ANALİTİK ASİSTAN — User Guide

Part coding, drawings, revisions, library and cutting-plan assistant for SolidWorks

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> Module names (YAVER, MUAVİN, KATİP, TERZİ, KILAVUZ…) are product names and stay the same in the English interface. What the program writes into your files — custom property names, the "Not required" mark and the column headings of the purchase and planning lists — follows the company language chosen at setup. This guide quotes them as written in a company set up in English (e.g. "Part Description", "Raw Material", "Surface and Heat Treatment"); in a company set up in Turkish they are the Turkish names ("Parça Tanımı", "Hammadde Malzemesi", "Yüzey ve Isıl İşlem", "Gerekmiyor"). The program reads both, so files coming from either kind of company keep working.

1. What ANALİTİK ASİSTAN is

ANALİTİK ASİSTAN does the work that a SolidWorks designer repeats on every project, following your company's own rules: it codes parts and assemblies with your code structure and saves them in the right project folder; it produces drawings, PDF, DXF, STEP and bills of materials; it corrects recorded data in bulk; and it tracks revisions and whether the drawings in the workshop are up to date.

The program runs on Windows with SolidWorks 2020 or later and finds the installed version by itself. Company settings (the company profile) are set up once and shared by all users. The program does not send your design data (files, codes, reports) to us or anywhere else. Only for license activation and trial registration does it send the license text, an irreversible identifier of the computer and the program version to analitikasistan.com (section 15). The Odoo connection goes only to your company's own Odoo server; license renewal and extension forms open in the browser.

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2. Installation and first start

Installer

Run the installer (AnalitikAsistan_Kurulum_<version>.exe, or the same file inside the ZIP). If Windows says "unknown publisher", choose "More info" and then "Run anyway". Setup adds ANALİTİK ASİSTAN, ATÖLYE KONTROL and this guide to the Start menu.

Welcome screen

On first start the program shows the Welcome screen, which explains in three steps what the program does and how to begin. If there are no company settings yet, the main button is START SETUP. You can reopen the screen later with the "First steps" link at the bottom of the panel.

Release notes

What changed in each version is inside the program: the "Release notes" link at the bottom of the panel (or the button in the About window) shows the notes of all versions, newest first, in the screen language; no internet is needed. On the first start after the program is updated, "What's new in this version" opens once by itself.

Setup wizard: quick setup

In the first step enter the usage type, the company name and the main folder of your files, then press "Quick setup ›". The other steps are filled with recommended settings and the Summary opens; "Save profile" completes setup. You can change the code structure, lists and other settings later by reopening the wizard.

Setup wizard: detailed setup

"Detailed setup ›" shows the nine steps one by one: Company, Code structure, Protected codes, Revision, Folders, Lists, Approval, ERP, Summary. In every step "Save" stores what has been entered so far.

Usage type

Type	When
1	One person, one computer.
2	One person, files on a server.
3	Several people, files on a server (manager approval can be switched on).
4	Several people, SOLIDWORKS PDM.

Other computers: linking to the shared profile

If the program has already been set up on another computer in your company, choose "Link to the SHARED profile already set up in our company" in the first step of the wizard and point to the shared profile. All computers then work with the same settings.

First setup and SETUP CHECK

When the profile is saved, the program prepares what is missing: folders, drop-down lists, copies of SolidWorks' own templates and, if SolidWorks is open, drawing templates with a title block. SETUP CHECK checks the folders, lists and templates later on as well and fixes what is missing with one click. While SolidWorks is closed, items that depend on SolidWorks settings show "could not be checked" rather than a problem; open SolidWorks and run the check again.

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3. First try: the sample assembly

To try the modules without touching your own files, press "Create a sample assembly" on the Welcome screen (SolidWorks must be open). A small uncoded assembly is built in SolidWorks: a bearing group made of a shaft, a flange, a cover, a bearing housing and a pin. The assembly is placed in your company's project layout, in a sample customer and project folder numbered with 9s.

Suggested order: first code the assembly with HIZLI KODLAMA, then produce the drawings and lists with MUAVİN, and correct a property with KATİP. If the sample assembly already exists, it is opened, not rebuilt.

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4. Main panel and SolidWorks task pane

The main panel arranges the modules in four groups: DESIGN AND CODING, OUTPUT AND CHECKS, LIBRARY AND WORKSHOP, HELP. Below each button is a short description of what the module does. The status line at the bottom of the panel shows whether SolidWorks is open, the PDM state and the company profile; clicking the "License: ..." line opens the LICENSE window.

SolidWorks task pane

When the program connects to SolidWorks, an ANALİTİK ASİSTAN tab opens in the task pane on the right of SolidWorks: module buttons, the status line and "Show the main panel". If you do not want it, switch it off in SETTINGS ▸ Appearance.

Working indicator

When the program cannot respond during a long job (SolidWorks saving, producing PDF or STEP and so on), a small "working" box with the animated logo and the elapsed time appears next to the open window. If the job takes more than 2 minutes it says it is taking long: SolidWorks may be waiting for an answer in a dialog box.

SETTINGS

Here you set the program's width on screen, its placement next to SolidWorks, which modules open in the strip on the left, the coloured title band of module windows, the task pane, the screen language and the QR guard when saving drawings.

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5. Code structure and coding

Code structure

The setup wizard offers a ready template with five parts: company/customer, project/machine, group, assembly, part. For each part you choose the type (letters, digits), the number of characters, a fixed prefix and the empty value, as well as the separator. Machines whose code must never change are marked as "protected".

YAVER: coding the tree step by step

Starts from the top assembly and walks the tree; it brings every part, sub-assembly and group whose code or data is missing in front of you in turn. The customer and project numbers are read from the top assembly; the part number is the next free number. APPLY saves the file under its code name in the right folder, writes the custom properties and redirects the link in the assembly to the new file; it also moves the drawing and produces the QR code. Standard and purchased parts are replaced by their library counterparts. There are PASS, SKIP and UNDO for the last coding; if the work stops halfway, it continues where it left off. If the customer folder carries its name next to its number (e.g. "<number>-<name>") and there is no folder with the bare number, the new project is set up in that folder; no second customer folder is opened.

HIZLI KODLAMA: a ready tree in one go

Codes the whole tree of a project whose data is already in the model, in a single operation. It first makes a plan: which file gets which code (To be coded tab). If there is a blocker, it touches no file and lists the blocker on the "Blockers and warnings" tab. START CODING asks for confirmation; the operation cannot be undone. Files left behind with their old names after coding are moved to the Recycle Bin with your approval.

MEKANİKÇİ: new parts and assemblies

Opens a new part or assembly already coded and with its properties filled in.

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6. Checking and correcting

ANALİZ

Checks the whole tree of the top assembly and produces a colour-coded Excel report; it writes to no file.

Finish and heat treatment: the "Not required" mark

A part whose coating, paint and heat-treatment fields are all empty is a ⚠ warning in ANALİZ; it is not counted as an error and does not stop sending for approval. Mark a part that needs no treatment as "Not required" in the "Surface and heat treatment" box at the top of the finish block in YAVER or MEKANİKÇİ, or in the column of the same name in KATİP; ANALİZ then shows a green "not needed". The mark and the treatment fields do not go together: choosing one empties the other; if both are filled, ANALİZ warns that they are contradictory. The same box exists for the post-weld treatment of welded or soldered assemblies. In KATİP, "Select those

awaiting a treatment decision" selects parts whose fields are empty and that are not marked; you are asked before "Not required" is applied in bulk.

Surface finishing: grinding, honing, lapping…

Apart from coating, paint and heat treatment, the part's finishing process is entered too: in YAVER and MEKANİKÇİ, click the "Finishing process" box under the finish block and tick one or more processes in the window that opens (grinding, honing, lapping, superfinishing, polishing, brushing, burnishing); next to it are roughness Ra (µm), finishing allowance (mm) and a note saying which surface. If no finishing is needed, tick the "Not required" box beside it. The mark and the finishing fields do not go together: choosing one empties the other. Ra and allowance are stored with a point whatever separator you type ("0,8" → "0.8"; company setting). The values are written to the file only when filled. A part without finishing is a ⚠ warning in the last "Finishing" column of ANALİZ and does not stop sending for approval; documents in the protected (standard machine) tree get no such warning. KATİP has five columns; with "Finishing Process" selected, "Multiple…" gives more than one process, "Select parts awaiting a finishing decision" selects parts whose fields are empty and not marked; a field you empty is removed from the file. The process and Ra lists are edited in the Lists step of the setup wizard. Important: an older version of the program removes the finishing data when it applies a part again; everyone should update before finishing is entered.

TAMİR

Completes the missing properties found by ANALİZ on a single screen.

KATİP: correcting recorded data

Corrects all custom properties of finished parts in bulk. Protected fields such as the code, weight, material property and revision are locked. Select rows in the table, enter the Field and the new value and press APPLY TO SELECTED; changes wait in the table first, and SAVE AND CHECK IN writes them to the files. There are also NEW PROPERTY…, DELETE PROPERTY and MOVE TO SCHEMA NAME….

KATİP and Excel

EXPORT TO EXCEL exports the whole table to Excel: locked cells are grey and protected, material and list fields are drop-down lists. Read back the file you corrected and saved in Excel with IMPORT FROM EXCEL…: only the changed cells appear in the preview as "current → in Excel"; APPLY TO TABLE turns them into pending changes and SAVE AND CHECK IN writes them. A cell emptied in Excel empties the value; the property stays.

ARAÇLAR

Show in folder, where a part is used (Where Used), file information, ASSEMBLY DIAGRAM, K-factor calculation and RAW MATERIAL SIZE: the bounding box of the part, or of all parts in the assembly, is measured and "Stock Size" (raw material size) is written in bulk (round parts as Ø×length, others as a box; machining allowance and rounding come from your company settings). The allowance can be set separately by the part's "Machining Type" (turning, milling, wire EDM) in the Lists step of the setup wizard; the machining type is chosen in YAVER and MEKANİKÇİ and written in bulk with KATİP; if none is chosen, the general allowance is used. ASSEMBLY MATES: all mates of the active assembly and its sub-assemblies are listed with type, the components they connect, value (distance / angle), alignment and the assembly they are defined in; the document is not changed. If a component is selected in SolidWorks, the list opens filtered to that component; another

component can be chosen in the "Component" box. If there are mates with errors, warnings or suppressed mates, a warning is shown after reading and their rows are coloured ("Problems only" shows just those). Double-click a row to select the mate in SolidWorks; EXPORT TO EXCEL writes the list into the project's XLSX DOSYALARI folder. ASSEMBLY DIAGRAM shows the structure of the assembly card by card with quantities; EXPORT gives HTML (for sharing), SVG, Excel (collapsible tree, machine total per document, summary) and PDF. The PDF is a printable tree list: with indent lines, the item number, code, description, material, configuration, class, code status / vault, quantity under its parent and total quantity; it follows the filters on screen (hidden classes are left out of the PDF and the filter is shown in the header).

HESAP: engineering calculations

Choose the calculation in the list on the left, enter the inputs and press CALCULATE; the result, warnings and the formula used appear on the same screen, and PDF REPORT gives a one-page calculation report. Calculations: bolt tightening torque and preload (simplified VDI 2230), pin shear and bearing; shaft diameter (bending + torsion), bearing life L10, parallel key length; motor power / torque / gearbox, belt and pulley, chain, spur gear (module, Lewis); beam deflection and stress, fillet weld, hydraulic / pneumatic cylinder force. Standard product dimensions such as profiles, circlips and keys come from the Analitik Parts Library. The calculations support design; they do not replace proof against standards / regulations or an expert review.

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7. Drawings and outputs (MUAVİN)

Drawings

The drawing of a single part or assembly, or all drawings of a project in bulk. The sheet size (A4 / A3) is chosen from the size of the part; the front view shows the part's longest dimension horizontally, and the largest standard scale that fits the sheet is used; sheet-metal parts get a flat-pattern view. The code, title, material and weight in the title block come from the part's properties; "drawn by" and "checked by" are asked once. The corner of the drawing carries the QR code that is scanned in the workshop.

Take dimensions from the model

Optional: on part drawings, dimensions marked "for drawing" in the model and hole callouts are added to the drawing. Parts drawn without dimensions or imported from STEP get no dimensions; review the drawing of a part with many dimensions.

Outputs

PDF, DXF, STEP, JPG and the part's QR code go into the project's own folders; QR codes are collected in the main assembly's folder. If QR codes were left in other folders, you are asked at the end of the job whether to collect them; the files are moved, not copied. JPGs are taken on a plain white background and your SolidWorks settings come back when the job is done. Revisioned file names: only the code at first release, then code_R01, code_R02…

QR on save (QR guard)

The QR code also comes automatically when you save a drawing directly in SolidWorks: while the program is open, if a drawing being saved has no QR, or its QR does not match the code or revision of the drawing's model, it is added to or renewed in the corner of the sheet. Drawings in library, template, database and old-revision folders are not touched. If SolidWorks freezes or is restarted, the guard finds the open SolidWorks again. If you do not want it, switch it off in SETTINGS ▸ Appearance; it can be switched off for the whole company in the drawing section of the company profile.

Export

The parts and sub-assemblies you select in the tree are exported into their own folders as STEP, IGES, Parasolid (X_T), STL, 3D PDF, eDrawings, JPG, drawing PDF, DXF or DWG — for example files going to a supplier. All sheets of a multi-sheet drawing are in a single DXF/DWG.

Renew title block

The sheet format (title block) of existing drawings is replaced in bulk by the company's current template. Views, scale and drawing data are kept, the QR is added again and the revision does not increase; the original of every drawing is copied into an "ESKİ ANTET" (old title block) folder next to it. A drawing whose sheet format is already up to date is not touched (not copied, not saved). In a drawing without views (e.g. the top assembly drawing) part or assembly is told from the model file with the same name.

Bill of materials (BOM)

The assembly's Purchasing and Planning lists are exported to Excel; there is a separate "Rev" column next to the "Kod" (code) column. The part's method (Üretim / Hazır / Norm / Montaj) is in the "Yöntem" column; the Planning list has a "Montaj Türü" (assembly type) column before the code. Both lists, and the end of the ERP CSV, have finishing process, Ra, finishing allowance and finishing note columns. If an ERP is selected in the profile, both lists (before the QR) and the very end of the ERP CSV also have the "Reference Code" column of standard and purchased parts. The column layout of the Odoo CSV is a company setting: in a new installation "Montaj Türü" comes right after "Yöntem" (column E); earlier installations keep the previous layout ("Assembly Type" in column X). A drawing is taken and saved once for a part in the list that has none; drawings and QR codes of standard and purchased library parts are also saved in their own folders, so these parts are not opened again for later lists. For ERP export see section 13.

Raw material order list

MUAVİN ▸ List ▸ "Raw material order list" totals the raw material of the project's manufactured parts by material and section: a part whose "Stock Size" (raw material size) is Ø×length becomes a round bar, a part with three dimensions a flat / square / block bar (section = the two smaller dimensions, length = the largest), a sheet-metal part (thickness and flat-pattern size written by TERZİ ▸ FLAT PATTERN) a sheet. Bars are combined into stock lengths with TERZİ's cutting engine (stock length and saw allowance from the TERZİ settings; if a row in the shared "Profile Stock Lengths" table matches the group — e.g. "Round bar" → 3000 — that length): the number of stock bars needed and the scrap are shown, and a part longer than the stock length is listed separately in the note; sheets are listed by thickness with count and m^2. If "Raw Material" is filled in on the part (e.g. cold drawn), that material is used. Parts without a raw material size are counted separately; measure them with ARAÇLAR ▸ RAW MATERIAL SIZE, and use TERZİ ▸ PROFILE for profile parts.

Cost report (estimate)

MUAVİN ▸ List ▸ "Cost report" calculates an estimated cost of the assembly from the company's price table: raw material (price per kg per material; if "Stock Size" is filled in, the weight of the blank or plate, otherwise the part weight), laser cutting (cut length of the flat pattern × price per m), bending (number of bends × price), machining (chips removed from the blank × price per kg), assembly labour for each assembly item, and the unit price of purchased/standard parts. The price table is "Cost Prices.xlsx" in the shared Tables folder ("Maliyet Fiyatları.xlsx" in a Turkish-language company); the first time it is created with your materials and the project's purchased/standard codes and empty prices, then opened — you enter the prices. The report is written to the XLSX DOSYALARI folder with Summary, Items and Assemblies sheets; an item with a missing price or weight is shown in the "Note" column and the total is short by that much. Because it contains prices, it is not checked into the vault. The result is an estimate; it does not replace a quotation or the accounting cost.

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8. Revisions, workshop and distribution

REVİZYON

The code and file name do not change; the revision is kept in the "Revision" property (empty means R00, then R01, R02…). A description of the change is mandatory and is written to the shared revision log. If the part loses interchangeability (form, fit, function), a new code is suggested instead of a revision. An assembly's revision increases only when its own content changes.

Archive

When the output of a new revision is produced, the old PDFs and DXFs are moved to the "OLD REVISIONS" folder ("ESKİ REVİZYONLAR" in a Turkish-language company) and their pages are stamped "VOID".

ATÖLYE KONTROL

On the workshop computer the drawing's QR code is scanned with a USB QR reader; the screen shows CURRENT or OLD DRAWING in large letters and, if it is old, the current revision and the description of the change. No SolidWorks and no server are needed. It opens directly from the ATÖLYE KONTROL shortcut in the Start menu.

DAĞITIM

Records which drawing went to whom and with which revision; when the revision increases, it lists from whom it must be taken back. If a drawing you want to send is at an old revision, you are warned first.

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9. Library

KÜTÜPHANEDE ARA

Searches the standard part, purchased part and profile section library; next to the result it suggests "similar ones are in this folder". If the word you search for appears nowhere, you are told so and the closest results are

shown — if a similar part exists, do not draw and save it again. No SolidWorks needed. A part found can be dragged into the assembly in SolidWorks.

ERP catalogue tab

Search for a product in the ERP by code or name, select it and press ADD TO ASSEMBLY. If a model with the same code exists in the library, that file is added; otherwise a "virtual component" (no file; its name and Parça Kodu are the ERP code, its Parça Tanımı the name in the ERP) is added to the assembly open in SolidWorks. The library file's properties are not changed.

KÜTÜPHANEYE KAYIT

Saves a part into the company library in the right folder with the right name. In multi-user setups, saving to the library can be limited to authorised people. If an ERP is selected in the profile, the ERP's internal reference ("Reference Code") is also written to standard and purchased parts (in YAVER and MEKANİKÇİ too); it is optional, and if it is left empty ANALİZ and KONTROLÖRCÜ warn.

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10. Cutting plans, sheet metal and press brake (TERZİ, ABKANT)

Profile cutting plan

READ FROM ASSEMBLY groups the profile parts by section and material; quantities are the project totals. Stock lengths and the saw allowance are yours (the lengths the company buys can be kept in one place, in "Profile Stock Lengths.xlsx" in the shared Tables folder: the "Stock table" button opens it, or creates it from the profiles in the list; a row applies when every word of its profile text appears in the group's name — "box profile" for all box profiles, "Box Profile 40x40x2 · S235" only for that one; a profile with a manually entered length keeps it, and a length from the table is marked "table" in the list); the plan chooses the most efficient length for each bar and shows the scrap and the offcuts going back to stock. It gives an Excel list for the workshop; the new bars in the plan also come out as a combined purchasing list by profile and stock length (a PURCHASING sheet in the workbook and a separate "Profil Satın Alma" file; parts cut from offcuts in stock are not included). CHART shows the plan bar by bar as coloured strips (parts with length and code, saw allowances, the offcut going to stock in green, scrap in red); new bars cut to the same pattern are shown once as "× n". PDF writes the same chart for the workshop into the project's PDF DOSYALARI folder.

Sheet nesting

Parts are added from the assembly, from DXF or by hand; cutting gaps per machine, plate stock; PDF and DXF per plate. The nesting can be corrected by dragging and dropping on the canvas. The utilisation of the plate is shown in large figures in the top right corner of the canvas (75% and above green, 50–75% orange, below 50% red); at low utilisation a smaller sheet or planning together with other parts is suggested.

Flat patterns and K-factor

Flat-pattern DXF or DWG from SolidWorks' sheet-metal data; flat patterns are separated by material and thickness. The shared sheet bending table (material, thickness, inner radius, K-factor) writes the table's K into

the bends. Cut length, number of bends, sheet thickness and flat-pattern size are written into the part's properties.

ABKANT: bending plan and 3D simulation

Plans in which order, and with which punch and die, a sheet-metal part is bent on the press brake, and shows it step by step in 3D. Open the sheet-metal part in SolidWorks (or select it in an assembly) and press GET FROM SOLIDWORKS; a part in the TERZİ ▸ FLAT PATTERN list also comes with "OPEN IN ABKANT". The program reads the angle, direction, inner radius and flat-pattern line of each bend; while reading it briefly unfolds and refolds the part, does not save it, and compares its own bending calculation with SolidWorks' bent part. For each step it states the die (V opening by thickness), the punch, the back-gauge dimension (X), the required force and whether the part has to be flipped; it warns if a flange is too short for the die or the force exceeds the machine or the tooling. Change the order with ▲ ▼ and the die and punch from the list. ▶ Play animates the bends on the machine (top face of the part blue, bottom face sand-coloured). PDF REPORT gives the sheet for the workshop: flat pattern, step table and a picture of each step; SAVE PROJECT writes the plan to a .abkant file. Every step is checked at several intermediate angles for collisions with the punch, die, upper beam, lower table and columns: a colliding step shows a red ✕ in the list, and the colliding surfaces are red in the 3D view and in the report. If you did not choose the punch yourself, other suitable punches (long straight, gooseneck) are tried automatically to avoid the collision. SUGGEST ORDER looks for an order without collisions, with few flips and few tool changes; the suggested order is applied with your approval. This is a geometric simulation: it does not replace a workshop trial or the machine program.

Press-brake tools and machine

TOOL TABLE opens the "Press Brake Tools.xlsx" file in the company's Tables folder ("Abkant Takımları.xlsx" in a company set up in Turkish), with Punches, Dies and Machines sheets. On first opening it comes with typical dimensions of common European-style tools; enter the dimensions from your own catalogue, the segment lengths of sectioned tools and the permitted load (kN/m), put an x against your favourite tool, then "Reload table". Table length, distance between columns, throat depth, upper beam and lower table thickness on the Machines sheet are used in the collision check. The tensile strength (Rm) is read from the part's SolidWorks material, otherwise estimated from the material name; it can be changed in the window.

YÜKLEME: container, trailer and truck loading plan

Plans how load items such as machines, sub-assemblies, crates or pallets are placed in a vehicle. Items come with FROM ASSEMBLY (top-level components of the open assembly), FROM SELECTION (components selected in SolidWorks), ADD ITEM or FROM EXCEL (in a company working with ANALİTİK KONSTRÜKTÖR the first button is GET FROM KONSTRÜKTÖR: every open document you choose becomes one item); the same part appears once with its quantity. Size is read from the component's box, weight from the model's mass; if your assembly's up axis is not Y, choose it in "Up axis". If you mark an item "Stackable on top", lighter items are stacked on it when the floor is full, never heavier than the item itself. Vehicles (20' / 40' / 40' HC container, trailer, truck) are in "Loading Vehicles.xlsx" in the company's Tables folder ("Yükleme Araçları.xlsx" in a Turkish-language company); "Vehicle table" opens it, add your own vehicles. PLAN places the items on the floor, rotated 90° where needed, moves what does not fit to the next vehicle and shifts the load so the centre of gravity is in the middle; it warns about overload, an off-centre centre of gravity and items that do not fit any vehicle. Each vehicle can be seen as top + side view or in 3D (rotate with the mouse). PDF gives a loading plan

with one page per vehicle, EXPORT TO EXCEL a list of items with vehicle and position for every unit; both are written to the assembly's PDF / XLSX folders. The layout is a suggestion: check lashing, securing and road rules at shipping.

PALET / SANDIK: pallet and skid base before loading

Sizes the timber skid base (pallet) placed under a machine or crate. The load comes with GET FROM SOLIDWORKS from the open assembly or part (size from the box, mass and centre of gravity from the model; if your assembly's up axis is not Y, choose it in "Up axis") or is typed in; if the centre of gravity is empty, the middle of the load is used. Coordinate systems in the model whose names begin with KALDIRMA are read as lifting points; otherwise the program suggests four sling points at the skid ends. CALCULATE chooses the base size (load + edge margin), the number and section of the skids and the deck board thickness (clear spacing between skids at most 80 cm; from 1000 kg gross the skids are at least 120 mm high so that forklift forks fit — TIS/GDV packaging handbook), and checks to Eurocode 5 (C24 timber) deck board bending, skid bending (crane lift, and forklift at the fork spacing), skid shear and deflection. The utilisation of each check must be at most 1; a failing row is red. For the chosen transport (road, rail, sea A / B / C) it checks tipping with the CTU Code accelerations and calculates the lashing forces needed. The base can be seen as top + side view or in 3D. PDF gives a one-page pallet report (views, checks, cutting list, lashing forces), EXPORT TO EXCEL the timber cutting list, summary and checks; both are written to the model's PDF / XLSX folders. SEND TO YÜKLEME passes the pallet (outer size and total mass: load + tare) to YÜKLEME as an item; if the YÜKLEME window opened by this button is still open, the item is added to it, so you can collect several pallets in one loading plan. SHIPMENT FILE writes the outer size, tare and contents of the pallet or crate to a file to be imported as a crate record into the shipment of the ANALİTİK BÜTÜNLEŞİK YÖNETİM SİSTEMİ ("... - Pallet.json" or "... - Crate.json", in the model's SEVKİYAT DOSYALARI folder). SOLIDWORKS MODEL builds the pallet or crate in SolidWorks as parts and an assembly (a "... - Crate" folder next to the model; parts with the same size share one file; the load and diagonals are not modelled); a crate with 40–50 components takes about a minute, and the assembly stays open. The design load is the load's weight multiplied by the Eurocode partial safety factor (γ_G 1.35, EN 1990) and the crane's dynamic factor (φ2, EN 1991-3; default 1.2 — HC2 class crane, about 0.3 m/s hoisting speed); change φ2 in Settings to suit your crane. As no separate source was found for forklifts, the same φ2 is used. This is a preliminary design: check nail, screw and bolt connections and lashing at shipping separately.

Closed crate

When Closed crate is chosen under "Type", the program also builds the walls and the lid on top of the pallet (rules of the TIS/GDV packaging handbook and Eurocode 5). The inner size is the load plus the inner clearance (at least 30 mm, 50 mm with VCI or desiccant); the base size follows from the inner size and the wall thickness. The lid beams are spaced at most 800 mm apart; the number and section of the beams and the lid sheathing thickness are chosen to use the least timber. If "Stackable" is ticked (the default), the lid is designed for a stacking load of 1.8 t/m^2 and the side wall posts with a section that carries it (at least 50 × 100); if it is cleared, the lid is designed for its own weight only and the PDF says DO NOT STACK. For crane lifting, the sideways compression of the slings on the lid (0.3 times the gross weight) is checked with buckling in the end lid beams. The wall / lid sheathing can be 24 mm boards or OSB/3; with boards, diagonals are added to the walls when the inner height exceeds 1500 mm. The cutting list, tare and outer size include all parts of the crate; in the 3D

view the sheathing is transparent. PDF and Excel are written as "... - Crate", and SEND TO YÜKLEME passes the crate as an item.

Overseas packing

When "Desiccant, ISPM 15, marks" is ticked under "Overseas", the type becomes Closed crate and the transport Sea C (you can lower it to A or B; a warning appears if road or rail stays selected), and the inner clearance is at least 50 mm. The inside of the crate is lined with a moisture barrier foil; the desiccant needed is calculated with the DIN 55474 method: the water in the air of the inner volume, the water given off by timber left inside the barrier and the water passing through the foil during the protection period are added up, and one desiccant unit absorbs 6 g of water. The default foil is aluminium composite (water vapour permeability 0.1 g/m^2·day); if PE is chosen, enter the manufacturer's value. The default protection period is 180 days. Keep timber chocks outside the barrier; if they stay inside, enter their mass (the desiccant need rises a lot). For ISPM 15 the solid wood parts that need heat treatment and the mark are listed (OSB is exempt). The marking plan is given with ISO 7000 numbers: this way up, keep dry, centre of gravity, sling here, stacking limit or do not stack, gross mass and size text, ISPM 15 mark; their positions are numbered on the side and end faces on the second page of the PDF. Excel has an OVERSEAS sheet. If VCI is used, take the dosage from the manufacturer.

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11. Material and finish suggestions (KILAVUZ)

Opens with the "KILAVUZ — Decision Support" button when you create a new part in MEKANİKÇİ and while coding in YAVER. It has four tabs: Material & Heat Treatment, Surface Treatment & Coating, Tolerances & Fits, Geometric Tolerances. You choose what the part does, the suggestion comes with its reasoning, and the assign button writes it into the form. The final decision is the designer's.

Current values of the part

KILAVUZ opens with the material, heat treatment, hardness and coating values in the form (in YAVER, of the open part) and points out conflicts between them: a heat treatment that does not suit the material (e.g. case hardening a structural steel), a hardness on a material that cannot be hardened or that cannot be reached, a hardness entered with an empty heat treatment, zinc on aluminium, anodising on steel, electrolytic plating on high-strength steel (hydrogen embrittlement). The material tab opens at the place of the part's material in the rules.

Material and alternatives

The "Suggestion and alternatives" table below the suggestion shows the other options for the same use and 2–3 alternatives to the material: density, an assessment of welding, machining and corrosion, and when to prefer it; selecting a row also shows the typical tensile strength and the hardness that can be reached. The row you double-click (or select with "Use the selected row") becomes the result. Values are typical; exact values depend on the section and the delivery condition.

Surface treatment and coating

You are asked for the material (steel, stainless, aluminium, copper alloys), the environment, the precision, any special requirement (food contact, electrical conductivity, continuous high temperature) and the heat-

treatment state; the result of the material tab fills in the material and the heat-treatment state. The suggestion comes with the pretreatment, the dimensional growth (for electrolytic plating the thickness on each surface, twice that on a diameter; for anodising half inwards, half outwards) and compatibility warnings: hydrogen embrittlement and the embrittlement relief bake, the galvanising bath softening a hardened part, the order of operations (heat treatment and grinding before coating), hard chrome lowering fatigue strength, anodising 7xxx and cast aluminium, galvanic pairs in wet conditions. The exact time and temperature are set by the coater's specification.

Matching your company's lists

The suggestion is written into the form with its counterpart in your company's list: for materials the equivalent name (e.g. "C45 (1045)" → "1.0503 (C45)"), for coatings, paints and heat treatments your company's name for the same process (e.g. "Hard Chrome Plating" → "Hard chrome"), for hardness the range that overlaps most. If there is no counterpart, the window says so and the suggestion is written as it is. In a company whose data language is English, the treatment names KILAVUZ suggests are in English.

Your company's own rules

"Edit the rules (Excel)..." on the material tab opens the "KILAVUZ Material Rules.xlsx" file in the shared Tables folder ("KILAVUZ Malzeme Kuralları.xlsx" in a Turkish-language company); if the file does not exist, it is created from the built-in rules. Once you change the rows and save, KILAVUZ uses your company's rules; if the table cannot be read, it falls back to the built-in rules.

KILAVUZ Gerekçesi (rationale)

Assigning also sends a short summary of the choice; when the part is saved it is written with the date into the "KILAVUZ Rationale" custom property (e.g. "KILAVUZ 31.10.2026 · General machined part / High load / fatigue → 1.7225 (4140 - 42CrMo4), Quench and Temper, 28-35 HRC"). If you later change a value that KILAVUZ assigned by hand, the rationale is not written; YAVER keeps the rationale already in the part when it codes it again.

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12. Manager approval

In multi-user setups (types 3 and 4) it can be switched on in the Approval step of the setup wizard. When it is on, the panel shows ÖN ONAY ANALİZ (pre-approval analysis), ONAY and SON ONAY ANALİZ (final-approval analysis) instead of ANALİZ. The designer sends finished work for manager approval from the ONAY window; the manager (or a deputy) approves it or asks for a fix. Decisions are written to the KARAR subfolder of the shared record folder: give write permission on this folder only to the manager and the deputies (IT sets it); SETUP CHECK warns if a user who is not an approver can write to it. On the Pending tab, "Approved by me" lists only the submissions the manager approved. MUAVİN's approval gate is off in a new company; choose "Warning" or "Block" in the Approval step.

Approval gate: Warning and Block

The gate comes into play when output is produced in MUAVİN (Save As, Export) for an unapproved part; approved revisions and "old work" saved before the start date are not questioned. With "Warning" a reason is

asked for, recorded, and the output is produced. With "Block" no output is produced for the unapproved part; three ways are offered: EMERGENCY EXCEPTION (all outputs with a written reason; the manager is e-mailed), DRAFT PDF (only the drawing PDF, every page stamped as a draft — "DRAFT — NOT APPROVED" in an English-language company — written to the draft folder in Documents without being checked in; no JPG, STEP, HTML or QR) or CANCEL. The company can switch off the emergency exception or the DRAFT PDF in the Approval step. Every choice is recorded.

Staged approval: drawing stage and final approval

Optional; switched on in the Approval step (off in a new company). When it is on, approval has three stages: PRE-APPROVAL (model), DRAWING STAGE, FINAL APPROVAL. For a part that has pre-approval and a ready QR code, the designer states in ONAY ▸ Drawing Stage whether they will produce the drawing themselves or assign it to a drafter. The drafter finishes the drawings in My Drawing Tasks and asks the designer for approval; in Drawing Approvals the designer marks each drawing OK or to be fixed. Once the drawings are complete and JPG, HTML and PDF have been produced from MUAVİN, the designer chooses FINAL APPROVAL on the Send tab; SON ONAY ANALİZ must be clean. If the revision goes up, the stages start again. The list of drafters comes from "Drafters" in the Approval step.

Finish and heat treatment in approval

The item line also shows the part's finish and heat-treatment state: "no treatment needed", "surface/heat empty" or contradictory; the manager sees it too. The finishing process shows with its own label ("finishing empty", "finishing not required"); if both finish/heat treatment and finishing are empty, a single label "surface/heat/finishing empty" is shown. An empty field does not stop the submission. If the company wishes, it selects "Empty finish/heat treatment: MANDATORY" with a start date in the Approval step of the wizard: for parts saved after that date, if the fields are empty and "Not required" is not ticked, ÖN ONAY ANALİZ counts it as missing and the submission stops; parts saved earlier stay at warning.

KONTROLÖRCÜ pre-review

In the manager's ONAY screen, with a submission awaiting a decision selected, there is a "KONTROLÖRCÜ ▶ pre-review" button. The program opens the submission's parts, assemblies and drawings read-only in a separate, hidden SolidWorks; it does not touch the open SolidWorks or the files. It checks material and finish/heat-treatment compatibility, mass, interference, floating parts, bolt hole diameters, tolerances on fitted pairs, overall dimensions on the drawing, undimensioned diameters and the title-block scale. Findings are written as drafts into the item notes and the general note; nothing written by hand is overwritten, and the decision and the sending remain the manager's. Interference findings are marked "unstable measurement" and must be checked by hand. For coaxial fits between two parts of different materials an early warning about the difference in thermal expansion is given; this is not a calculation, the decision is yours. At pre-approval no drawing is required; at final approval a part without a drawing is marked to be fixed. An empty material is not an error for a part made from a purchased or standard part. If a part cannot be read, the review ends with a red "MISSING" warning; an unreadable part does not count as clean and you are asked again when sending the decision. Movement clearances, strength and the visual readability of drawings are outside this check.

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13. ERP export

Export by file

Odoo, Logo, Netsis or Mikro is selected at setup. MUAVİN ▸ List ▸ ERP export writes the stock cards and the product tree into that ERP's import file (CSV for Odoo, a two-sheet Excel for the others; column names in the ERP's language).

Sending directly to Odoo

With a live Odoo connection the preview shows how many cards are new and how many will be updated; on confirmation the stock cards and the product tree go to Odoo, and existing records are not duplicated. The password or API key is kept in the Windows credential store and is not written to the shared profile; if the address starts with http://, the key travels over the network unencrypted, so use https:// for an Odoo reached from outside the company (the wizard warns about such an address); the wizard has "Test connection".

From the ERP into the assembly

KÜTÜPHANEDE ARA ▸ ERP catalogue (see section 9). A company using Logo, Netsis or Mikro points to the product list (Excel / CSV) taken from its ERP as the "Product list" in the ERP step of the wizard.

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14. Help: PUSULA and DUYURULAR

PUSULA

Type your question; questions with a definite answer are answered from the program's rules, the others with quotations from this guide and the release notes. It does not connect to the internet or to any artificial-intelligence service.

DUYURULAR

Announcements within the team. When there is a new announcement, a badge appears on the panel button.

Training videos

Short videos showing the modules in use in real SolidWorks are on the analitikasistan.com/kaynaklar page.

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15. Trial, license and renewal

The program runs as a 30-day trial with all features. The start of the trial is recorded on this computer and, if there is internet, also at analitikasistan.com (with an irreversible identifier derived from the computer). Uninstalling and reinstalling the program or setting the computer's clock back does not extend the period. A license is a signed text; it is pasted into the LICENSE window or loaded from a file, and the program checks the signature without connecting to the internet. When asking for a license, state the identifier shown in the

LICENSE window (Copy): for type 1 the computer identifier, for type 2 the Windows user name, for types 3 and 4 the company name.

A type 2, 3 or 4 license is activated once on each computer. When the license is saved, the program sends the license text (for type 2 this includes the Windows user name in it, which is not stored on the website), an irreversible identifier of the computer and the program version to analitikasistan.com; the computer name and information about your files are not sent. An activation is valid for 60 days; the program renews it by itself when there is internet. Without internet the program runs fully for 14 days: within that time connect to the internet and press "Activate now" in the LICENSE window, or send the request to the seller with "Request activation by e-mail (offline)" and paste the code you receive (it starts with AE1.) into the license box. A license activates on a limited number of computers (unless the license says otherwise, 2 for type 2 and twice the number of people for types 3 and 4); if a computer has been replaced, write to the seller to have the old one removed. A type 1 license is already bound to one computer and needs no activation. A company that never connects to the internet is given an OFFLINE LICENSE: the license file lists the IDs of the allowed computers (the "Computer ID" shown in the LİSANS window); these computers need no activation and the license is valid for its whole term, and the license line says "offline, N computers". A computer that is not on the list cannot use the license; for a new computer, send its ID to the seller and ask for a new license file. With a type 3 or 4 license limited to a number of people, users working at the same time are counted in the shared folder; if there is internet, while the program is open it also reports to the site every 10 minutes the license text, the computer ID and a random session ID made anew at each start (no person or computer name is sent; the session record expires by itself after 20 minutes), and it uses the larger of the two counts. If the limit is reached, the program opens in read-only mode; open it again when someone closes the program.

30, 7 and 1 days before the license expires, the program reminds you once at start-up and can open the renewal request form. During the trial and after it ends, "Request an extension ↗" in the LICENSE window opens the extension form. If the period ends, the program switches to read-only mode: reports and search work, modules that produce designs do not open; your files stay where they are.

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16. FAQ and troubleshooting

It says "could not connect to SolidWorks"

Open SolidWorks and wait until it has fully loaded; open the module again. If a question box is open in SolidWorks, close it first.

The program seems not to respond

If the working indicator is visible, the job is still running; wait for SolidWorks to finish. If it says it is taking long, check whether a window in SolidWorks is waiting for an answer.

The suggested code is not what I expected

Check the code structure in the "Code structure" step of the setup wizard; the customer and project numbers are read from the code of the top assembly. If the top assembly has no code, code it first.

I cannot correct a part in KATİP

The note on the row gives the reason: library parts, parts also used in other projects (shared) and protected fields are locked.

The installer cannot be downloaded

If the browser says the file is "not commonly downloaded", choose "Keep", or download the ZIP. At Windows' "unknown publisher" warning choose "More info" and "Run anyway".

The problem persists

Open the ERROR LOG on the panel; describe what happened to PUSULA, or report it with the log to iletisim@analitikasistan.com.
