Welcome to Phoenix by HP

Phoenix is a browser-based centralized application specifically developed for data scientists and engineers like you. Phoenix lets you connect to multiple data-stores across local and cloud networks, so you can access the correct data and packages, wherever they are.

Compute locally or in the cloud without interruption to manage development, data, and model environments. Dive in now to learn more about how you can use the Phoenix platform to harness your data in a more meaningful way!

System Requirements

Installing & Uninstalling Phoenix

The Phoenix team is constantly improving the installation process to help get you started as smoothly as possible. Check back here in later releases for updated installation instructions.

Before you begin the EAP installation below, remove any versions of Phoenix you currently have on your machine.

Phoenix EAP 1 Installation

Users for EAP Releases 1 & 2 should follow these steps to install Phoenix:

1. Download the Phoenix installer here (link coming soon).
2. Reboot your machine when prompted.

   Note: If you decide not to reboot your machine, Phoenix installation will pause until you manually reboot.

3. Grant the installer administrative permissions when prompted.
4. The installer will automatically check for an existing Phoenix distro and leverage it to install and configure the necessary images. If none exists, it will add one for you and proceed with the installation.

Note: The required images can take up to an hour to download.

5. To start using Phoenix, select the Phoenix desktop icon that's created when the installation is complete. Or, search for it manually in the Windows search bar.

Uninstalling Phoenix

Use the following instructions to uninstall your EAP version of Phoenix:

1. Type ***Add or remove programs*** into the Windows search bar and select it in **System Settings**.
2. Search for Phoenix in the **Apps and Features** search bar, or scroll to select it manually.
3. Select **Uninstall** to remove Phoenix from your machine.

   Note: The uninstaller does ***NOT*** remove the wsl distro and some associated Phoenix files. To remove them from your machine, manually select the files and move them to your recycle bin.

Contacting Support

Phoenix users can rely on a dedicated support team to help resolve conflicts in a pinch.

To contact support:

1. Click on the **Support** tile at the bottom of the utility bar.
2. Specify the operating system in use when you encountered the problem.
3. Provide the **Case ID** (if applicable) for your ticket.
4. Enter a brief description of the problem, what you were doing when it occurred, and how the observed behavior differs from what is expected.
5. Attach any files (if applicable), screenshots, or other documents to provide support agents with as much context as possible.
6. Click on **Submit** to reach out to support.

One of our support agents will reach out to you via email to help resolve your problem. Resolution usually only takes a few hours, but wait at least a business day for support before reaching out about the same problem again.

Navigating the Projects Page

If you're a returning Phoenix user, you can find and manage your teams' projects on a single pane of glass, directly from the Projects Page.

To find the project you're looking for:

- The **In Progress** tab displays all projects that haven't yet been marked as completed.
- The **Favorited** tab displays projects you highlight, so you can easily access the higher priority projects you monitor frequently.
- The **Archived** tab displays all the completed projects you're listed as a team member for.

Clicking the icon in the upper-right corner of a project tile triggers the project options dropdown box to appear. Each project tile gives you the option to duplicate it, save the project as a template, archive it, or to directly share it with other stakeholders.

Connecting Data to a Project

When you finish adding details, use your AWS credentials to connect to your existing projects. This page lets you connect to remote storage and create a new dataset with your data from AWS.

To connect your data:

1. Enter your Connection Name, AWS Account, and the name of the bucket(s) you want to connect.
2. Enter your IAM role ARN. Then, click on **Connect to Remote Storage** to create a new connection.
3. Name your dataset, select your connection, and set a dataset path. Then, click on **Create Dataset** to save your new dataset.

Now that your dataset is up and running, add your collaborators to get started on your new project.

Inviting Team Members

After you connect your AWS data to Phoenix, you're ready to start collaborating.

To invite more team members:

1. Enter their email address in the provided field.
2. Enter their role to delegate responsibility among collaborators.
3. Click ***Add more team members*** to increase your team's size, then repeat steps 1 and 2 until your team is complete.
4. Click on **Create Project** to get started.

During EAP, you can invite up to ten (10) teammates to collaborate on a project. You can add or remove team members at any time from the project details page. Starting with the controlled launch, the maximum number of collaborators permitted on a team will depend on your specific license agreement.

Using a Predefined Workspace

Using one the predefined Phoenix workspaces is as easy as following three simple steps:

1. Choose the workspace size that is most compatible with your machine(s).
    - **Small**: For basic libraries, small workspaces offer lightning-fast startup times, but sacrifice compute and power to do so.
    - **Medium**: The recommended setup for most users. Medium workspaces have the most balance of power and speed.
    - **Large**: For the most complex experiments, this is the most powerful workspace Phoenix can offer.

    Note: You'll get a warning if your machine can't optimally run your selection.

2. Name your new workspace.
3. Click on **Start Workspace** to start working on your project.

Creating a Custom Workspace

For projects that need more nuanced customization, you can set up your own workspace from scratch.

To choose your environment:

1. Select an image from the options in the dropdown menu.
2. Add any custom libraries you want to use in your workspace, then click **Save**.

Phoenix recommends a configuration based on the specifications outlined in previous steps, but you can always adjust them to better suit your needs.

To manually configure your environment:

1. Enter the number of cores your project requires.
2. Specify the amount of memory your project requires.
3. Toggle the GPU on / off to satisfy your project's graphical requirements.
4. Name your workspace.
5. Click on **Start Workspace** to begin working on your project.

Reusing a Workspace

To reuse a workspace:

1. Select the **Reuse a Workspace** page at the top of your screen.
2. Scroll to view previously used workspaces, or find a specific one using the search bar. You can search by workspace name, library name, or task type.
    - Click on the checkbox beside the workspace you want to select.
3. Name your workspace.
4. Click on **Start Workspace** to begin working on your project.

You can also view the memory, cores, and GPU required for each workspace to run optimally to help determine which workstation best suits your needs.

Using the Monitoring Page

Phoenix runs the tools you select natively, so you can use all your favorite DS applications directly from the application.

To start monitoring:

1. Click on **Add a Tool** to connect the monitoring tool you want to use for your project.
    - Phoenix currently only supports ML Flow and Tensorboard, but will support more DS applications in the coming releases.
        - list the supported applications users can refer to here

2. Click the icon in the upper-right corner of the DS tool panel to enter fullscreen mode. Or, simply use the application as you normally would from the default panel if you.

The **Monitor** panel gives you real-time access to your project's GPU, CPU, and Memory consumption, and Phoenix AI recommends configuration improvements based on your usage trends.

Using the Notebooks Page

Each tab on the **Notebooks** page represents a different notebook and you can have up to __ running simultaneously.

To create a new service:

1. Click on **Create a Service**.
2. Select a model from the dropdown.
3. Name the service.
4. Click on ***Browse a local file*** in the Bento ML panel to find a container to run your service in.
5. Click on **Create Service** to save your settings and create a new service.

Your project's GPU, CPU, and memory consumption appear in the corner of the screen, so you can visualize the effects of the tests you run in real-time. When you create a new service, it automatically becomes viewable from the **Published Services** tab.

Data Fabric

In Phoenix data directories are mounted in the container so users can use the data to train machine learning models and run other high-velocity tests.

To create a new dataset:

1. Navigate to the Data Fabric page, then click on **Add new dataset**.
2. Enter a name, description, and the storage type for your dataset.
3. Specify your provider and complete the corresponding fields that populate. Currently, Phoenix only supports AWS S3.
4. Specify the download and upload settings that apply to your dataset.
5. When you're finished, click Save to create the new dataset.

From the Datasets home page, you can use the upload feature to copy local datasets into a cloud provider. The home page also lets you upload changes to a local dataset you previously downloaded.

Note: Datasets are also viewable and can be managed from the Notebooks tab on your project's home page.

Team Settings

You can view and search for current members of your team under the **My team** tab. Pending invitations appear under the **Pending** tab until invitees accept or decline the invitation sent to their email address.

You can also use the **Pending** tab to view the date an invite was sent, or you can resend an invitation to pending invitees by clicking on Remind. Uninvited users who request access to your team will appear under the **Requests** tab.

To manually add users to an existing team:

1. From the Team Settings page in the **Account** tile, click on **Invite team member**.
2. Enter their email address and choose their role from the dropdown.
3. Click on **Add more team members** and repeat step 2 for each user you want to add.
4. When you're finished adding members, click on **Invite** to add them to your team.

Phoenix sends each new member an invitation to join the team via email. You can remove users from your team or edit individual user policies at any time by clicking on the more options icon [icon screenshot].

Billing

The billing tab lets admin users view and manage their Phoenix subscription information, view the status and date of past invoices, and see how many remaining seats are included in their license.

Manage team members

Manage subscription

Generate a PO

Speak with an expert

Company Details

The company details page is functionally the admin home page, where admin users can view and change the company name, billing email, and company logo.

Admins can also use the company details page to delete the Phoenix account associated with their team and related projects. After deleting your account, you will lose access to your projects, team information, and other associated data.