and routing, and thermal management—fundamentally lock in the future failure rate and service life of a product during the drawing board phase.

DFR/DFV requires shifting our perspective from traditional "product safety" to a more comprehensive "product usage safety". It focuses not only on whether the product works under ideal conditions but also on how it can operate robustly in real-world user scenarios filled with "disturbance factors." It even considers how the product can fail safely or degrade gracefully under partial abnormal conditions, rather than suffering catastrophic failure. For example, an intelligent robotic lawnmower must not only trim grass accurately but also have predefined safety strategies to avoid accidents when encountering unexpected obstacles or experiencing temporary sensor malfunctions.

This chapter will delve into the complete system for building intrinsic product robustness. We will start with the core tool of Failure Mode Prevention (DFMEA), learning how to systematically identify and prevent potential risks. Then, we will detail design considerations for product adaptability in harsh environments, including thermal, vibration, and corrosion challenges. Finally, we will explain how to reserve interfaces and space for testing and verification through a Design Verification Plan (DVP), ensuring reliability goals are quantitatively verified, forming a closed-loop management process from design to verification. The ultimate goal is to help enterprises create high-quality products that not only meet specification sheet requirements but also withstand the tests of time and the market.

12.1 Failure Mode Prevention: The Process and Integration of DFMEA (Design Failure Mode and Effects Analysis)

DFMEA is a systematic, prevention-oriented reliability design analysis technique. It proactively identifies potential failure modes within a product during the design stage (from the initial concept or earlier), analyzes their potential consequences and impacts, and preemptively formulates optimization measures to mitigate risks at the source, thereby enhancing product quality, reliability, and safety. Its core value lies in transforming quality issues from "firefighting" to "fire prevention". Industry practice shows that effective DFMEA implementation can significantly reduce high costs associated with late-stage design changes and shorten the R&D cycle.

DFMEA Complete Process Detailed Explanation (New 6-Step Method)

Modern DFMEA follows a structured six-step process to ensure comprehensive and consistent analysis. The following elaborates on this 流程.

Step	Core Objective	Key Activities & Outputs
1. Definition & Planning	Define the analysis focus, plan resources and responsibilities.	Determine the specific object of analysis (e.g., a specific subsystem or component). Form a cross-functional team. Develop an analysis plan. Gather basic information like design drawings, technical requirements, and historical failure cases.
2. Structure Analysis	Decompose the product into manageable hierarchies, clarifying component relationships.	Break down the system into systems, subsystems, components, and sub-components using a structure tree. Clearly show physical/logical interface relationships between elements using a boundary diagram.
3. Function Analysis	Define the intended function and requirements for each structural element.	For each element from the structure analysis, define its function (verb + noun + modifier). Derive quantitative requirements (e.g., performance, durability). Form a function network or matrix.
4. Failure Analysis	Systematically derive potential failures, establishing failure chains.	Based on the function analysis, derive potential failure modes (e.g., loss of function, function degradation) for each function. Analyze their failure effects (on higher-level systems/customers) and failure causes (design weaknesses). Build the "Failure Mode - Failure Cause - Failure Effect" chain.
5. Risk Analysis	Assess failure risk, determine optimization priority.	Rate the Severity (S), Occurrence (O), and Detection (D) for each failure chain. Calculate the Risk Priority Number (RPN) or determine Action Priority (AP) to identify highest-risk items.
6. Optimization	Develop and implement measures to reduce risk.	For high-risk items, the team formulates and implements design optimization measures (e.g., change material, add redundancy). Re-rate S, O, D values to verify effectiveness. Update the DFMEA report.

Definition & Planning: This step is the foundation. Clearly define the boundaries of the analysis. Assemble a cross-functional team including engineers from design, process, quality, and materials to ensure comprehensive perspective.

Structure Analysis: Aims to visualize and layer the complex product. A structure tree decomposes the product level by level; a boundary diagram clarifies interfaces and interactions, helping identify potential interface failures.

Function Analysis: Ensures the design intent of each structural element is clear. Function descriptions should be quantitative. Higher-level functions are decomposed to lower-level elements, ensuring requirements are fully cascaded.

Failure Analysis: The core reasoning step. Systematically consider all potential failure scenarios for each function. For example, for a door handle's "open door" function, failure modes could be "fails to open" or "excessive opening force." Effects might be "inability to escape in emergency," and causes might be "return spring fatigue fracture" or "handle jamming."

Risk Analysis: Aims to quantify the risk level of each failure mode to focus resources. The new FMEA recommends using Action Priority (AP) over traditional RPN. AP uses lookup tables based on S, O, D combinations to assign High (H), Medium (M), Low (L) priority, focusing more on preventing high-severity failures (especially safety-related).

Optimization: The ultimate goal of DFMEA is to drive design improvement. For high-risk failure modes, the team must formulate and implement corrective actions. For example, for the risk of a door handle failing to deploy after impact, an action might be adding an independent mechanical redundant release mechanism. After implementation, re-assess the risk level to close the loop.

Evaluation Criteria for Severity (S), Occurrence (O), Detection (D) and Risk Prioritization (RPN) & Measure Optimization

Risk assessment is the basis for deciding improvement priority in DFMEA.

Evaluation Criteria:

Severity (S): Assesses the seriousness of the failure effect, especially on customer (end-user) safety. Rated 1-10, higher score indicates more severe effect. Failures causing injury or violating regulations typically rate 9 or 10.

Occurrence (O): Assesses the likelihood of the failure cause occurring. Rated 1-10, higher score indicates higher frequency. Based on experience with similar designs, reliability data, or engineering judgment. Using mature, reliable designs effectively lowers the Occurrence rating.

Detection (D): Assesses the ability of current design verification methods (e.g., calculation, simulation, prototype testing) to detect the failure cause or mode before mass production or customer delivery. Rated 1-10, higher score indicates greater difficulty of detection.

Risk Prioritization and Measure Optimization:

Traditional Method: Risk Priority Number (RPN). Calculated as RPN = S × O × D. Higher RPN indicates higher risk. Teams often set a threshold (e.g., 100) for taking action.

Optimization Strategies should be based on the failure chain characteristics:

Reduce Severity (S): Often most difficult, as severity is determined by the effect. May require changing the design concept or adding redundant safety mechanisms.

Reduce Occurrence (O): Most effective prevention strategy. Achieved by using mature designs, optimizing design parameters (e.g., increasing safety factors), using more reliable materials/components.

Reduce Detection (D): Improve detection capabilities. This can be done by introducing more accurate simulation analysis, adding mandatory prototype tests, or increasing the rigor of test standards.

LEDA Practice Recommendation: At LEDA, we treat DFMEA as a dynamic, continuously updated knowledge base. After each project, the DFMEA report is archived. Its analysis results and optimization measures become valuable corporate intellectual assets, providing crucial input for subsequent similar product designs, thereby enabling experience transfer and continuous design improvement. We strongly recommend integrating DFMEA reviews into key product development milestones to ensure tight coupling with design activities.

12.2 Design for Environmental Adaptability: Thermal Management, Vibration Suppression, and Corrosion Protection Design Considerations

Design for environmental adaptability is key to ensuring a product maintains functional reliability and performance stability throughout its lifecycle when facing complex and varying operating environments (e.g., temperature, vibration, corrosion). It requires designers to implement systematic protective designs from thermal, mechanical, chemical, and other dimensions proactively, rather than reactively.

Thermal Design: Heat Path Planning and Material CTE Matching

The core goal of thermal management is to control the operating temperature of critical components within their safe operating range. Statistics show about 55% of electronic device failures are directly related to excessive temperature.

Heat Path Planning: Effective thermal design starts with clear heat path planning. Heat transfer from the source (e.g., CPU, power device) to the environment follows a series "thermal resistance chain": Junction → Die/package → Case → Thermal Interface Material (TIM) → Heat sink → Ambient. The total thermal resistance determines the temperature rise ($\Delta T = P \times R\theta\_total$) for a given power. Each stage must be optimized to avoid bottlenecks. For high-power devices, using high-thermal-conductivity materials and ensuring good interfacial contact is crucial.

Coefficient of Thermal Expansion (CTE) Matching: This is critical for long-term reliability. Different materials expand/contract at different rates with temperature changes. Significant CTE mismatch generates substantial thermal stress at connections (e.g., die attach, BGA solder joints), leading to joint cracking, trace failure, or interface delamination. Solutions include:

Using CTE transition materials.

Introducing compliant connections (e.g., using underfill).

Selecting matched substrates (e.g., ceramic substrates like AlN for high-power devices).

The table below compares core properties of common thermal management materials for selection reference.

Material	Thermal Conductivity (W/m•K)	CTE (ppm/°C)	Characteristics & Application Suggestions
Copper	~400	~17	Excellent conductivity, but heavy, costly, and high CTE mismatch with chips. Suitable for local heat spreaders.
Aluminum Alloy	180-220	~23	Optimal cost-performance, lightweight, easily machined. Primary choice for heat sink bodies.
High-Thermal Graphite Sheet	1500+ (in-plane)	Substrate-dependent	Anisotropic (excellent in-plane conductivity). Used for spreading heat in compact spaces (e.g., smartphones), requires combination with metal substrate.

Material	Thermal Conductivity (W/m•K)	CTE (ppm/°C)	Characteristics & Application Suggestions
Aluminum Nitride Ceramic (AlN)	140-180	~4.5	Excellent CTE match with silicon, good electrical insulation. Ideal substrate for high-end power modules, but costly.

Vibration Design: Natural Frequency Avoidance, Damping, and Shock-Resistant Structures

Vibration and shock can cause structural fatigue, fastener loosening, part wear, or instantaneous functional failure. Effective vibration design aims to isolate or dissipate vibrational energy.

Natural Frequency Avoidance: Every structure has natural vibration frequencies (modes). Design must involve CAE modal analysis to calculate these frequencies for the product and key components, ensuring they avoid major external excitation frequencies (e.g., engine RPM, road inputs) or internal excitation frequencies (e.g., fan speed). Typically, the system's natural frequency should be designed to be >1.4 times or <0.7 times the primary excitation frequency to avoid resonance amplification.

Damping and Shock Absorption Structures: When resonance cannot be fully avoided, damping and isolation measures are needed.

Dampers: Dissipate vibrational energy as heat through internal friction in viscoelastic materials (e.g., rubber, silicone), used to suppress resonance peaks.

Isolators: Insert elastic elements (e.g., rubber mounts, wire rope isolators) between the vibration source and the equipment to block energy transmission.

Shock Absorption Structures: For impact (short, severe acceleration), design energy-absorbing structures like internal foam, metal plastic hinges, or specialized packaging, which absorb energy through material/structural deformation.

Corrosion Protection Design: Material Selection, Surface Treatment, and Sealing Design

Corrosion weakens structural strength, destroys electrical connections, and causes product failure. Corrosion protection requires a tiered strategy based on the expected environment (e.g., salt spray, acid/alkali atmosphere, humidity).

Material Selection: The first line of defense. For harsh environments (e.g., marine, chemical), prioritize inherently corrosion-resistant materials like stainless steel (SUS304/316), titanium alloys, aluminum alloys (require surface treatment), and high-performance engineering plastics (e.g., PPS, PEEK).

Surface Treatment Processes: Adding a protective layer to the substrate is cost-effective.

Metal Parts: Processes include electroplating (e.g., zinc, nickel/chrome), electroless plating (e.g., Ni-P alloy), Dacromet (chromate-free zinc-aluminum coating), or anodizing (for aluminum).

Sealing Design: Preventing corrosive agents from entering the product interior is the most efficient method. The key is blocking ingress paths.

Static Sealing: Use gaskets (silicone, fluoroelastomer), sealants (polyurethane, silicone), or potting compounds (epoxy, silicone) to seal housing seams and cable entries.

Condensation Prevention: For enclosed equipment prone to internal condensation due to temperature differences, include desiccants inside the housing or design breather valves that equalize pressure while blocking moisture.

LEDA Summary: Design for environmental adaptability is a systematic endeavor where thermal, vibration, corrosion, and other factors are often coupled (e.g., vibration can compromise seals, accelerating corrosion; high temperatures accelerate material aging, affecting damper performance). At LEDA, we utilize multi-physics simulation and accelerated life testing to simulate and verify product performance under real-world conditions during the design phase, enabling co-optimization to ensure products possess the robustness to withstand harsh environments from the inside out.

12.3 Design Verification Plan and Report (DVP&R): Designing for Testability

The Design Verification Plan and Report (DVP&R) is a core management tool and the "acceptance benchmark" for ensuring the successful transition of a product from design blueprint to a reliable physical entity. It is a living document whose core value lies in translating abstract, dispersed design requirements and reliability goals into a set of specific, executable, and traceable test verification procedures, ultimately demonstrating through a formal report that the design indeed meets all intended objectives. In essence, the DVP&R answers three key questions: "What do we verify? How do we verify it? What constitutes a pass?" .

12.3.1 Developing a Comprehensive DVP&R: Ensuring Complete Coverage and Efficient Execution

A rigorous DVP&R is not merely a checklist of test tasks but also a basis for project management. Its development should follow a systematic process and be closely integrated with design activities .

1. DVP&R Development Process: From Requirements to Plan

Input Design Requirements and Specifications: The source of all verification activities is the product design objectives, including customer standards, international/national standards (e.g., ISO, GB/T), corporate standards, and regulatory requirements. These documents define the performance, reliability, and safety indicators the product must achieve .

Integrate FMEA Outputs: The high-risk failure modes identified by the Design FMEA (Failure Mode and Effects Analysis) are the most critical input for developing targeted test items. A key task of the DVP&R is to verify, through specific tests, whether the design measures implemented to address these failure modes are effective .

Decomposition and Integration (Layered Verification): Complex systems require a layered verification strategy. Verification tasks start at the vehicle level, are progressively broken down to the system level, subsystem level, and finally to the component level. This ensures that interfaces and functions at all levels are fully verified, while also guaranteeing testing efficiency and focus .

Define Plan Details: For each test item, clearly define its method, standards, sample requirements, resource allocation, and schedule, forming an executable plan .

2. Core Components of a DVP&R

A complete DVP&R document typically consists of a header and a body section, with core components as shown in the table below :

Component Category	Specific Item	Description & Requirement
Header Information	Project/Product Information	Clearly identify the product code, name, and version under verification.
	Project Team Members	List key responsible persons and their contact information for design, testing, quality, project management, etc.
	Document Version Number	Ensure the team always uses the latest controlled version.
Core Body Elements	Test Item Number	Unique identifier for tracking and management.
	Test Item Description	Clearly describe the test content, e.g., "High-Temperature Operating Life Test".
	Test Standard/Reference	Specify the test standard followed, e.g., "ISO 16750-4".
	Test Method	Describe the test setup, steps, and conditions in detail.
	Acceptance Criteria	Crucial. Defines the pass/fail boundaries for test results; must be quantifiable and judgeable.
	Sample Quantity & Type	Specify the number of samples required and their status (e.g., prototype, production part).
	Test Phase	Indicate whether it is Design Verification (DV) or Production Verification (PV).
	Reliability Requirement	For life tests, specify the target (e.g., "R90C90" meaning 90% reliability with 90% confidence).
	Responsible Person/Department	Clearly identify the entity responsible for test execution and result confirmation.
	Planned Cycle & Results	Schedule timeline, and a field for recording actual test results.

3. Test Flow Optimization: To improve efficiency and save samples, product engineers create test flow charts based on the DVP&R, logically grouping and sequencing related test items. For example, performing non-destructive performance tests first, followed by destructive environmental or life tests .

12.3.2 Designing for Testability: Reserving Physical Interfaces and Space

Excellent DFR design considers not only product function but also proactively considers how to facilitate verification. Reserving interfaces and space for test points and sensors during the product design stage can significantly enhance the accuracy, efficiency, and coverage of verification work .

1. Test Pad Design

For printed circuit board assemblies (PCBA), test points are the "windows" connecting internal signals to external test equipment, primarily used for In-Circuit Test (ICT), functional test, etc.

Layout Principles: Test points should be evenly distributed on the PCB and preferably located on the solder side (BOTTOM side) to avoid interference from top-side components with test probes and to simplify test fixture design .

Size and Spacing: The pad diameter for test points is recommended to be no less than 0.8mm, ideally between 0.8mm and 1.0mm. The center-to-center distance between test points and the distance from test points to tall components (e.g., capacitors, inductors) must provide sufficient clearance (typically ≥ 2mm) based on the selected probe size (e.g., 50mil, 75mil, 100mil) to prevent probe shorting and reduce fixture stress .

Accessibility: Ensure test point surfaces are clean, free of coatings (e.g., solder mask) or labels, to guarantee reliable contact with the probes .

2. Sensor and Measurement Interface Reservation

For complete machines or mechanical systems, physical interfaces need to be reserved for performance monitoring and reliability testing.

Mechanical Interfaces: Design standard threaded holes or mounting surfaces on key components (e.g., structural parts requiring vibration or temperature monitoring) to facilitate the attachment of accelerometers, thermocouples, etc.

Cable Routing: Reserve cable pass-through holes or cable tie points to ensure sensor cables can be routed neatly, avoiding strain or damage during testing, and preventing cables from introducing additional vibration or electromagnetic interference.

Functional Interfaces: For example, to facilitate EMC (Electromagnetic Compatibility) testing, consider integrating mounting points for antenna coupling boards or injection points for current probes during the design phase.

LEDA Practice Summary: At LEDA, we regard the DVP&R as the quality blueprint for product development. We strongly recommend reviewing the DVP&R and test interface design simultaneously during the design review phase. Through this forward-thinking approach of "designing for verifiability," we can not only avoid late-stage design changes caused by an inability to test or insufficient testing but also build a solid bridge connecting design, manufacturing, and quality, ensuring that the product delivered to the customer is fully verified and highly reliable. When test failures occur, the DVP&R also serves as the basis for root cause analysis and corrective action development, driving the continuous optimization of the test plan itself .

Chapter 13: DFM Collaborative Process and Organizational Support – The Bridge from Concept to Practice

After systematically exploring topics from specific processes (such as injection molding, die casting, sheet metal, 3D printing) to universal design guidelines (like tolerance analysis, Design for Assembly/Cost, and Design for Reliability), a core question emerges: How can we ensure that these valuable DFM knowledge, principles, and standards are consistently, reliably, and efficiently implemented in a real product development project, rather than remaining merely theoretical or dependent on the individual experience and initiative of engineers?

The DFM collaborative process and organizational support discussed in this chapter aim to answer this very question. Their purpose is to build a solid bridge connecting "DFM concepts" with "excellence in manufacturing practice." If the previous chapters provided the "tactics and arsenal" needed to win a battle, then this chapter focuses on establishing a "command system, logistical support, and operational regulations" that ensures victory in every engagement. This system elevates DFM from a specialized technique to a manageable, replicable, and optimizable core organizational capability.

Practice proves that without systematic process and organizational support, DFM easily falls into the dilemma of "easier said than done." Design teams might overlook manufacturability reviews under project schedule pressure; process teams, if involved too late, might only be able to reactively address issues in finalized designs; information silos between different functions can lead to distortion or loss of design intent. These challenges can cause well-intentioned designs to face manufacturing difficulties, potentially leading to cost overruns, delays, or quality risks. This highlights the critical importance of systematic control at the design source .

Therefore, this chapter moves beyond technical details to explore the management philosophy and operational mechanisms for achieving efficient DFM. We will delve into how to build cross-functional collaborative teams, defining their responsibilities and involvement points at various stages of product development; how to develop and maintain a dynamic set of corporate DFM standards and checklists, making them the common language and action guide for design and manufacturing teams; how to leverage digital collaborative platforms (e.g., PLM systems) to institutionalize and streamline DFM activities, ensuring traceability of issues and accumulation of experience; and finally, how to internalize DFM from a set of requirements into the organizational DNA striving for manufacturing excellence through knowledge management and cultural cultivation.

LEDA Perspective: At LEDA, we regard the DFM collaborative process as a core link in co-creating value with our clients. We firmly believe that a mature and reliable DFM collaborative process and organizational support system is the cornerstone for ensuring that innovative client designs are efficiently and high-quality transformed into successful market products.

13.1 Establishing a Cross-Functional Team (CFT): The Early Involvement Mechanism for Design, Manufacturing, Procurement, and Quality

The Cross-Functional Team (CFT) is the organizational cornerstone for implementing DFM. Its core goal is to break down departmental barriers, ensuring that product designs fully incorporate constraints and optimization suggestions from manufacturing, procurement, quality, and other aspects right from the design stage. This early involvement, starting from the concept phase, effectively prevents issues at the source, enhances product manufacturability, reduces costs, and speeds time-to-market .

13.1.1 Team Composition and Role Responsibilities

An effective DFM-CFT should consist of representatives from various departments to ensure all key perspectives are covered. The team leader (or process owner) is typically the project manager or a senior engineering lead with overall responsibility for the product's success. Their duty is to set common goals, drive the process, and make decisions. The core roles and their responsibilities are outlined in the table below:

Role	Core Responsibilities in DFM Review
Design Engineer	Explains design intent, leads design modifications, ensures product function and performance; optimizes the design based on manufacturability and quality feasibility suggestions (e.g., adjusting structures, tolerances).
Manufacturing/Process Engineer	Evaluates the manufacturability and assembly efficiency of the design proposal; provides information on factory equipment capabilities and process limitations; suggests ways to simplify assembly and optimize process routes.
Procurement Engineer	Assesses the availability, supplier resources, procurement lead times, and costs of materials, standard parts, and outsourced components; recommends alternative standardized components to reduce procurement risks and costs.
Quality Engineer	Evaluates design risks based on historical quality issues and standards (e.g., ISO), ensuring the product meets quality and reliability requirements; leads or participates in formulating inspection plans and acceptance criteria.
Supplier Representative	(Early involvement) Provides constraints and cost-optimization suggestions related to the manufacturing capabilities for key components or processes (e.g., precision casting, special surface treatment), avoiding late-stage design iterations.
Project Manager	Organizes DFM review meetings, tracks the implementation of optimization measures, ensures project progress according to plan, and coordinates resources.

LEDA Practice: At LEDA, we strongly recommend inviting process experts from key suppliers to join the CFT early. For example, in consumer electronics plastic part development, the early involvement of mold suppliers can provide invaluable experience regarding gate location and ejector pin layout, significantly improving the success rate of trial molds .

13.1.2 Involvement Points and Collaborative Mechanisms

The key to a successful CFT lies in its **early and continuous involvement** at critical milestones of the development process, rather than conducting a "post-design review" after the design is finalized. The following points clarify the involvement focus and collaborative rhythm for each role across product development stages :

Concept Phase: The CFT reviews product concepts for overall manufacturability, procurement feasibility of new materials/technologies, and potential quality risks.

Design and Development Phase: The team conducts regular DFM reviews synchronized with design milestones. Manufacturing engineers provide feedback on design details, procurement assesses component availability, and quality engineers help define critical-to-quality characteristics.

Prototyping and Validation Phase: The CFT analyzes issues found during prototyping. Manufacturing and quality engineers are crucial in validating the proposed manufacturing processes and inspection methods.

Production Ramp-up: The team supports the transition to mass production, addressing any lingering manufacturability issues and facilitating knowledge transfer.

This integrated approach ensures that DFM considerations are woven throughout the product lifecycle, bridging the gap between design intent and manufacturing reality .

13.2 Standardized DFM Review Process: Inputs, Checklists, and Output Templates

Establishing a standardized DFM review process is key to ensuring consistent, efficient, and traceable review outcomes. This process clearly defines the inputs, activities (checks), and outputs for each review.

13.2.1 Review Inputs

To ensure review effectiveness, the following materials must be prepared and distributed (recommended at least 1-3 working days in advance) to all CFT members for pre-review before the meeting:

Design Data: Latest 3D CAD models, 2D engineering drawings (with complete GD&T), Bill of Materials (BOM), and technical requirement specifications.

Manufacturing Resource Information: Information on existing factory equipment capabilities (e.g., machining accuracy, maximum processing dimensions), standard process libraries, etc.

DFM Checklist: A customized checklist for the specific product type (e.g., sheet metal, plastic).

Historical Data: Summary of historical quality issues for similar products, failure analysis reports, and competitor teardown analyses (where applicable).

13.2.2 Core Activity: DFM Checklist Example

The DFM checklist is a tool that transforms the design guidelines from previous process-specific chapters into actionable, assessable quantitative or qualitative checkpoints. The framework below is an example of a standard LEDA DFM checklist.

LEDA DFM Checklist Example (General Section)

Category	Check Item ID	Check Content & Design Guideline	Score (Example)	Severity
General Manufacturability	GEN-01	Is the part count minimized? Has the necessity of each part been evaluated using the three principles (relative movement, different	Pass / Partially Pass / Fail	High

Category	Check Item ID	Check Content & Design Guideline	Score (Example)	Severity
		material, service requirement)?		
	GEN-02	Is the usage rate of standard parts (e.g., screws, bearings) 85% or higher?	Pass / Partially Pass / Fail	Medium
Sheet Metal Design	SHEET-01	Is the bend radius greater than the material's minimum allowable bend radius (e.g., for SPCC, t=1.5mm, R_inner ≥ 1.0mm)?	Pass / Fail	High
	SHEET-02	Is the hole edge distance ≥ 2 times the material thickness to avoid deformation?	Pass / Fail	Medium
Injection Molding Design	PLA-01	Is wall thickness uniform? Are there potential sink marks or voids due to sudden thickness changes?	Pass / Fail	High
	PLA-02	Is sufficient draft angle provided (typically cavity ≥ 0.5°, core ≥ 0.75°)?	Pass / Fail	High
Assembly Design	ASSY-01	Are mistake-proofing geometric features (e.g., asymmetric locating pins) used to prevent assembly errors?	Pass / Fail	Medium
	ASSY-02	Are guiding features and gripping surfaces reserved for automated assembly?	Pass / Partially Pass / Fail	Medium

LEDA Practice: Our DFM checklist is a dynamic, living document. After each project, newly discovered design pitfalls or optimization points are updated into the checklist, forming part of the organization's accumulated knowledge.

13.2.3 Review Outputs

After the review meeting, the following outputs should be generated and distributed promptly (recommended within 24 hours) to achieve closed-loop management:

DFM Review Report: Summarizes the basic meeting information, overall conclusion (Pass / Conditional Pass / Fail), and key findings.

Issues and Actions Tracking Log: This is the most critical output. It should clearly list each item for optimization, the responsible person, and the planned completion date, and be tracked in the PLM system until closure.

Updated Design File List: Records all drawings and models requiring updates due to DFM optimization.

13.3 Application of a Digital Collaboration Platform Based on PLM

A Product Lifecycle Management (PLM) system is the core enabler for digitizing, visualizing, and tracing the DFM collaborative process. By establishing a single source of truth, it effectively solves the "data silo" problem, ensuring all members collaborate based on the same data version.

13.3.1 Unified Management of Design/Process Data

The core value of PLM lies in building a centralized digital repository that applies version control to all product data (3D models, drawings, BOMs, technical documents). This ensures data consistency from design to manufacturing, preventing errors caused by version confusion.

13.3.2 Online Review and Closed-Loop Issue Tracking

PLM systems support online design reviews and visualization. Reviewers can directly annotate and comment on 3D models. Feedback is automatically linked to specific design features, generating issue tracking tickets assigned to responsible persons with deadlines, with status updated in real-time. This forms a complete "Identify - Assign - Resolve - Verify - Close" closed loop.

13.3.3 Institutionalized Change Management (ECN) Process

Any design modification must go through the Engineering Change Notice (ECN) process initiated within the PLM system. The ECN process defines the complete path for change request, impact analysis (involving design, process, procurement, cost), approval, execution, and release, ensuring all changes are controlled and traceable.

13.3.4 Integration with Upstream/Downstream Systems (ERP/MES)

Deep integration of PLM with systems like ERP and MES enables seamless data flow. For example, the engineering BOM (eBOM) finalized in PLM can be accurately converted to a manufacturing BOM (mBOM) and transferred to the ERP system, guiding production and procurement, preventing manual transcription errors at the source.

LEDA Practice: At LEDA, we template and automate the DFM checklist process via the PLM system. When a designer submits a model for review, the system automatically triggers the review workflow and notifies CFT members. All review records, issues, and change history are archived in PLM, forming a complete product digital twin, providing a data foundation for quality traceability and knowledge reuse.

13.3.5 Knowledge Management: Continuous Accumulation Mechanism for DFM Case Library, Best Practices, and Lessons Learned

The true value of DFM lies in its ability for continuous improvement and inheritance. The goal of knowledge management is to convert tacit experience into explicit, reusable organizational assets, avoiding repeated mistakes, and accelerating the growth of new employees.

13.4.1 DFM Case Library Development

Establish a corporate DFM case library. Each case should include:

Problem Description: Clearly state the DFM issue in the initial design (e.g., "A sheet metal part's bend R-angle was too small, causing cracking").

Background Information: The product, material, and process type involved.

Root Cause Analysis: Analyze the cause of the problem from design and manufacturing perspectives.

Optimization Solution: The final improvement implemented (e.g., "Increased R-angle from 0.5t to 1.0t and added a relief notch").

Quantified Benefits: Specific gains from the optimization, such as cost reduction percentage, defect rate decrease, or assembly time savings.

13.4.2 Best Practices Extraction and Standardization

Regularly extract and formalize design guidelines and standard operating procedures from both successful projects and issues. For example:

Design Guideline Cards: Create concise reference cards for common DFM rules (e.g., "Plastic rib thickness should be 50%-60% of the main wall thickness") and integrate them into the CAD environment.

Standard Part Libraries and Design Templates: Maintain a library of DFM-verified standard parts and 3D templates for common modules in the PLM system. New designs can directly call these, ensuring compliance from the start.

13.4.3 Lessons Learned Retrospectives and Sharing Mechanism

Hold formal retrospective meetings after key project milestones (e.g., trial production completion, mass production handover). Focus on:

What DFM practices worked well in this project? Extract them as best practices.

What DFM problems were encountered in this project? Conduct in-depth root cause analysis and update the case library and checklist.

How can we improve our DFM process? Optimize collaboration rules and tools.

By establishing cross-functional teams, standardizing review processes, leveraging digital platforms, and institutionalizing knowledge management, enterprises can transform DFM from a technique reliant on individual experience into a systemic engineering capability that is replicable, cumulative, and continuously improvable. This ensures the "do it right the first time" philosophy is embedded into the DNA of every product.

Part 4: Industry Applications and Case Studies

Chapter 14: Industry-Specific DFM Quick Reference Guide

This industry-specific DFM quick reference guide aims to provide the most core and unique considerations for product design across different fields, helping you anchor key directions during the initial design phase.

14.1: Lightweighting, High Strength, Extreme Environments

Aerospace products pursue absolute reliability under extreme conditions. The core of their DFM lies in the ultimate balance between lightweighting, ultra-high strength/stiffness, and extreme environment adaptability.

Lightweight Design: Topology optimization and hollow/lattice structures are widely used to remove redundant material while ensuring strength. Materials with high strength-to-weight ratios are prioritized, such as titanium alloys, advanced aluminum alloys, and carbon fiber composites.

Materials and Processes: Materials must possess high specific strength, high specific stiffness, and excellent resistance to high/low temperatures, radiation, and corrosion. Critical structural components often use **integral machining** to reduce the number of fasteners and weight, thereby enhancing reliability.

Extreme Environment Adaptability:** Significant thermal expansion and contraction must be considered, avoiding thermal stress concentration through material matching and structural design. Electronic systems require sufficient thermal management and sealing design to cope with high/low-temperature cycles and vacuum or high-pressure environments.

Reliability and Redundancy: Adhere to the **"fail-safe"** principle and employ redundant design. The Worst-Case (WC) method must be applied for tolerance analysis of critical dimensions to ensure absolute safety. Key material and process parameters must be strictly documented to ensure full lifecycle traceability.

Maintainability: Designs must facilitate inspection and repair under harsh conditions, such as by setting up easily accessible test points.

14.2: Medical Devices: Biocompatibility, Sterility, Traceability

The core of DFM for medical devices is safety and effectiveness, focusing highly on biocompatibility, sterility assurance, and full-process traceability.

Biocompatibility: Materials that contact or are implanted in the human body must comply with biocompatibility standards such as ISO 10993. Surfaces should smooth and free of marks, avoiding sharp edges. The use of allergenic or toxic materials is strictly prohibited.

Sterility Assurance and Cleanability: Designs must be easy to clean, disinfect, or sterilize thoroughly (e.g., resistant to high temperature/pressure, ethylene oxide, or radiation). Avoid internal dead ends, blind holes, narrow gaps, and other areas difficult to clean. Consider disposable designs to prevent cross-contamination.

Reliability: For life-support equipment, reliability is the primary requirement. Strict Failure Mode and Effects Analysis (FMEA) is necessary, should be adopted for critical functional components.

Ergonomics: Designs must conform to the operating habits of medical professionals and the physiological structure of patients, ensuring ease of operation, comfort, and error prevention.

Traceability: Products must have a Unique Device Identification (UDI) to ensure full traceability from raw materials, production, sterilization to the end patient.

14.3: Automotive: Safety, Durability, Mass Production

The core of DFM in the automotive industry is achieving consistency and cost control at mass production scales (millions of units) while ensuring safety and durability.

Safety: As a bottom line, functional safety standards like ISO 26262 must be followed, including ASIL rating assessment and corresponding design. Structural design must meet crash regulations, optimizing energy-absorbing structures through crash simulation.

Durability and Reliability: Validation through accelerated life tests such as vibration, shock, thermal cycling, and salt spray corrosion is required. Thermal management design is crucial for high thermal load components like powertrains.

Corrosion Resistance and Weathering Resistance: For corrosion-prone areas like chassis and body, protective systems such as galvanized steel, aluminum alloys, or electrophoretic coating/spraying are needed. Exterior part materials must resist UV aging.

Mass Production Optimization: Designs must be extremely simplified, reducing part counts and employing structures friendly to automated assembly (e.g., snaps instead of screws). Tolerance design should be based on statistical methods (RSS) to balance precision and cost.

Modular Design: Modular and platform-based design is encouraged to reduce SKUs, lower supply chain complexity, and increase production flexibility.

14.4: Consumer Electronics: Aesthetics, Thermal Management, Compactness

The core of DFM for consumer electronics is achieving a high degree of integration among aesthetics, performance, and manufacturability within extremely compact spaces.

Compact Integration and Stacking: Utilize 3D stacking technologies like HDI for motherboards and System-in-Package (SiP). Maximize the use of three-dimensional space for high-density component layout.

Aesthetics and Texture: Appearance parts (e.g., housings) strive for flawless surfaces, avoiding sink marks, weld lines, and flow marks. Surface treatments like gloss, matte, and texturing (texture painting) are widely used. CMF (Color, Material, Finishing) design is key.

Thermal Management: Heat dissipation within limited space is a major challenge. Carefully plan heat paths, comprehensively utilizing thermal interface materials, vapor chambers, graphite sheets, heat pipes, or vapor chambers.

Manufacturability and Cost: Aimed at mass production (hundreds of millions), designs must be optimized for highly efficient automated assembly. Fastening techniques like snaps and ultrasonic welding are heavily used. Extreme cost sensitivity necessitates deep Value Engineering (VE) analysis.

14.5: Industrial/Robotics: Motion Accuracy, Rigidity, Durability

The core of DFM for industrial equipment and robots is ensuring motion accuracy, structural rigidity, and durability under long-term, heavy-duty conditions.

Motion Accuracy and Stability: The base structure must possess high rigidity to suppress vibration and ensure positioning accuracy. Dynamic balancing design is needed for moving parts. Precision transmission systems must avoid resonance points.

Structural Rigidity: Enhance rigidity through reasonable rib and plate layout and closed-section design. Key areas like bearing seats and guide rail mounting surfaces must have sufficient support strength and* **Durability and Maintainability:** For key wear parts like bearings, gears, and guide rails, designs must facilitate inspection, adjustment, and replacement. Provide centralized lubrication points or automatic lubrication interfaces. Consider dust and chip prevention design.

Modularization and Interface Standardization: Modular design is recommended for easier product family development. Mechanical (e.g., flanges) and electrical (e.g., connectors) interfaces should be standardized to improve interchangeability.

Conclusion: Collaborating with LEDA to Transform DFM into a Sustainable Competitive Advantage

As you reach the end of this guide, you have deeply explored the complete DFM knowledge system we have systematically outlined for you—from specific processes to collaborative workflows. The 初衷 of this guide is not merely to provide an extensive list of technical parameters, but to convey a core philosophy: in the challenging expedition of hardware innovation, excellent products are not just "designed," but are crucially "manufactured." The true decisive power stems from integrating design and manufacturing deeply from the very beginning, forging the genes of product excellence at the source.

I. Beyond a Reference Book: A Collaborative Guide from Knowledge to Action

This guide crystallizes the nearly two decades of engineering wisdom and manufacturing experience of the LEDA team. Its value extends far beyond that of a reference book; it embodies our commitment to being your "Design for Manufacturability co-creator." We deeply understand that a design perfect in simulation might fail in the market due to manufacturing inefficiencies, and a prototype excelling in the lab might stumble during mass production ramp-up due to minor process variations. Therefore, the essence of the **"Precision by Design"** we advocate is to "build-in" precision and reliability into the product blueprint through forward-looking design and collaboration, rather than relying on late-stage inspection and correction.

II. The Core Value of DFM: A Trio of Efficiency, Cost, and Competitiveness

By systematically applying the principles elaborated in this guide, you will be able to effectively navigate the complexity of product development and achieve multi-dimensional value enhancement:

Accelerate Innovation Time-to-Market: Significantly shorten product development cycles by reducing design iterations and late-stage modifications, helping you seize market opportunities faster.

Optimize Total Lifecycle Cost: Achieve savings not only in material and manufacturing costs but also substantially reduce after-sales maintenance and quality risk costs by enhancing quality and reliability.

Build the Foundation of Quality and Reliability: Resolve potential issues during the design phase, enhancing product robustness at the source and building brand reputation trusted by the market.

Enhance Supply Chain Resilience: Simplify supply chain management and improve flexibility in responding to market changes through standardized and modular design.

III. Partnering with LEDA: Co-creating the Future of Intelligent Manufacturing

The completion of this guide marks the starting point of our in-depth collaboration. What LEDA Intelligent offers is more than manufacturing capability; it is a collaborative platform integrating expert wisdom, digital tools, and efficient processes. forward to engaging in your next project by:

Acting as your extended engineering team: Getting involved at the concept stage, providing manufacturability analysis based on rich experience and data.

Providing end-to-end digital collaboration: Utilizing advanced platforms like PLM to enable seamless data flow and closed-loop management from design and simulation to production.

Continuous optimization and co-creation: Working with you to review each project, feeding new experiences back into our DFM knowledge base and processes for mutual growth in capability.

Finally, we sincerely thank you for your interest in this guide. LEDA Intelligent remains your trusted partner. We look forward to working hand-in-hand with you to efficiently and precisely translate excellent designs into successful products in the competitive market, jointly defining a new future for intelligent manufacturing.