

DRAWING SECTION VIEW SOLIDWORKS SAVOI

MASTERING THE DRAWING SECTION VIEW IN SOLIDWORKS FOR SAVOI DESIGNS

DRAWING SECTION VIEW SOLIDWORKS SAVOI IS A FUNDAMENTAL SKILL FOR ANYONE INVOLVED IN ENGINEERING, DESIGN, OR MANUFACTURING, PARTICULARLY WHEN WORKING WITH COMPLEX PARTS AND ASSEMBLIES. THIS POWERFUL TOOL WITHIN SOLIDWORKS ALLOWS FOR CLEAR VISUALIZATION OF INTERNAL FEATURES, HIDDEN DETAILS, AND INTRICATE GEOMETRIES THAT WOULD OTHERWISE BE IMPOSSIBLE TO DISCERN FROM STANDARD ORTHOGRAPHIC PROJECTIONS. UNDERSTANDING HOW TO EFFECTIVELY CREATE AND MANIPULATE SECTION VIEWS IS CRUCIAL FOR GENERATING ACCURATE TECHNICAL DRAWINGS, FACILITATING COMMUNICATION BETWEEN DESIGN AND MANUFACTURING TEAMS, AND ENSURING THE INTEGRITY OF FABRICATED COMPONENTS. THIS COMPREHENSIVE GUIDE WILL DELVE INTO THE INTRICACIES OF GENERATING SECTION VIEWS IN SOLIDWORKS, FOCUSING ON APPLICATIONS RELEVANT TO SAVOI'S DESIGN STANDARDS AND COMMON MANUFACTURING PRACTICES. WE WILL EXPLORE THE DIFFERENT TYPES OF SECTION VIEWS, THE PROCESS OF CREATING THEM, ADVANCED CUSTOMIZATION OPTIONS, AND BEST PRACTICES FOR OPTIMIZING THEIR CLARITY AND UTILITY.

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UNDERSTANDING THE PURPOSE OF SECTION VIEWS IN SOLIDWORKS DRAWINGS

SECTION VIEWS ARE INDISPENSABLE IN TECHNICAL DRAWINGS AS THEY REVEAL THE INTERNAL STRUCTURE OF AN OBJECT. IN THE CONTEXT OF SOLIDWORKS, A SECTION VIEW EFFECTIVELY "CUTS" THROUGH A MODEL TO EXPOSE HIDDEN FEATURES SUCH AS HOLES, POCKETS, RIBS, AND INTERNAL CHAMBERS. THIS LEVEL OF DETAIL IS CRITICAL FOR PRECISE MANUFACTURING AND ASSEMBLY INSTRUCTIONS, ESPECIALLY FOR COMPONENTS DESIGNED FOR DEMANDING APPLICATIONS LIKE THOSE OFTEN FOUND IN SAVOI'S PRODUCT LINES.

WITHOUT SECTION VIEWS, INTERPRETING COMPLEX PARTS WOULD BE SIGNIFICANTLY MORE CHALLENGING, LEADING TO POTENTIAL ERRORS IN PRODUCTION. THEY BRIDGE THE GAP BETWEEN THE 3D MODEL AND THE 2D MANUFACTURING BLUEPRINT, ENSURING THAT ALL NECESSARY INFORMATION IS CONVEYED CLEARLY AND UNAMBIGUOUSLY. FOR ENGINEERS AND DESIGNERS WORKING ON SAVOI PROJECTS, MASTERING THIS FEATURE TRANSLATES DIRECTLY INTO IMPROVED DESIGN VALIDATION AND REDUCED MANUFACTURING LEAD TIMES.

WHY SECTION VIEWS ARE ESSENTIAL FOR SAVOI PROJECTS

SAVOI'S ENGINEERING PROJECTS OFTEN INVOLVE INTRICATE MECHANICAL DESIGNS WITH TIGHT TOLERANCES AND SPECIFIC FUNCTIONAL REQUIREMENTS. SECTION VIEWS ARE PARAMOUNT IN THESE SCENARIOS FOR SEVERAL KEY REASONS:

- **REVEALING INTERNAL FEATURES:** CRUCIAL FOR UNDERSTANDING THE DEPTH AND SHAPE OF INTERNAL FEATURES LIKE THREADS, COUNTERBORES, AND COOLING CHANNELS.
- **ILLUSTRATING ASSEMBLY RELATIONSHIPS:** ESSENTIAL FOR VISUALIZING HOW COMPONENTS FIT TOGETHER INTERNALLY, IDENTIFYING POTENTIAL INTERFERENCES, AND CLARIFYING ASSEMBLY PROCEDURES.
- **DIMENSIONING INTERNAL GEOMETRY:** ALLOWS FOR ACCURATE DIMENSIONING OF FEATURES THAT WOULD BE OBSCURED IN STANDARD VIEWS, SUCH AS THE WALL THICKNESS OF A HOUSING OR THE DIAMETER OF AN INTERNAL BORE.
- **QUALITY CONTROL AND INSPECTION:** PROVIDES A CLEAR REFERENCE FOR QUALITY CONTROL PERSONNEL TO VERIFY CRITICAL INTERNAL DIMENSIONS AND FEATURES DURING INSPECTION.
- **FACILITATING MANUFACTURING:** GUIDES MACHINISTS AND FABRICATORS IN UNDERSTANDING THE TOOLING PATHS, DEPTHS OF CUTS, AND OVERALL INTERNAL GEOMETRY TO BE PRODUCED.

CREATING STANDARD SECTION VIEWS IN SOLIDWORKS

SOLIDWORKS OFFERS INTUITIVE TOOLS TO CREATE VARIOUS TYPES OF SECTION VIEWS. THE MOST COMMON ARE THE STANDARD SECTION VIEWS, WHICH ARE BASED ON CUTTING PLANES ALIGNED WITH THE MODEL'S PRIMARY AXES.

THE BASIC SECTION VIEW CREATION PROCESS

TO CREATE A STANDARD SECTION VIEW, YOU TYPICALLY START BY SELECTING THE "SECTION VIEW" COMMAND FROM THE "DRAWING" OR "VIEW LAYOUT" TAB. YOU THEN CHOOSE THE TYPE OF SECTION VIEW AND DEFINE THE CUTTING PLANE. FOR STANDARD VIEWS, THIS OFTEN INVOLVES SELECTING ONE OF THE PRINCIPAL PLANES (FRONT, TOP, RIGHT) OR AN EDGE OF THE MODEL TO ALIGN THE CUTTING PLANE.

ONCE THE CUTTING PLANE IS DEFINED, SOLIDWORKS PROMPTS YOU TO PLACE THE SECTION VIEW ON THE DRAWING SHEET. THE SOFTWARE AUTOMATICALLY GENERATES THE VIEW, SHOWING THE CUT SURFACE AS SOLID, TYPICALLY FILLED WITH A SECTIONING PATTERN, AND THE REMAINING VISIBLE GEOMETRY AS LINES. THE DIRECTION OF THE SECTION IS INDICATED BY AN ARROW ON THE PARENT VIEW.

TYPES OF STANDARD SECTION VIEWS

SOLIDWORKS PROVIDES SEVERAL TYPES OF STANDARD SECTION VIEWS TO SUIT DIFFERENT MODELING SCENARIOS:

- **B-B (OR OTHER LETTER DESIGNATIONS):** THIS IS THE MOST BASIC TYPE, CREATED BY A SINGLE CUTTING PLANE. THE CUTTING LINE IS SHOWN ON THE PARENT VIEW, AND THE SECTION VIEW IS GENERATED PERPENDICULAR TO IT.
- **ALIGNED SECTION VIEW:** THIS VIEW IS USED WHEN YOU NEED TO SHOW FEATURES THAT ARE NOT COLLINEAR. THE CUTTING PLANE CAN BE BENT TO INCLUDE THESE FEATURES IN A SINGLE VIEW, MAINTAINING A STRAIGHT REPRESENTATION FOR CLARITY.
- **HALF SECTION VIEW:** USEFUL FOR SYMMETRICAL PARTS, A HALF SECTION VIEW SHOWS ONE-HALF OF THE OBJECT IN SECTION AND THE OTHER HALF IN EXTERNAL VIEW. THIS CONSERVES DRAWING SPACE WHILE REVEALING INTERNAL DETAILS.

ADVANCED SECTION VIEW TECHNIQUES FOR COMPLEX GEOMETRIES

FOR MORE INTRICATE DESIGNS COMMONLY ENCOUNTERED IN SAVOI'S ADVANCED ENGINEERING PROJECTS, STANDARD SECTION VIEWS MAY NOT BE SUFFICIENT. SOLIDWORKS OFFERS ADVANCED TECHNIQUES TO HANDLE THESE COMPLEXITIES EFFECTIVELY.

OFFSET SECTION VIEWS

AN OFFSET SECTION VIEW IS CREATED WHEN THE CUTTING PLANE NEEDS TO STEP OR OFFSET TO PASS THROUGH MULTIPLE FEATURES THAT ARE NOT ALIGNED ALONG A SINGLE STRAIGHT LINE. THIS IS ACHIEVED BY DEFINING A SERIES OF CONNECTED STRAIGHT LINE SEGMENTS THAT FORM THE CUTTING PLANE. THE VIEW THEN REVEALS ALL FEATURES INTERSECTED BY THIS STEPPED PLANE.

THIS TECHNIQUE IS INCREDIBLY USEFUL FOR SHOWING FEATURES LOCATED IN DIFFERENT PLANES OR AT VARIOUS DEPTHS WITHIN A COMPLEX PART. FOR EXAMPLE, AN OFFSET SECTION COULD REVEAL A PORT ON ONE SIDE OF A HOUSING AND A MOUNTING HOLE ON ANOTHER, ALL WITHIN A SINGLE, EASILY INTERPRETABLE VIEW.

BROKEN-OUT SECTION VIEWS

A BROKEN-OUT SECTION VIEW ALLOWS YOU TO REMOVE A PORTION OF A PART TO REVEAL INTERNAL FEATURES WITHOUT CUTTING THROUGH THE ENTIRE COMPONENT. THIS IS PARTICULARLY EFFECTIVE FOR LARGE ASSEMBLIES WHERE A FULL SECTION VIEW MIGHT OBSCURE TOO MUCH OF THE OVERALL ASSEMBLY. THE BOUNDARY OF THE BROKEN-OUT SECTION IS DEFINED BY A SKETCH, AND THE DEPTH OF THE CUT IS SPECIFIED.

THIS METHOD IS EXCELLENT FOR HIGHLIGHTING A SPECIFIC INTERNAL FEATURE WITHOUT SACRIFICING THE CONTEXT OF THE SURROUNDING GEOMETRY, MAKING IT IDEAL FOR DETAILED VIEWS OF SPECIFIC FUNCTIONAL AREAS WITHIN A SAVOI SUB-ASSEMBLY.

ROTATED SECTION VIEWS

WHEN DEALING WITH ANGLED FEATURES, A ROTATED SECTION VIEW CAN BE INVALUABLE. THE CUTTING PLANE IS ORIENTED AT AN ANGLE, AND THEN THE RESULTING SECTION IS ROTATED TO APPEAR AS IF IT WERE PERPENDICULAR TO THE VIEWING DIRECTION. THIS MAINTAINS THE VISUAL INTEGRITY OF THE ANGLE WHILE STILL PRESENTING A CLEAR SECTIONAL REPRESENTATION.

THIS IS PARTICULARLY USEFUL FOR COMPONENTS WITH ANGLED MOUNTING POINTS OR INTERNAL FEATURES THAT ARE NOT PARALLEL TO THE PRIMARY AXES, ENSURING ACCURATE REPRESENTATION OF THEIR TRUE FORM.

CUSTOMIZING SECTION VIEW PROPERTIES FOR CLARITY

ONCE A SECTION VIEW IS CREATED, ITS APPEARANCE CAN BE CUSTOMIZED TO ENHANCE CLARITY AND ADHERE TO SPECIFIC DRAWING STANDARDS, SUCH AS THOSE POTENTIALLY USED BY SAVOI.

CONTROLLING SECTION LINE APPEARANCE

THE APPEARANCE OF THE SECTION LINE ITSELF, INCLUDING ITS WEIGHT, STYLE, AND COLOR, CAN BE MODIFIED. THIS IS IMPORTANT FOR DISTINGUISHING DIFFERENT TYPES OF SECTION VIEWS WITHIN A SINGLE DRAWING OR FOR ADHERING TO COMPANY-SPECIFIC LINE STYLE CONVENTIONS.

SIMILARLY, THE LETTER DESIGNATION (E.G., A-A, B-B) ASSOCIATED WITH THE SECTION VIEW CAN BE CHANGED TO FOLLOW A SPECIFIC PROJECT OR COMPANY NUMBERING SCHEME. THIS ENSURES CONSISTENCY AND AIDS IN NAVIGATING COMPLEX DRAWINGS.

MANAGING SECTION FILL PATTERNS

SOLIDWORKS ALLOWS EXTENSIVE CUSTOMIZATION OF THE SECTION FILL PATTERN, WHICH VISUALLY REPRESENTS THE MATERIAL THAT HAS BEEN CUT. YOU CAN CHOOSE FROM A WIDE ARRAY OF STANDARD PATTERNS OR CREATE CUSTOM PATTERNS. THE SCALE AND ANGLE OF THE PATTERN CAN ALSO BE ADJUSTED.

FOR SAVOI PROJECTS, IT'S OFTEN BENEFICIAL TO USE SPECIFIC FILL PATTERNS OR ADJUST THEIR DENSITY TO CLEARLY DIFFERENTIATE BETWEEN DIFFERENT MATERIALS WITHIN AN ASSEMBLY OR TO MEET INDUSTRY-SPECIFIC DRAFTING STANDARDS. A WELL-CHOSEN FILL PATTERN CAN SIGNIFICANTLY IMPROVE THE READABILITY OF THE SECTIONED AREA.

VISIBILITY OF HIDDEN LINES AND EDGES

WITHIN A SECTION VIEW, YOU HAVE CONTROL OVER THE VISIBILITY OF HIDDEN LINES AND EDGES. BY DEFAULT, HIDDEN LINES ARE OFTEN SUPPRESSED TO AVOID CLUTTER. HOWEVER, IN CERTAIN SITUATIONS, DISPLAYING HIDDEN LINES CAN PROVIDE CRUCIAL CONTEXT ABOUT THE GEOMETRY BEYOND THE CUT PLANE.

THIS GRANULAR CONTROL ALLOWS YOU TO FINE-TUNE THE INFORMATION PRESENTED, ENSURING THAT ONLY THE MOST RELEVANT DETAILS ARE VISIBLE, THEREBY REDUCING VISUAL NOISE AND IMPROVING THE OVERALL CLARITY OF THE TECHNICAL DRAWING FOR SAVOI'S ENGINEERING TEAMS.

BEST PRACTICES FOR SECTION VIEWS IN SAVOI-SPECIFIC DRAWINGS

APPLYING BEST PRACTICES ENSURES THAT SECTION VIEWS ARE NOT ONLY INFORMATIVE BUT ALSO ALIGN WITH THE RIGOROUS STANDARDS EXPECTED IN ENGINEERING DRAWINGS FOR COMPANIES LIKE SAVOI.

CHOOSING THE RIGHT SECTION TYPE

THE FIRST AND MOST CRITICAL STEP IS SELECTING THE MOST APPROPRIATE SECTION VIEW TYPE FOR THE FEATURE YOU INTEND TO ILLUSTRATE. FOR SYMMETRICAL PARTS, A HALF SECTION CONSERVES SPACE. FOR COMPLEX INTERNAL PATHWAYS, AN OFFSET OR BROKEN-OUT SECTION MIGHT BE NECESSARY. ALWAYS CONSIDER WHAT INFORMATION IS MOST CRITICAL FOR THE INTENDED AUDIENCE – BE IT MANUFACTURING, ASSEMBLY, OR INSPECTION.

STRATEGIC PLACEMENT OF SECTION VIEWS

PLACE SECTION VIEWS STRATEGICALLY ON THE DRAWING SHEET TO MINIMIZE DRAWING CLUTTER AND ENSURE LOGICAL FLOW. OFTEN, A SECTION VIEW IS PLACED DIRECTLY IN LINE WITH ITS CORRESPONDING CUTTING LINE ON A PARENT VIEW. HOWEVER, FOR CLARITY, IT CAN ALSO BE PROJECTED TO OTHER AREAS OF THE SHEET.

ACCURATE DIMENSIONING OF SECTIONED FEATURES

ALL CRITICAL DIMENSIONS RELATED TO THE SECTIONED FEATURES MUST BE CLEARLY AND ACCURATELY DIMENSIONED. THIS INCLUDES DEPTHS, DIAMETERS, WALL THICKNESSES, AND ANY OTHER RELEVANT MEASUREMENTS. ENSURE THAT DIMENSIONS DO NOT OVERLAP WITH SECTION LINES OR FILL PATTERNS UNNECESSARILY.

CONSISTENCY WITH SAVOI'S DRAFTING STANDARDS

IF SAVOI HAS ESTABLISHED SPECIFIC DRAFTING STANDARDS FOR LINE WEIGHTS, TEXT SIZES, FILL PATTERNS, AND VIEW PLACEMENT, IT IS IMPERATIVE TO ADHERE TO THEM. THIS ENSURES CONSISTENCY ACROSS ALL DRAWINGS AND FACILITATES SEAMLESS INTEGRATION INTO THEIR EXISTING DOCUMENTATION PROCESSES.

TROUBLESHOOTING COMMON SECTION VIEW ISSUES

EVEN WITH EXTENSIVE EXPERIENCE, USERS MAY ENCOUNTER CHALLENGES WHEN CREATING OR MANIPULATING SECTION VIEWS IN SOLIDWORKS. ADDRESSING THESE COMMON ISSUES CAN SAVE SIGNIFICANT TIME AND PREVENT DRAWING ERRORS.

SECTION VIEW NOT CUTTING THROUGH THE DESIRED FEATURES

THIS OFTEN OCCURS WHEN THE CUTTING PLANE IS NOT CORRECTLY POSITIONED OR ORIENTED. CAREFULLY REVIEW THE CUTTING PLANE DEFINITION AND ENSURE IT INTERSECTS THE INTENDED GEOMETRY. SOMETIMES, SELECTING A DIFFERENT REFERENCE EDGE OR PLANE CAN RESOLVE THIS.

INCORRECT SECTION FILL PATTERN OR APPEARANCE

IF THE SECTION FILL PATTERN APPEARS INCORRECT, IS TOO DENSE, OR IS MISSING ENTIRELY, CHECK THE "DOCUMENT PROPERTIES" FOR SECTION LINE SETTINGS AND THE "SECTION VIEW" PROPERTY MANAGER. ENSURE THE CORRECT PATTERN IS SELECTED AND THAT IT IS ENABLED FOR THE VIEW.

HIDDEN LINES OBSCURING CRITICAL INFORMATION

IF HIDDEN LINES ARE CAUSING CONFUSION, YOU CAN SELECTIVELY SUPPRESS OR HIDE THEM FOR THE SECTION VIEW OR SPECIFIC COMPONENTS WITHIN IT. CONVERSELY, IF IMPORTANT INFORMATION IS BEING MISSED, CONSIDER ENABLING HIDDEN LINE VISIBILITY FOR THAT PARTICULAR VIEW.

MASTERING THE DRAWING SECTION VIEW IN SOLIDWORKS IS A CONTINUOUS LEARNING PROCESS. BY UNDERSTANDING THE FUNDAMENTAL PRINCIPLES, UTILIZING ADVANCED FEATURES, AND ADHERING TO BEST PRACTICES, YOU CAN CREATE HIGHLY EFFECTIVE AND ACCURATE TECHNICAL DRAWINGS THAT ARE ESSENTIAL FOR ANY ENGINEERING PROJECT, ESPECIALLY THOSE WITHIN THE EXACTING STANDARDS OF SAVOI. THE ABILITY TO CLEARLY COMMUNICATE COMPLEX INTERNAL GEOMETRY IS A HALLMARK OF PROFESSIONAL ENGINEERING DOCUMENTATION.

Q: HOW DO I CREATE A SECTION VIEW THAT CUTS THROUGH MULTIPLE ANGLED FEATURES IN SOLIDWORKS?

A: TO CUT THROUGH MULTIPLE ANGLED FEATURES, YOU WILL LIKELY NEED TO USE AN OFFSET SECTION VIEW. THIS INVOLVES CREATING A SKETCH FOR YOUR CUTTING PLANE THAT CONSISTS OF CONNECTED LINE SEGMENTS, ALLOWING YOU TO "BEND" THE PLANE TO FOLLOW THE DESIRED PATH THROUGH THE ANGLED GEOMETRY. ONCE THE SKETCH IS COMPLETE, USE IT TO DEFINE THE CUTTING PLANE FOR YOUR SECTION VIEW.

Q: WHAT IS THE DIFFERENCE BETWEEN A SECTION VIEW AND A DETAIL VIEW IN SOLIDWORKS?

A: A SECTION VIEW CUTS THROUGH A MODEL TO REVEAL INTERNAL FEATURES. A DETAIL VIEW, ON THE OTHER HAND, IS A MAGNIFIED VIEW OF A SPECIFIC SMALL AREA OF A LARGER PART OR ASSEMBLY, USED TO SHOW INTRICATE DETAILS THAT MIGHT BE TOO SMALL TO SEE CLEARLY IN THE MAIN VIEWS.

Q: CAN I APPLY DIFFERENT SECTION FILL PATTERNS TO DIFFERENT MATERIALS WITHIN THE SAME SECTION VIEW IN SOLIDWORKS?

A: YES, YOU CAN. BY USING THE "PROPERTIES" OF INDIVIDUAL BODIES OR COMPONENTS WITHIN AN ASSEMBLY, YOU CAN OVERRIDE THE DEFAULT SECTION FILL PATTERN AND ASSIGN A DIFFERENT ONE. THIS IS CRUCIAL FOR CLEARLY DIFFERENTIATING

BETWEEN VARIOUS MATERIALS IN COMPLEX ASSEMBLIES.

Q: HOW CAN I ENSURE MY SECTION VIEWS ARE COMPLIANT WITH ISO OR ASME DRAFTING STANDARDS?

A: SOLIDWORKS PROVIDES PRE-DEFINED TEMPLATES AND DOCUMENT PROPERTIES THAT CAN BE CONFIGURED TO MEET VARIOUS INTERNATIONAL DRAFTING STANDARDS LIKE ISO AND ASME. ENSURE YOU SELECT THE APPROPRIATE STANDARD IN YOUR DRAWING'S DOCUMENT PROPERTIES AND CUSTOMIZE LINE WEIGHTS, TEXT STYLES, AND FILL PATTERNS ACCORDINGLY.

Q: MY SECTION VIEW'S CUTTING LINE IS NOT APPEARING ON THE PARENT VIEW. WHAT SHOULD I DO?

A: THIS USUALLY MEANS THE CUTTING LINE IS EITHER COINCIDENT WITH AN EDGE OR IS TOO CLOSE TO THE EDGE OF THE PARENT VIEW TO BE DISPLAYED CLEARLY. TRY REPOSITIONING THE SECTION VIEW OR ADJUSTING ITS CUTTING PLANE SLIGHTLY. YOU CAN ALSO CHECK THE "DISPLAY OPTIONS" FOR THE SECTION VIEW TO ENSURE THE CUTTING LINE IS SET TO BE VISIBLE.

Q: HOW DO I DIMENSION FEATURES THAT ARE REVEALED BY A BROKEN-OUT SECTION VIEW IN SOLIDWORKS?

A: YOU CAN DIMENSION THESE FEATURES JUST AS YOU WOULD IN ANY OTHER VIEW. SELECT THE DIMENSIONING TOOL AND THEN CLICK ON THE RELEVANT EDGES OR POINTS OF THE INTERNAL GEOMETRY. THE DIMENSIONS WILL ACCURATELY REFLECT THE SIZE OF THE FEATURES REVEALED BY THE BROKEN-OUT SECTION.

Q: IS IT POSSIBLE TO HAVE MULTIPLE SECTION VIEWS FROM THE SAME CUTTING PLANE IN SOLIDWORKS?

A: YES, YOU CAN CREATE MULTIPLE SECTION VIEWS THAT ORIGINATE FROM THE SAME CUTTING PLANE. YOU WOULD CREATE THE INITIAL SECTION VIEW, AND THEN YOU CAN USE THE "SECTION VIEW" COMMAND AGAIN, SELECTING THE SAME CUTTING PLANE DEFINITION, TO PLACE ADDITIONAL SECTION VIEWS ELSEWHERE ON THE DRAWING SHEET.

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